Modifying the W6PQL LNA PCB for 144 through 902 MHz

Gary R. Greene, W2ZV ©2011
w2zv@w2zv.com

Introduction

I purchased a few of Jim Klitzing's (W6PQL) LNA (Low Noise Amplifier) boards from his web site (Ref 1) and started to model the design layout in preparation for using a different device. His schematic is the same outline as the Avago Application Note 1299 for the ATF-54143, and his board lends itself well to modification. By replacing the ATF-54143 with the Mini-Circuits SAV-541+ device the noise figure was improved by 0.2 dB at 1296 using Jim's parts, although it may not be unconditionally stable with that part substitution. I thought that by using a higher Q inductor in place of the surface mount inductor, I could improve the noise figure even more. Both the ATF-54143 and the SAV-541+ only have published Gamma Optimum down to 500 MHz by the manufacturer, although there are several designs for 432 MHz and 144 MHz by Ole Nykjær, OZ2OZ (Ref 2), Peter Hoefsloot, PA3BIY (Ref 3), Zdenek Samek, OK1DFC (Ref 4) and others for the ATF-54143. As far as my research has gone, there is no other published design on the internet or available literature for a SAV-541+ device at those frequencies, beyond my article published in the SVHF Society conference proceeding last April (Ref 5). My goal is threefold: using a widely used PCB, to add my best effort for an easy to build, repeatable design using the SAV-541+ device at 144 MHz through 902 MHz to the literature, second, to design and build an unconditionally stable Low Noise Amplifier, that can get a very good noise figure (best case under 0.3 dB below 902MHz) and third, by using off board input and output filters, to obtain good overload resistance with high IP3 for terrestrial use, an improvement over the filtering of my first design, one of the "Further Research" goals of my first paper.

Design Approach

My goal was to use the SAV-541+ device, which has a maximum input rating of better than +16dBm. The configuration for best noise figure consistent with a high IP3, based on the data sheet, means running the device at 40 mA of current and a Vds of about 3.0 V, unlike the Avago ATF54143, which likes 60mA for better noise figure. At this current level, the IP3 (Output) is about +30 dBm with no difference between a Vds of between 3 V and 4 V at 900 MHz, although it has a slightly higher IP3 at 2 GHz. An estimate of +1 dBm IP3 at the input (+26 dBm IP3 at the output) or better at 2 Meters still makes the LNA as good, or better, than many receivers and transverters in use today. For in band intermodulation distortion, your receiver must handle greater than +20 dBm IP3 at the input to make full use if its range. Note that the 902MHz LNA in this paper, at 60mA, will have about +10 dBm IP3 (Input).

I have detailed the design steps in my first paper (Ref 5), so I will not bore you here. Software tools included Amateur Radio Designer (ARD) and RFSim99 which available on the internet (Ref 6). The program RFSim99 is now free (but not open source) which works under XP or XP simulation under Windows 7, just like ARD, which is not available any more from the ARRL. RFSim99 is a nice program for linear simulation like ARD, and has an easy-to-use input editor, instead of the spice-style text entry of ARD. It only takes a few minutes to get use to the "right click in the *exact* middle of the part" to get the edit feature to work. It does not have an optimization feature except for an input/output Auto Match that adds a LC network on each port, but it does support transmission lines. Using the export of microstrip S-Parameters from RFSim99 and placing them in the input file for ARD allows for support of microstrip circuits in

ARD, but without optimization of the microstrip. Since I am using a fixed PCB design, this fits my needs just fine.

One of the lessons learned after my first article was that for 432 MHz and below, the difference between the RO4003 (low tan-delta) board and the FR4 board to obtain the low noise figure of this design is not significant. Jim had given me a FR4 PCB to try, and the results did not come out as well the first time, but after making the source ground points closer to the device, everything cleared up very nicely. The final design allows for this change.

Design Details

The schematic for 144 and 222 MHz is shown In Figure 1 below:

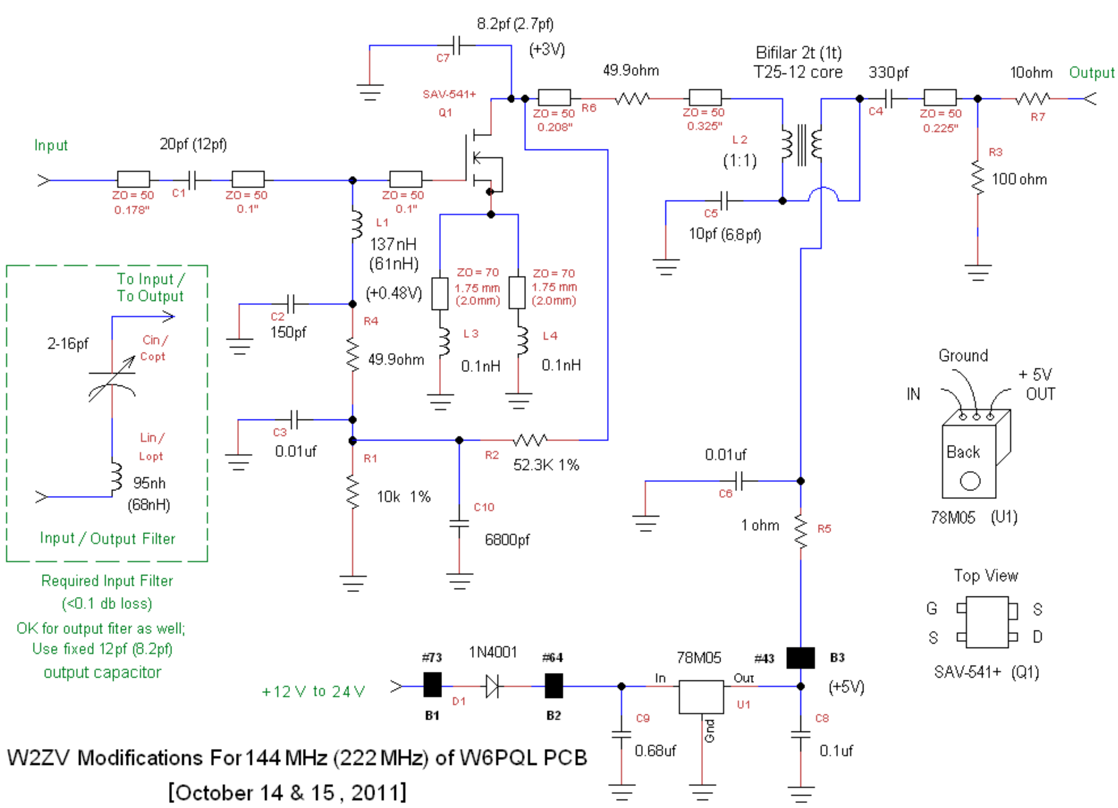

Figure 1: W6PQL PCB LNA as modified for 144 MHz and 222 MHz

For this design, the input filter is not optional on 144 through 432 MHz, only the output filter is. To call it an input filter is a bit of a stretch, as it is more of an input matching circuit, its filter abilities are marginal, and the matching becomes more of a liability on 902MHz, and so is not used there. This makes the output filter more important for 902MHz, but is easy to implement at that frequency. The design for 144 MHz and 222 MHz uses a toroid for both output matching and some selectivity, different from my first design. The Ah-ha moment came when I realized that the same style filter for input and output would work in the same box I use, without shielding, in part due to the gain of less than 30 dB and the low Q of the filters.

The schematic for 432 and 902 MHz is shown In Figure 2 below:

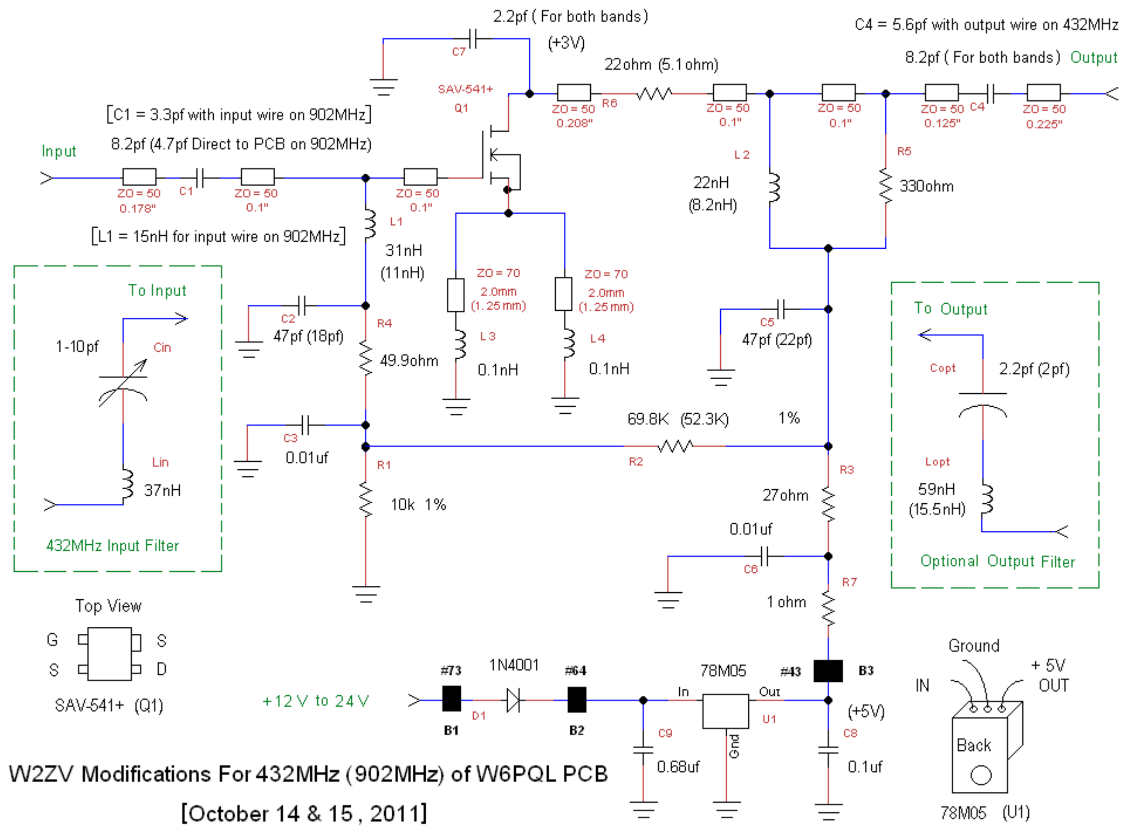

Figure 2: W6PQL PCB LNA as modified for 432 MHz and 902 MHz

The 432 MHz design remains almost unchanged except for the output filter, as it seems to be the best to date. I have tried several changes without success. On 902 MHz I tried the application note circuit at 60mA that got great input return loss, good output return loss, and a measured noise figure of 0.66dB, while my modifications ended with a good return loss (-14dB vs. the application note’s -22dB), the same output return loss (of -17dB) and a noise figure of 0.43dB. The gain of this LNA is down to about -4dB at 432MHz, for a reasonably good band pass with very good IP3. I tried to simply modify the 432 MHz design at 40mA for good input match at 902MHz and got a noise figure of 0.38dB, but it had the same gain at 432MHz with a noise figure of 0.40dB, so it is wide as a barn door, and not a good choice for terrestrial work in the same neighborhood as other hams. I believe 432 MHz is the sweet spot for this device, so as the circuit detunes below 902, the inherent device improvement makes up for the detuning, and so the noise figure and gain stays essentially flat from 425 MHz to 925 MHz. This is a good choice for a UHF scanner LNA but not for a single band design. If you wish to build the 222 MHz circuit using my previous paper's layout (see Ref 5), replace the output components starting with the 22 ohm resistor from the old 144 MHz design and it will work well, the output values work for both bands due to the broad nature of the old design. Again, see my previous paper for a blow by blow description of the circuit.

Implementation

In an effort to assure my goal of a repeatable design, let me go into some detail in the process of building these LNA's.

I use a large cookie tray with a small piece of plywood in the middle to build the PCB. My tools include digital calipers, good wire strippers, good tweezers and the usual hand tools, the AADE inductance and capacitance checker, a VOM, a 12 V supply, and a Noise Figure Meter and cables. If you do not have access to a noise figure meter, then you need some way to measure the gain, as most problems show up as low gain. Note that peaking the input filter for the most gain will not usually lead to the best noise figure. The next choice up the chain is peaking on a weak, stable beacon, so long as you use an input attenuator to make it just barely above the noise. Having an antenna relay cycle between a 50 ohm load and the weak signal would be the best approximation to a noise meter. Use a scope on the output audio and tune for the max ratio (not difference) between no signal and signal. Then take it to a VHF conference to check it.

Before you mount components to the PCB, first use a sharp blade to cut the notch out on the trace leading away from the drain, as R6 will be placed there. Use the tip to peel the notch away and check for an open with an ohm meter.

Next, measure 1.75mm for 144 MHz, 2mm for 222, 2mm for 432 MHz, or 1.25mm for 902 MHz from the device leads far edge on the source lines and solder a tab of copper or broad device lead clipping on each source line to the ground next to it. I use copper foil from mouser or the hobby craft shop if you can find some there. Use silver bearing solder, which needs a slightly hotter iron, if you have any. Note that moving the shorting strips will change the gain more than the noise figure, which changes very slowly, and changing the location during tune up becomes problematic in this design, as we reuse the grounds and the board in the box can become unstable if the short is out too far when it is in the box. I get pretty good repeatability with the location values I have shown.

The circuit board was constructed as per usual for an SMD style board. I use 0603 resistors for R5 and R6 for the 432/902 MHz design, as the pads are small and I like to see part of the second pad to solder to after placing the part on the first pad. For the same reason, I use a type 0805 resistor for R3, while the three remaining normal resistors are OK with 1206 parts, although R4 is currently 0805 size, so I can use the same part for R6 on the 144/222 MHz design. Note that for the 432/902 MHz design I placed the output SMD L2 near the device and the swamping resistor R5 near the output C4, the same as Jim's design, which is different from my previous work. In an effort to keep the pads from pulling up with the coil on 432 MHz and below, I use the FR4 PCB and put the coil on last. For the 144 MHz design I also moved the bypass side of the coil behind the device to give the coil more room from end to end, since L2 is now a toroid and its leads can go almost anywhere.

I have had issues with a short below the L2 inductor (for 432/902MHz), which will show up as low gain at about 10 dB vs. the 20+ dB normal gain. Visually check when mounting this part.

Among the last two surface mount parts to be mounted are the device Q1 and the capacitor C7, which is mounted between the ground next to the source and the drain land at the bend. Note I replaced the diode D1 with a 1 ohm resistor, named R5 for the 144/222 MHz design, and named R7 for the 432/902 MHz design. Please see Figure 3, a view of the 432 MHz board without the input coil in place yet, along with the regulator with the capacitors installed, as shown below:

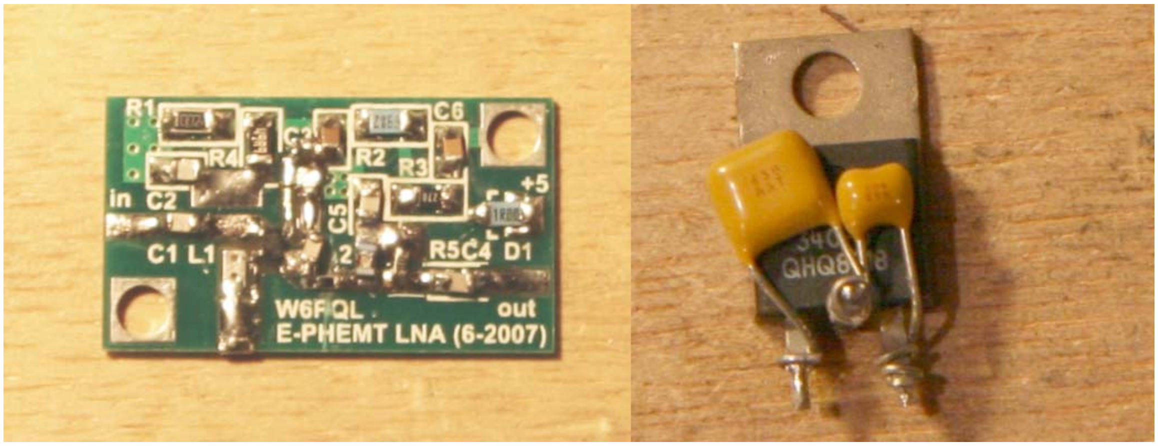

Figure 3: View of the 432 MHz board without the input coil in place yet, along with the regulator

Using a toroid for the 144/222 MHz design leads to an interesting layout. This assumes an FR4 board is used. The RF hot lead of the toroid comes from the trace going to R6 (the resistor installed at the cut in the land), and heads up towards the toroid hanging above the board. The center tap of the toroid, the output capacitor, and the "tuning" capacitor can meet at a point in space above the board, where the output capacitor gets connected to the output trace, the "tuning" capacitor wraps over the edge to ground, and the 100 ohm round resistor goes from the output trace, runs along the edge, and the other end wraps over the board to ground. Alternately, you can wrap a piece of copper foil over the edge of the board to sit next to the output trace. Solder it to the back of the board, place a 1206 or 0805 100 ohm resistor from the output trace to the new ground foil, and tack the "tuning" capacitor lead to the new ground foil. This assumes you have a supply of copper tape to use for this purpose, I leave the choice up to you. The power for the toroid comes from the pad for R3 that is bypassed by capacitor C6, tack solder that toroid lead there. For the 144 MHz design only, with this connection point up in space, we can now use the land bypassed by capacitor C5 to function as the cold end of L1. So we put C2 in the C5 position, place R4 in the R2 position, leave the old R4 and C2 positions free, and run L1 from the gate trace to the land bypassed by the C2 in the C5 position. Note that the 222 MHz unit has a small coil and so this is not needed here, so we put L1 in the traditional place, as used on the 432/902 MHz design, and can make use of this free land to tie the center tap of the toroid to, if you wish. Then the 6.8 pF capacitor C5 goes in the place marked C5 and we jumper the leaded capacitor C4 from that land to the output trace.

For both the 144 MHz and 222 MHz designs the DC feedback resistor R2 goes from the drain trace out parallel to the source trace, and gets bypassed with a capacitor C10 from the shorting tab over to the floating end of the resistor R2 that sits on the bare board next to the shorting tab. Install the resistor R2 to the drain trace after installing C7 first, butting head to head, and then align the capacitor C10 to the ground at other end of R2. Now cut 38mm (1.5") of wire wrap wire or #22 wire, strip each end, solder one end to the junction of R2 and C10, run the wire under the board, and tack solder the other end to the old R4 pad adjacent to resistor R1 and capacitor C3 for 144 MHz, or to the old R2 pad adjacent to capacitor C3 for 222 MHz. This leaves the big land next to the gate unconnected for the 144 MHz design. Shorting this to the gate trace will increase the gate capacitance, which is not a good thing, but is OK on 6 meters, which I will leave for another paper.

Figure 4 below shows the layout with the toroid but before the coil L1 is installed. Note that for 222MHz the toroid has just one turn (just stick the wire pair through and bring alternate colors together, one from each end, to form the center tap) while for 144 MHz the toroid has 2 turns (stick the wire pair through, wrap around and stick the wire through again). For the wire pair I use two colors of #30 enamel coated wire, or two colors of wire wrap wire in a pinch, and twist 8 to 10 turns per inch before you start. Two inches (50mm) of the wire pair is more than enough for 222 MHz, while add another ½ to 1 inch (75mm total) for 144 MHz gives you the extra length needed while building the transformer.

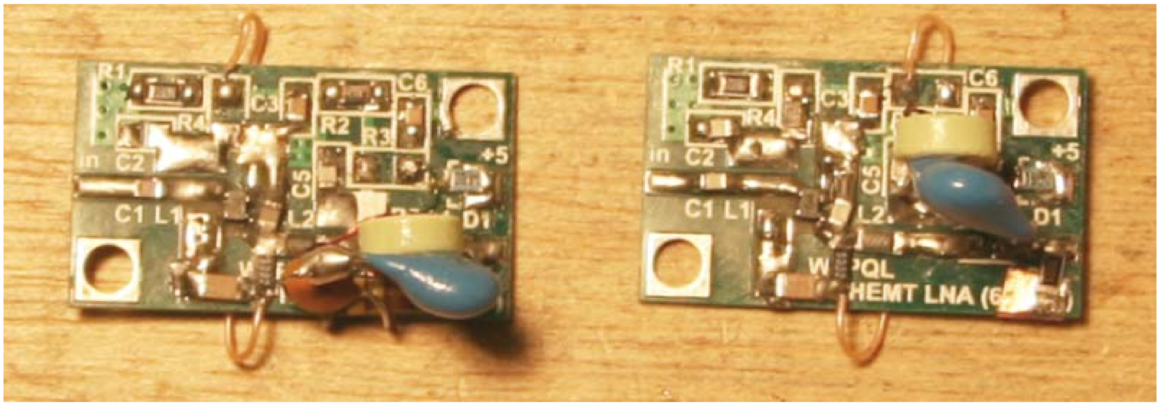

Figure 4: View of the 144 MHz board and the 222 MHz board without the input coil in place yet

In an effort to make the input coil very repeatable, start with 145mm of #20 coated wire for 144 MHz, 90mm for 222 MHz or 65mm for 432 MHz, and clean off the last ¼ inch (6mm) on each side by scraping with a blade against the plywood all the way around. For 144 MHz, wrap 3 turns around a 3/8” dowel or drill bit, evenly spaced, with the two leads tangent to the rod or drill and keep them the same length, and then go around another quarter each, for a total of one half turn, and leave the leads tangent to the coil form (do not bend out). Place a small “J” hook on each end pointing towards each other. The hook to hook distance should just fit the distance from the middle of the gate trace to the center of the bypassed land behind the device, about 10mm across, which makes the coil 7mm high. For 222 MHz, wrap 3 turns around a 7/32” dowel or drill bit, close space, with the two leads tangent to the rod or drill and keep them the same length. Bend each lead up and keep them aligned and place a small “J” hook on each end. When you remove it from the form it will spring apart slightly, and this is what I want to have happen. For 432 MHz wrap 2 turns around a 7/32” drill bit and follow as above for 222 MHz. For all three coils, the total height from the "J" hook to the top of the coil is 14 mm, which leaves the top of the coil 4 mm below the cover. I tried winding the input coil using #18 wire, but it was stiff enough to pull the PCB lands when stretching or squeezing the coil, while #22 wire deformed too much during the squeezing process. For 222 MHz and 432 MHz I usually place one lead on the gate trace next to capacitor C1, and the other lead on top of the capacitor C2 lead over the pad, so that the leads for the coil can be farther apart than the pads for L1 alone allows. Pictures showing the coils during and after the process are shown in Figure 5 below:

Figure 5: L1 coil winding pictures

Winding the input filter coil is different. You want it plenty stiff, as the capacitor hangs off the end, and the length is not so critical, since the capacitor is adjustable. Here, however, using #16 silver plated wire helps the Q. For 144/222 MHz use 145mm of wire and wrap 3 turns (2 turns for 222 MHz) over 0.35” total length on a 3/8” dowel or drill bit, and for 432 MHz use 130mm of wire and wrap 1 and 1/2 turns over 0.4” total length on the 3/8” dowel or drill bit and leaving plenty of wire coming off each end. One end is wrapped around the end of the variable capacitor, and the other end is moved to the center and down to wrap around the center pin of the input connector. To fit the box, wind the coil clockwise from above, starting from the top and going down, as shown below. Pictures showing 432 MHz Input coil with the variable capacitor and the 432 MHz output filter LC, along with the 222 MHz output filter LC are shown for in Figure 6 below:

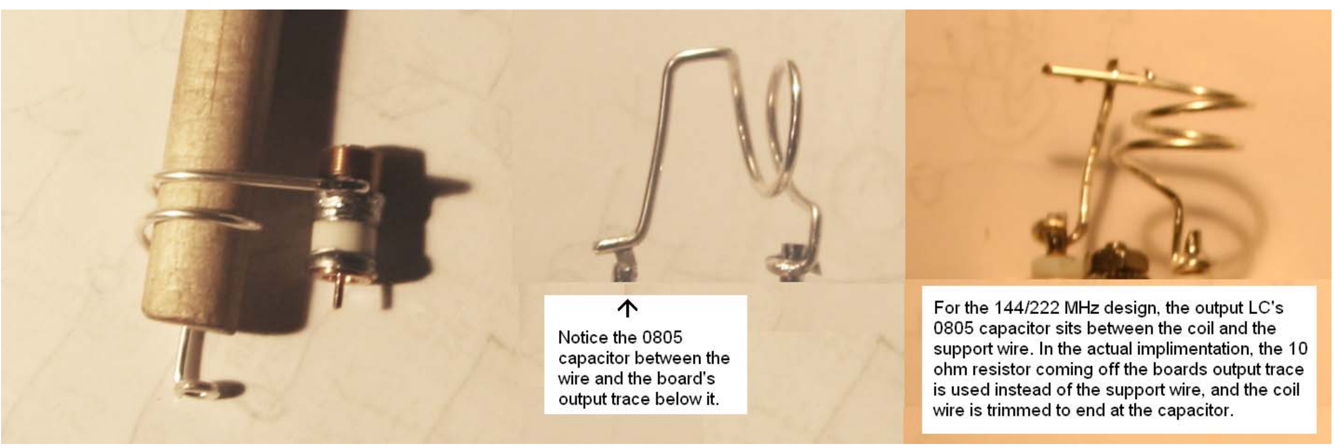

Figure 6: Finished filter coil pictures

For the 902 MHz output coil, take about 30mm of #16 wire and make one full turn the full distant from the center pin of the output connector to the output trace of the board on a 3/16" drill bit, about 9.5mm (3/8") edge to edge, and leave less than 2.5mm (1/10") of lead length at each end. Tin the center pin, and solder the 0805 capacitor of 2.0 pF to the top of the pin. While holding the coil on the output trace using pliers, place the other end over the capacitor, tack solder it in place and hold the coil very still till the solder is solid. If the other end is not touching the output trace DON'T bent the wire but instead, while holding the coil with pliers, touch up the solder joint at the capacitor so the other end is resting on the output trace again. When you are sure the coil is in place, you can then tack solder the output trace end. You can go back and forth to touch up your work, so long as you wait for the solder to become solid before going to the other joint. You can see the output coil in the left unit in Figure 8 below. Note that the 902 MHz L1 coil is a SMD device with a Q between 80 and 90, and has much better repeatability than I can obtain.

I selected the box based on low cost, availability, and robustness. This box comes with or without mounting ears on the cover, which works out well, since the LNA is mounted on the bottom of the box, and the mounting ears can be used to secure the case without obstructing the connectors. I selected N connectors as the standard, and used two of the mounting holes to mount the board, using stainless 4-40 screws, and threading the holes in the box using a 4-40 tap (or M3 for metric hardware). Leaving the connectors mounted square leaves the board in an awkward position (it just does not fit well), so I canted the connectors so the board fits well inside. I tried to mount the connectors in a diamond shape, and again this left the board in a somewhat awkward position, but at least it fit. By awkward, I mean that the output connection is not near enough to the pin of the output connector. The final physical layout is shown in figure 7 below, all the hole sizes are dependent on the hardware used (metric or English, type of feed through, and type of connector):

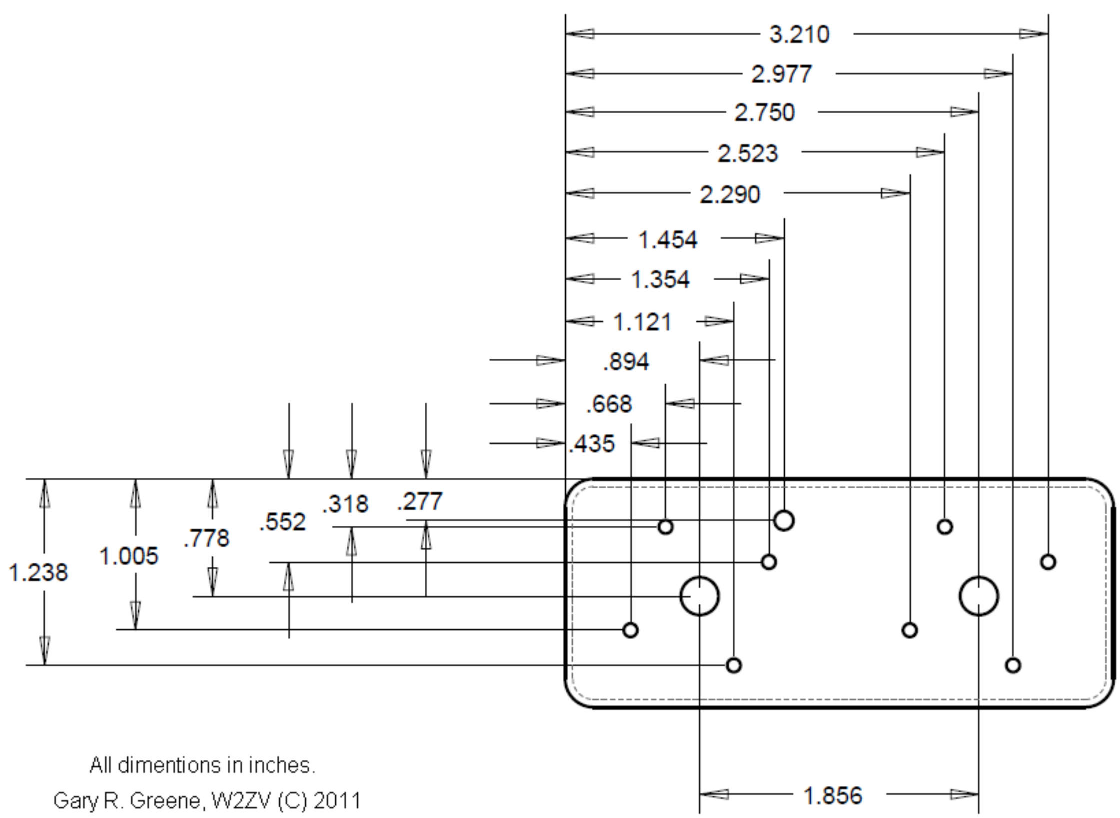

Figure 7: Final layout of the box

The input connector has the input coil for 144 MHz through 432 MHz, or has a short #16 jumper wire to the input trace on the 902 MHz design, else the input trace goes to a wire from the bottom of the variable capacitor. I use a stainless steel split C lock washer on each mounting screw as spacers below the PCB to raise it up above the rough case metal box floor, and use ground lugs as washers above them. This keeps the small size hex nuts tight without the lock washers digging into the PCB, as there is little room for washers on top of the board on one side, although you can use a second set of split C lock washers on the top side a well, but be careful to not destroy capacitor C6. Since the supports are one of the mounting ears of the connector, no additional ground lug is required, but you can add a second ground point if you desire. In that case dress the lead directly to the bottom of the board near the signal connection point, and use heavy wire or braid for maximum effect. The power feed through capacitor goes next to the board near the input connector on the mounting screw side. The regulator is mounted on the wall on the same side as the feed through at the other end of the box. I start installing the regulator by bending the pins up, installing the two bypass capacitors from center pin to each end, clip the excess lead length (see Figure 3 above), and then mount it in the case after using the part as a template for the single mounting hole. I use a truss head stainless 4-40 screw and a nut with a captive lock washer, as you are limited by which size nuts you can use if you use a 6-32 screw. Place the #73 bead over the feed through and put a "J" hook on the un-banded end of the diode and solder the diode to the feed through, holding the bead in place. Place the #64 bead on the banded end of the diode, hold it in place with a "Z" bend, and wrap the end of the diode wire over the first pin of the regulator, left (feed through) to right (regulator) just as it is drawn on the schematic. Use heat shrink tubing as tubing to protect the diode leads if you think the leads may touch the case. Use a 80mm (3") piece of #22 wire to connect the output of the regulator (after it has been tested for 5 Volts output) and route it to the board, placing the final bead on the wire end just before the PCB connection. A picture of the completed board in the case, with and without the input filter, is show in Figure 8 below:

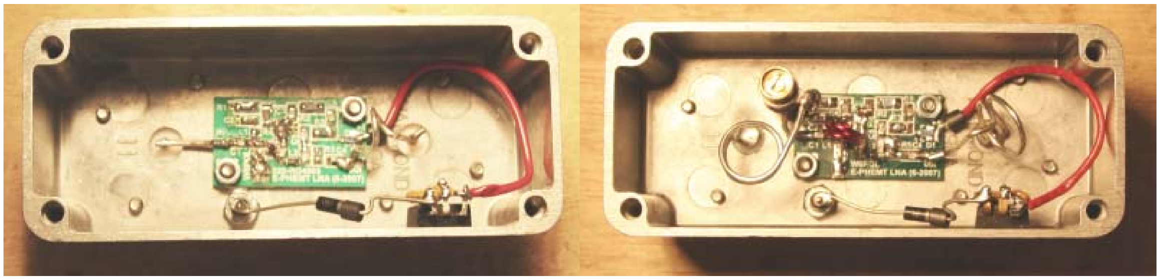

Figure 8: Finished 902 MHz and 432 MHz boards in case with the power connected.

The last touch is to place two pieces of absorber ½” by 1” each on each end of the cover. See the detail in Figure 9, as shown below:

Figure 9: Finished box and cover inside with absorber applied.

A list of parts and their part number from Mouser Electronics (Ref 7) is show in figure 10 below:

Case, Wire and Input/Output Filter parts

Quantity	(Highest $ items first)	Mouser # / Other source
1x1 Inch	Absorber ***	517-AB2100
6 Inch	#20 Wire ***	566-8050
144/222	Piston Cap (2-16 pF)	659-GAA16001
432 MHz:	Piston Cap (1-10 pF)	659-GAA10002
6 Inch	#16 Silver coated wire ***	Craft or Jewerly supply
0	Hammon box with ears	546-1590AFL
1	Hammon box plain	546-1590A
1	Feed through	800-4400-037LF
2	N-Conn	523-82-6101-RFX
9	4-40 (or M3) screws	Local Hardware
4	Split-C lock washer	Local Hardware
3	#4 Solder Terminal Lugs	534-7328
4	4-40 nuts (small size)	534-4694
	Input/Output Filter Coils:	
1	Copt - 12 pF 0805	581-08051A120KAT2A
	[Copt - 8.2 pF 0805]	581-08052U8R2CAT2A
	[Copt - 2.2 pF 0805]	140-CC501N2.2C-RC
	[Copt - 2.0 pF 0805]	581-08051U2R0BAT2A
144 MHz:	145mm #16 3 T 0.35" tall on 3/8" ID form (95 nH)	
222 MHz:	130mm #16 2 T 0.35" tall on 3/8" ID form (68 nH)	
432 MHz:	110mm #16 1.5 T 0.4" tall on 3/8" ID form (37 nH / 59 nH)	
902 MHz:	40mm #16 1 T 0.375" tall on 3/16" ID form (15.5 nH)	
****	High Price due to minimum quantity order amount.	

Power Supply Parts

Quantity	Item Detail	Mouser # (Ref 7)
1	U1 - 78M05 (+125 to -40C)	863-MC78M05BTG
1	C9- 0.68 uF 25V	810-FK14X7R1E684K
1	B1 - Bead #73	542-FB73-110-RC
1	B2 - Bead #64	542-FB64-110-RC
1	B3 - Bead #43	542-FB43-110-RC
1	C8 - 0.1 uF	594-K104K15X7RF5TL2
1	D1 - 1N4001	625-1N4001-E3/73

Printer Circuit Board Parts		
Quantity	**Item Detail [Other Bands]**	**Mouser # / Other source**
1	PCB	W6PQL (Ref 1)
1	Q1 - SAV-541+	Mini Circuits (Ref 8)
1	C1 - 20 pF 0805	581-08052U200GAT2A
	[C1 - 12 pF 0805]	581-08051A120KAT2A
	[C1 - 8.2 pF 0805]	581-08052U8R2CAT2A
	[C1 - 4.7 pF 0805]	581-08052U4R7C
	[C1 - 3.3 pF 0805]	581-08052U3R3CAT2A
1	C2 - 150 pf 0805	80-C0805C151J5G
2	[C2, C5 - 47 pf 0805]	80-C0805C470J5G
	[C2 - 18 pf 0805]	80-C0805C180J5G
2	C3, C6 - 0.01 uF 0805	80-C0805C103K1R7210
1	C4 - 330 pF Leaded 3KV	810-CK45-B3FD331KYNN
	[C4 - 8.2 pF 0805]	581-08052U8R2CAT2A
	[C4 - 5.6 pF 0805]	81-GRM40C5R6C50D
1	C5 - 10 pF Leaded	581-SR212A100KARTR1
	[C5 - 6.8 pF Leaded]	80-C317C689D5G
	[C5 - 6.8 pF 0805]	581-08051A6R8CAT2A
	[C5 - 22 pF 0805]	581-08051A220J
1	C7 - 8.2 pF 0805	581-08052U8R2CAT2A
	[C7 - 2.7 pF 0805]	81-GRM40C2R7C50D
	[C7 - 2.2 pF 0805]	140-CC501N2.2C-RC
1	C10 - 6800 pF 1206	581-12062C682KAT2A
1	R1 - 10k 1% 1206	290-10K-RC
1	R2 - 52.3k 1% 1206	290-52.3K-RC
	[R2 - 69.8k 1% 1206]	290-69.8K-RC
1	R3 - 100 ohm 1206	652-CR1206FX-1000ELF
	R3 - 100 ohm Leaded	71-CCF60-100-E3
	R3 - 27 ohm 0805	71-CRCW080527R0JNEA
1	R4 - 49.9 ohm 1206	652-CR1206FX-49R9ELF
1	R5 - 1.0 ohm 5% 1206	660-RK73B2BTTE1R0J
	[R5 - 330 ohm 0603]	652-CR0603-JW-331GLF
1	R6 - 49.9 ohm 1% 0805	71-CRCW080549R9FKTB
	[R6 - 22 ohm 0603]	667-ERJ-3EKF22R0V
1	R7 - 10 ohm Leaded	660-MF1/4D52R10R0F
	[R7 - 1.0 ohm 5% 1206]	660-RK73B2BTTE1R0J
1	L1 - 137 nH on a 3/8" ID	145mm #20 3 T 7mm tall
	[L1 - 61 nH on a 7/32" ID]	90mm #20 3 T 3.5mm tall]
	[L1 - 31 nH on a 7/32" ID]	65mm #20 2 T 2.5mm tall]
	[L1 - 11 nH 0603]	81-LQW18AN11NG00D
	[L1 - 15 nH 0805]	81-LQW2BAS15NJ00L
1	L2 - T25-12 core 2t [1t] (bifi)	Amidon (Ref 9)
	[L2 - 22 nH 0805]	70-IMC0805ER22NJ01
	[L2 - 8.2 nH 0805]	81-LQW2BAS8N2J00L

Figure 10: List of parts and part numbers

Some of these parts are not cheap, eBay and your junk box can be your friend. I have selected microwave quality components only when it will make a difference. Please try to pick connectors with Teflon centers and silver or gold contacts if possible, at least for the input connector.

The SAV-541+ is available from Mini Circuits (Ref 8) and has a high price for single unit quantity. Note that their price for 20 is three times their price for a single piece. You can use an ATF54143 in a pinch, the Mouser # is 630-ATF-54143-TR1G. The circuit should be unconditionally stable, but the noise figure will be whatever it will be. The ATF-58143 may work here as well.

Testing Data

Testing consists of checking the DC voltages, and then checking gain and noise figure with the cover off, and retest with the cover on. The critical voltage at the drain should be about 3.0V and both sides of the 49.9 ohm resistor R4 should read the close, at about 0.48V if the board is not oscillating. Voltages on either side of R4 will have a low (close to one half or less) voltage if it is oscillating. I have found the board to DC check OK out of the box, but fail the DC check in the box. The fix has always been to make the shorting straps closer to the device. Check to be sure the solder connection is secure first. Note that if the cover gives different gain and noise figure with it installed or not, you can try and make the shorting straps closer to the device. It does not mean the unit is unstable necessarily, just that the box resonances are not conducive to this exact layout.

With DC checks OK, check the gain. You should get near 20 db or more if everything is OK (a bit less on 222 MHz and 902 MHz), else check for bad connections or bad input/output or bypass caps. Remember the possible issue with a shorted L2 I mentioned before. In that case the gain was low, but still about 10 dB. If the gain is OK, adjust the input filter capacitor in quarter turn increments until you find the lowest noise figure, then adjust in very small increments until you believe you have reached the minimum, stretch or squeeze the input coil and try again. Be careful not to pull a land or damage the parts below the coil while modifying the coil's shape. I have learned to test with the cover on (just pressed together), open to adjust the trimmer, cover and reset the noise meter to test again. Tuning for minimum, then putting the cover on might lead to different to results, so you may end up doing the open and close routine in the end any way. The case selected is great for this, squeezing it closed is every bit as good as screwing down the cover every time.

Gain and Noise Figure and Stability chart, and Band pass data are show in Figure 11 - 15 below:

Band	NF - Best	NF - Avg	K min	Gain	P1dB (Out - dBm)
144 MHz	0.28	0.313	1.35	23.19	4
222 MHz	0.28	0.315	1.86	17.75	1
432 MHz	0.22	0.242	1.27	23.55	9.5
902 MHz	0.43	N/A	1.22	20.3	10.2

Figure 11: Noise Figure, K, Gain and Power (1dB Compression) chart

Testing for IP3 using the one dB compression point is not accurate in this configuration. The toroid used on the 144/222 MHz design and the output attenuation used here play havoc with the numbers. Estimated results are about +1dB input IP3 at 144 MHz and +3dB input IP3 at 432 MHz when extrapolating from manufacture supplied data of +5dB input at 900 MHz, based on 40mA current. The equipment to provide an accurate test will not be available until after this paper is published, and is the subject of further research into the proper high IP3 output circuit for this design.

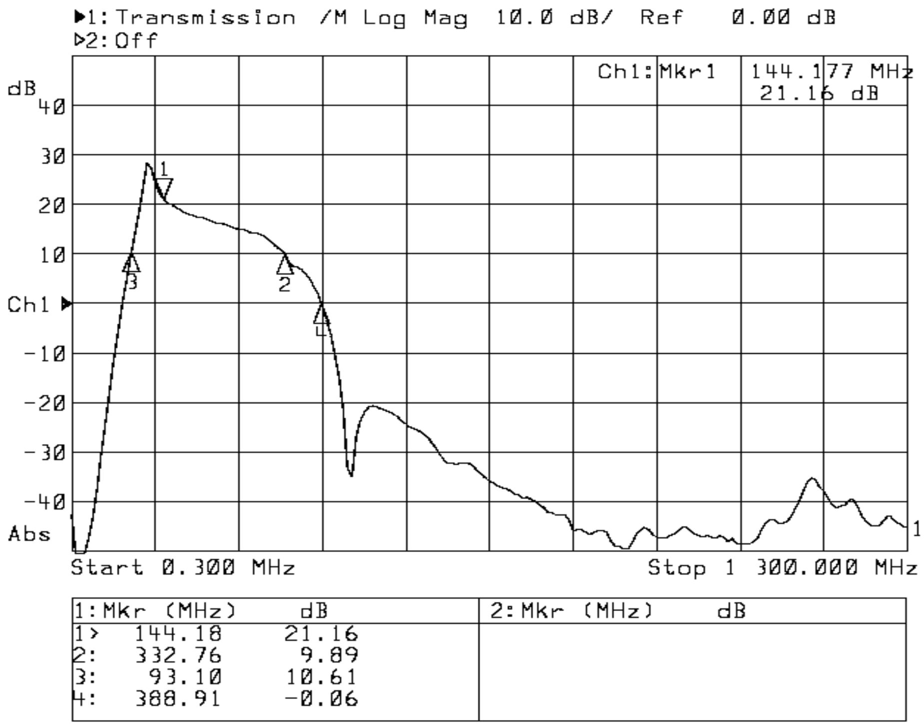

Figure 12: Band pass response with input/output filters (144 MHz)

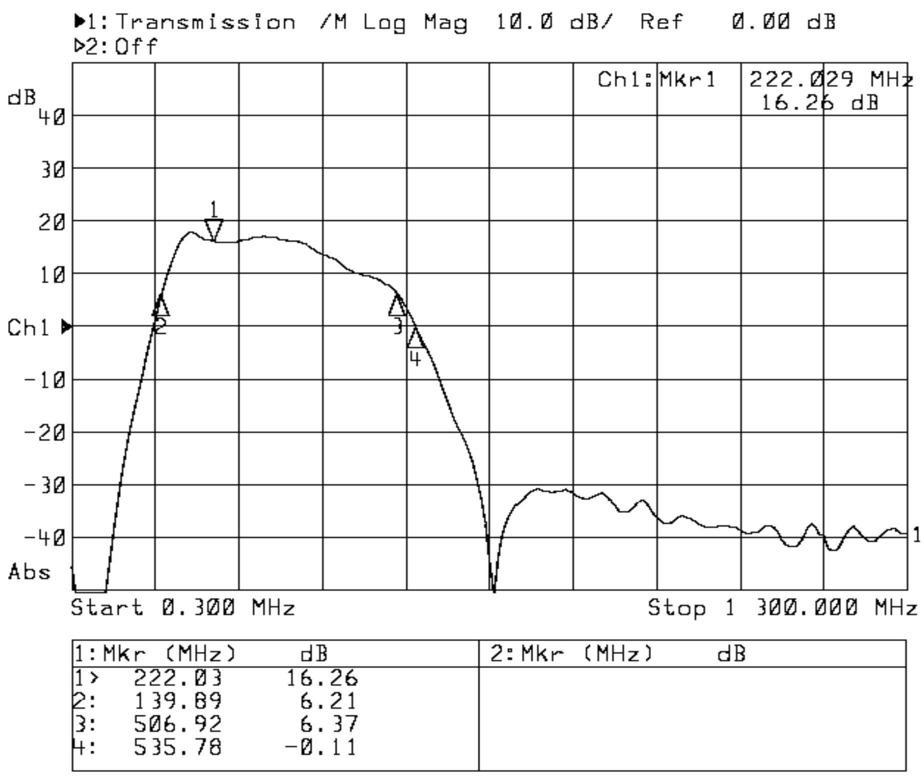

Figure 13: Band pass response with input/output filters (222 MHz)

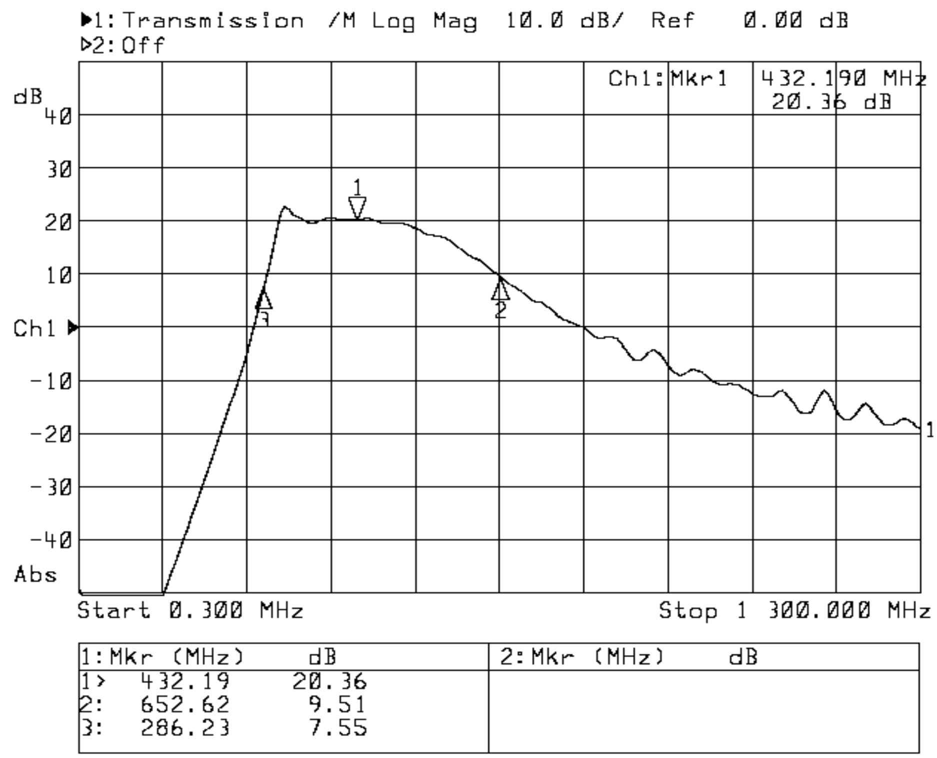

Figure 14: Band pass response with input/output filters (432 MHz)

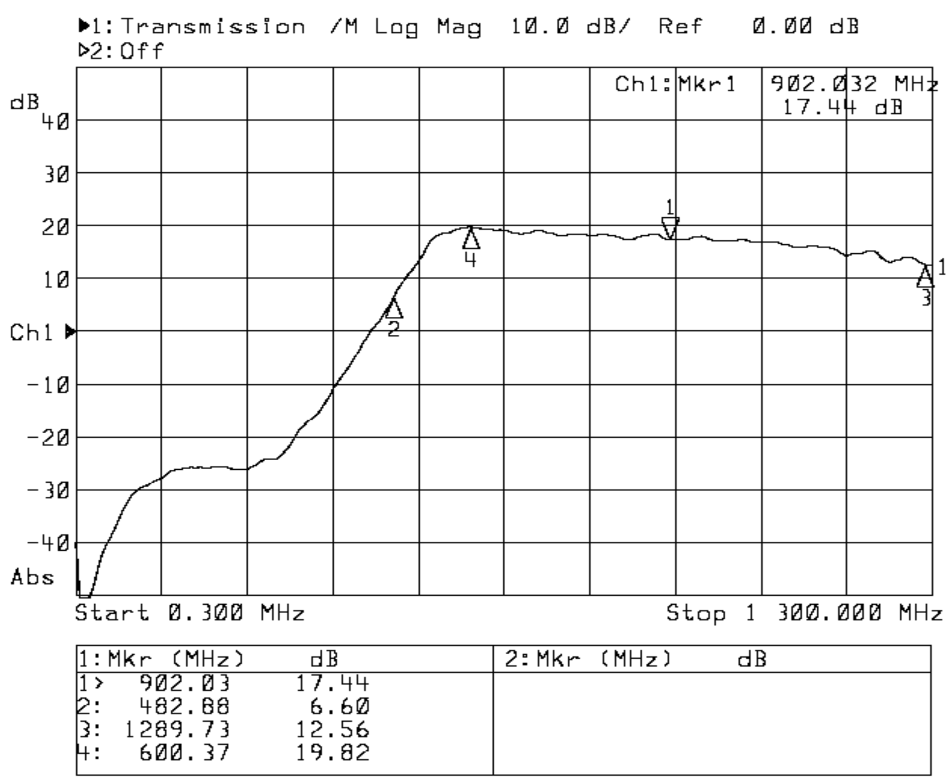

Figure 15: Band pass response with the output filter alone (902 MHz)

Conclusion and Further Research

While any step is not difficult for anyone who has had experience with surface mount construction, it does have many steps, and takes more labor than a simpler design.

Adding a relay that would short the input when the power is removed would be a simple addition. The G6K-2P-DC5 (Mouser # 653- G6K-2P-DC5) would just fit in the front, just RTV it to the side wall and power it from the 5 volts to keep the antenna connection open, tying the wiper to ground so when the power goes off the wiper shorts the antenna connection. This should not change the noise figure much, since any capacitive effect of the open lead is part of the circuit when it is tuned up, and is better then adding diodes back to back on the input. When transmitting into it from the rear I can assure you the device is history, and no amount of protection can guarantee the device will survive, so a second stage is advised when hot switching based on TX power detection. Such is life.

As I have said above, the equipment to provide an accurate test of IP3 will not be available until after this paper is published, and will be the subject of further research into the proper high IP3 output circuit design for this style LNA.

I was satisfied with the final performance of these LNAs, as they do their job quite well, and I am sure you will be satisfied also.

If you have question, your can contact me at w2zv @ w2zv.com and I will make an attempt to answer your questions.

Acknowledgements:

Thanks to Jim Klitzing, W6PQL, for making available PCB's for this design. Thank you Jim.

Thanks to Peter Manfre, WA2ODO, for helping me use his tuner for the initial measurements, for his help testing the units, and for his help and support with parts to get the design worked out. Thank you Pete.

Thanks to the CEM Corporation, my employer, for letting me use the shop and its tools, and allowing me to concentrate on finishing this paper before it was due. A big thank you to everyone at CEM.

References:

(1) Jim Klitzing's website for the LNA PCB is (http://www.w6pql.com/parts_i_can_provide.htm)

(2) Ole Nykjær, OZ2OE: "PREAMPLIFIER FOR 432 MHZ AND 1296 MHZ WITH MODERN HEMT-TRANSISTOR", VUSHF meeting held in Sweden in 2001, (http://oz2oe.dk/radio/atf54143/54143.html)

(3) Peter Hoefsloot, PA3BIY: "A very high dynamic range LNA for 144 MHz", *DUBUS* 1/2002 pp. 6

(4) Zdenek Samek, OK1DFC's website for the 432MHz LNA: (http://www.ok1dfc.com/EME/technic/ATF54143/index.html)

(5) Gary R. Greene, W2ZV: "Modifying the W6PQL LNA PCB for 144 and 432 MHz", Proceedings for the 2011 Southeastern VHF Society Conference, 4/2011 pp. 17

(6) RFSim99.exe can be downloaded at: (http://www.sandiego.edu/~ekim/e194rfs01/RFSim99.exe) or (http://www.tech-systems-labs.com/program-files/RFSim99.exe)

(7) The Mouser web site is (http://www.mouser.com/) or you can call them at: (800) 346-6873

(8) The Mini Circuits web site is (http://www.minicircuits.com/) or you can call them at: (718) 934-4500

(9) The Amidon web site is (https://www.amidoncorp.com/) or you can call them at: (800) 898-1883

Biography:

Gary R. Greene first got his license as a novice as WN2YEO in 1966, his Advance class license as WB2KVC and currently holds his Extra class as W2ZV. His roots are in Rochester, NY where he learned to love VHF, and his employment ranged from a 2-Way radio tech (for 8 years) though to Director of R&D and everything in between. He has a Masters in EE from RIT and is a member of the IEEE and the Carolina DX Association and holds 6 patents. He currently lives near Charlotte, NC with his wife Susan, and works for CEM Corporation as a Principal Senior Engineer, fixing hardware problems in software for a company that makes bench top microwave heating instruments and accessories for the process control and chemical industries. Our mantra is "we sell time".