

Filter Example

You get a filter design request:

I have had quite a few requests to do a family of filters for the ham bands.
Heck, maybe throw in an ISM band or two.

Mainly aimed at harmonics vs close in stuff.
So low Q.

And I can get them made on fiberglass for about \$10 dB less than on Rogers.

I would think loss would be minimal if the bandwidth was kept wide.

Any thoughts?

Hi all,

I ran a quick test to measure output power and second harmonic output of my HackRF clone in narrow band FM Tx mode.

Here are the results:

435 MHz +6 dBm, -26 dBm

902 MHz +8 dBm , -21 dBm

1296 MHz +5 dBm, -25 dBm

2304 MHz +13 dBm, -12 dBm

3456 MHz -3 dBm, -50 dBm

5760 MHz -11 dBm, none

From this create a specification:

Specification

Impedance Level 50 ohm

Connectors SMA, tempted to be cruel and use SMP's
which will require purchasing expensive
adaptors. Edge mount. After all, the spec was
not thought out.

Passband Frequencies Various

Passband Loss Low as practical

Stopband Rejection >40 dB. FCC requirement 60 dB total

Board Material Not Rogers.

Board Thickness 20 mil.

The board material is specified it a weird way. Rogers makes several excellent microwave board materials, but they tend to be costly. FR4 type materials generally are lossy and have uncertain dielectric constants making control of the final filter performance difficult. A specific vendor of PCB material would help.

Left to my own devices I come up with two. Panasonic Megatron 6. This is actually good low loss microwave material. Isola FR408hr is a little bit higher loss and also FR4 processing.

Material	Er	Dk
Megatron 6	3.40	.002
Isola FR408hr	3.68	.0092

The next option is the construction. Lumped filters would use one layout. Lumped filters are closer to ideal, and would likely require fewer sections. To get close to ideal the actual components would need to be small, 0402 in size which require a microscope to assemble.

Distributed filters are limited by the high impedance that can be generated by a microstrip line. This is typically ~93 ohms. TXLine is a free program to compute various transmission line dimensions available from AWR. Use low Z lines for the caps and high Z lines for the inductors.

Filter Prototype

There are a number of different types of filter response patterns. Butterworth have no ripple in the passband. Chebyshev have ripple in the passband or stopband but have steeper transitions to the stopband than Butterworth. Elliptic or Cauer filters have steeper transitions to the stopband and add transmission zeros.

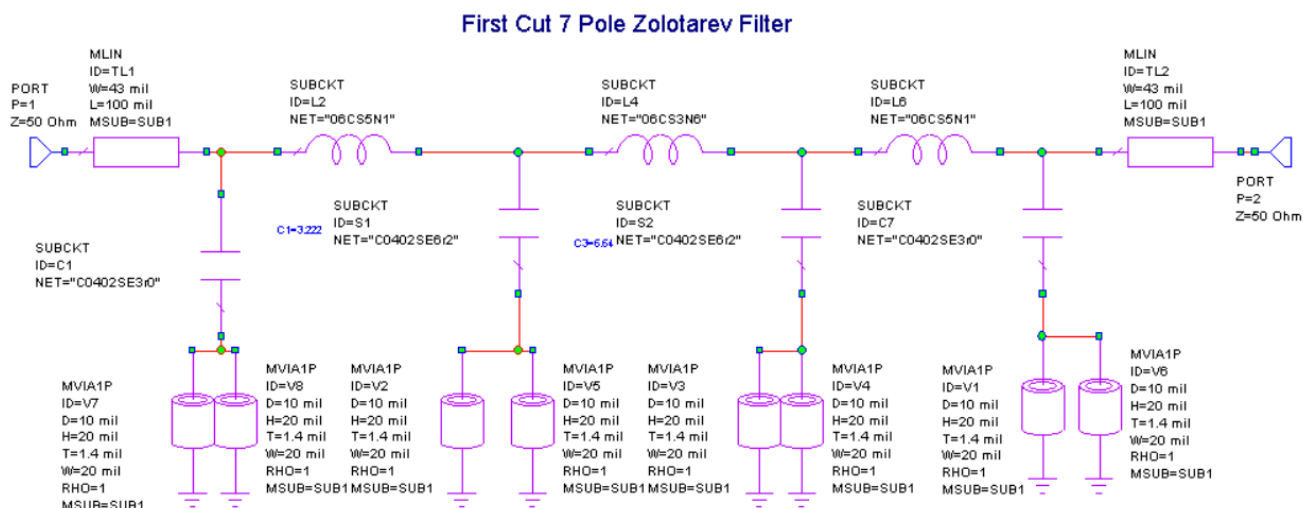
All these prototypes have low loss from zero Hertz to the corner frequency. Zero loss up to $\frac{1}{2}$ the stopband frequency is an unnecessary constraint. You are getting no rejection in that area. Zolotarev filters remove that constraint allowing a steeper passband to stopband transition; at the expense of more extreme internal impedance levels.

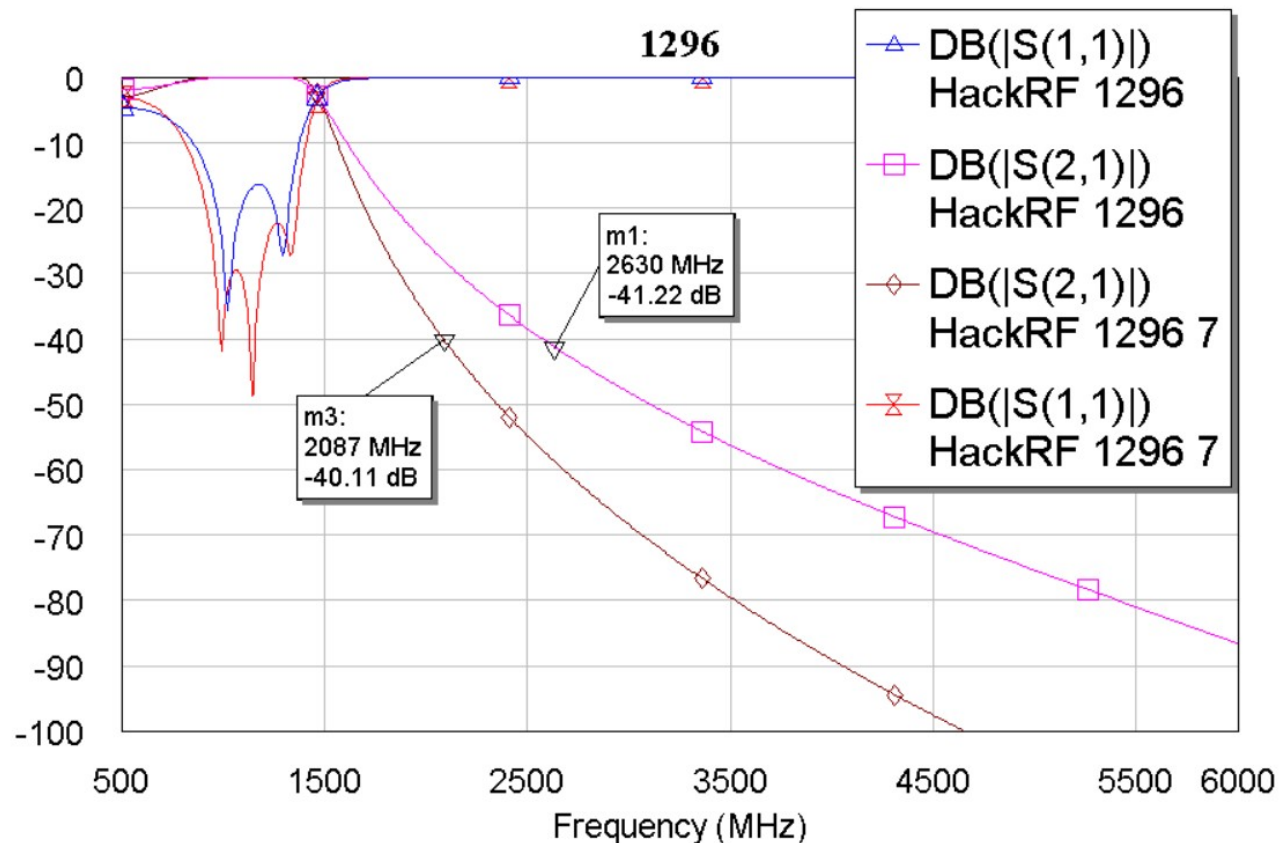
I denormalize the filters in a Mathcad 15 file. Mathcad 15 offers significant advantages over the new Mathcad Prime versions.

The advantage of keeping the work in a Mathcad file is ease of modification, even after you haven't used it for several months.

The first cut of a lumped filter for the 1296 band doesn't meet the 40 dB attenuation goal. Even adding a via inductance doesn't bring the series resonant frequency of the caps down enough to matter. Going to a 7 pole filter.

The 7 pole filter should have excess stopband attenuation, so lower ripple could be tolerated.





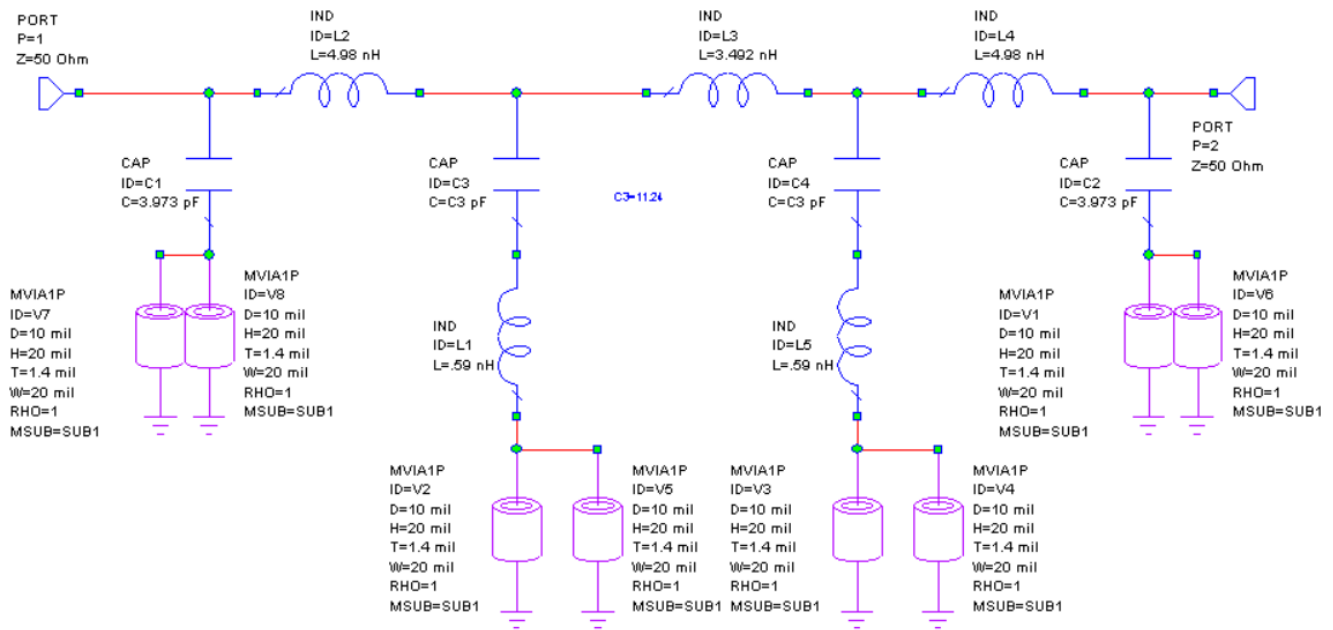
The ideal LC 5 and 7 pole filter responses.

Switching to capacitor input is quite close to the same, except adding ground via inductance to the large capacitors creates a trap...Leading us into the inductance of a physical capacitor.

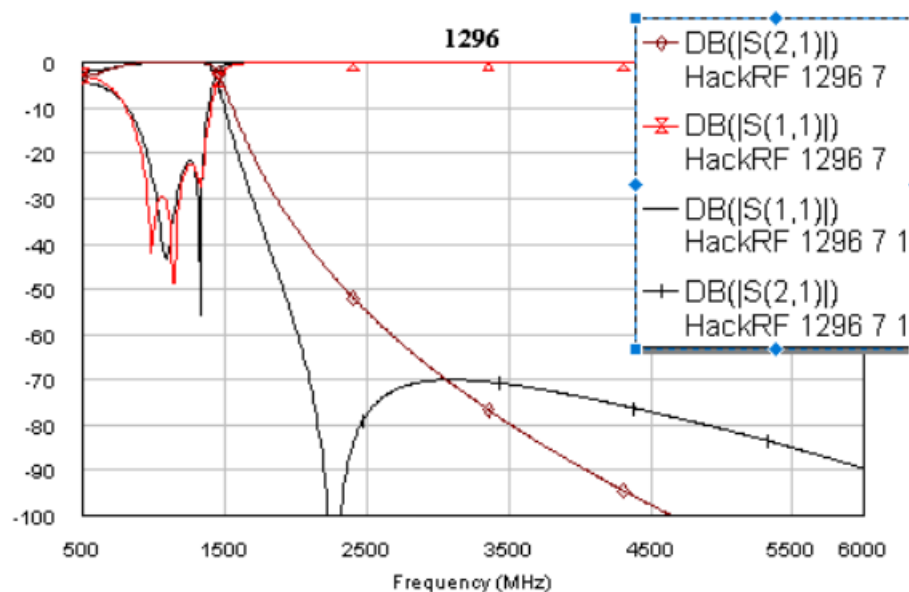
CHIP PKG	DIM (mil ²)	L(nH)
0402	40x20	0.59
0603	60x30	0.77
0805	80x50	0.84
ATC 100A	55x55	0.55
ATC 100B	110x110	0.77

You would expect the ground via's being ½ the length of the 0402 part to have about ½ the inductance. I will eventually go to an EM model of the vias, in an effort to get the most accurate model possible.

Continuing with a lumped version of the filter, replace the capacitors with a series LC model in MWO



C3 is a tune variable which allows me to quickly adjust the value and response. C1 gets the same treatment.



C1 and C7 get the same treatment to improve the inband match. The next step is to select the capacitors. The resonance about 2300 MHz is close to the 1350 corner frequency, probably making it sensitive to value. So I need small steps close to the tuned value and a dielectric which is good at 2600 MHz. AVX's Accu-P line has those characteristics. They also provide both a model and measured data. The measured data settles the value and the model will be used to predict the performance when the production tolerances are included in the yield prediction. While the series resonance of the shunt elements gives us faster attenuation roll off close to the passband, at higher frequencies the net inductance will cause the attenuation to decrease.

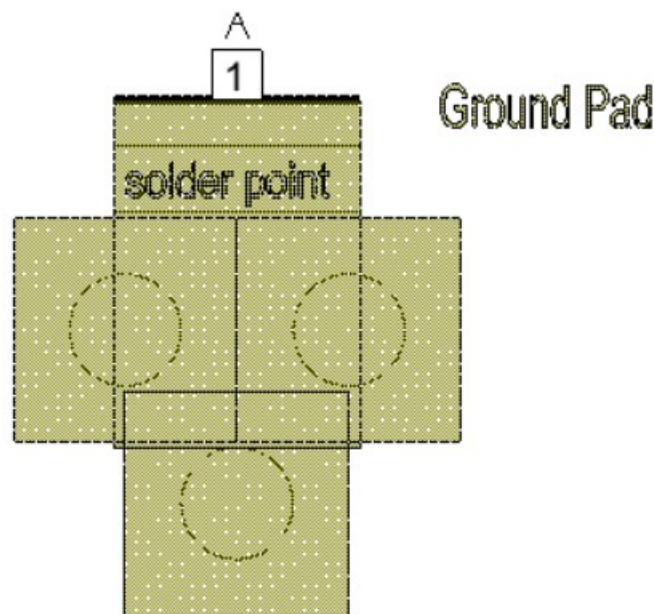
For the lumped filter inductors need to be chosen. Coilcraft 0603 HP and CS series work for the same reasons the capacitors did....measured data and an existing model that matched the data. I prefer to replace parts one at a time so I can see what disrupts the performance.

The inductors caused the filter corner frequency to drift down and the passband match to degrade. The corner is adjust by reducing C3 and C4. The match is adjusted by values of C1 and C7.

At this time you can begin the layout. MWO vendor libraries have foot prints associated with the parts. AVX 0402 capacitors and 0402 inductors have different footprints. Coilcraft has a variety of different 0402 inductors with several different footprints. While working with customers I have found that manufacturing may dictate what size pads are necessary. The PCB layout group may have standardized upon a foot print. Change the foot print in MWO to what your organization requires.

The ground pad needs EM analysis. Copy it from the layout of MWO into the EM Structure page. When we did the schematic simulation we used both one and two via's to ground the capacitors. Two vias offers significantly less inductance and higher resonance frequencies than one. Via's are considered to be low reliability items when doing hi-rel products. Two via's are the minimum.

Vias need an angular ring around the hole. The total is due to errors in the imaging, drill and registration. I start with the pad to be twice the diameter of the hole which requires advanced manufacturing capability. This is also a function of the number of layers. Via's under the part seem like the best electrically but they will fill with solder, maybe impierilling the integrity of the solder joint.



Ground Pad.

The circles are the 10 mil ground vias. The solder point is where the actual 0402 solder mask is eliminated. The solder mask prevents the via from pulling the solder away from the pad on the part through surface tension. The top via may need to be filled to prevent the solder to be pulled from the joint.

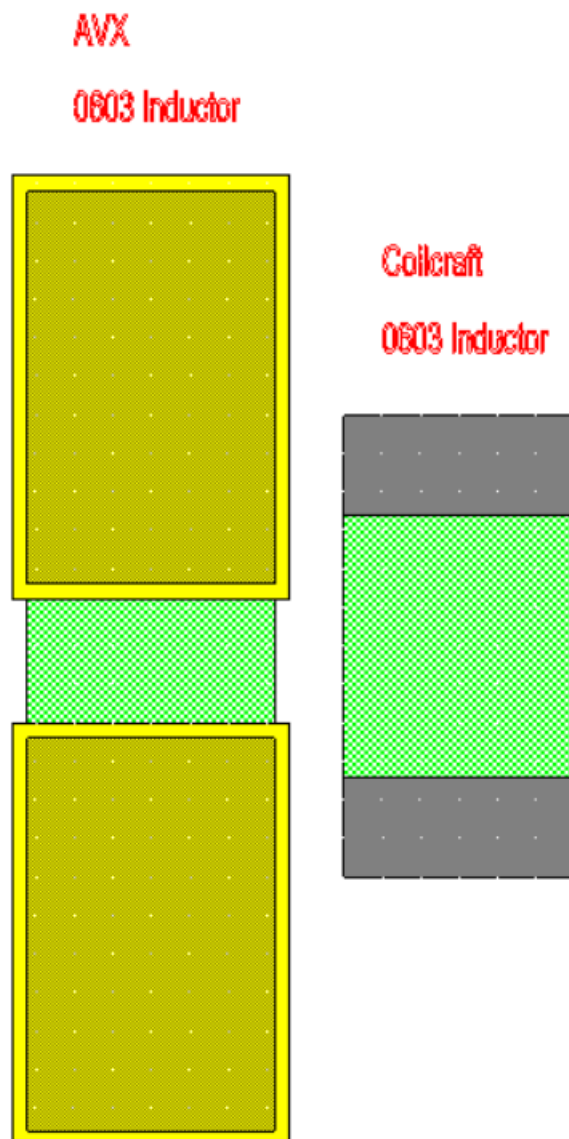
Real	Imaginary	Comment
-0.9946	0.1004	All data at 1300 MHz
.0046	1.7801	Perfect Conductor 3 vias
		1 oz copper 3 vias

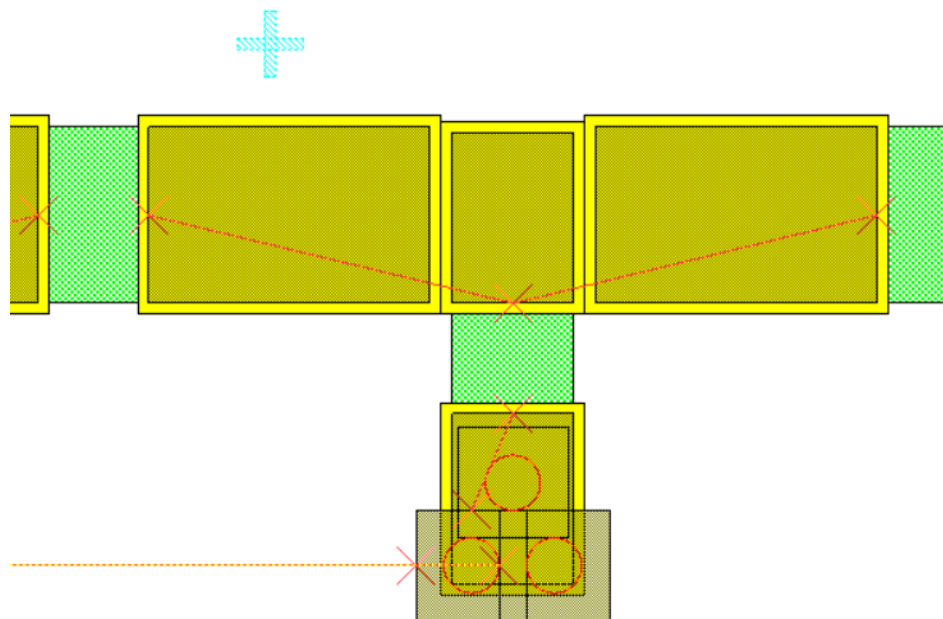
.0043	1.799	2 vias, 1 oz copper. Upper vias only.
.0048	2.2366	2 via, 1 top & lower
.0082	2.4747	1 via, top

The imaginary part shows a significant change in the reactance with the number of vias. The worst case, 2 via example is 0.4j ohms different from the 3 via example. It looks like the 3 via option is acceptable.

The Inductor Pads.

The Coilcraft 0603 footprint is shorter than the AVX 0402! The capacitor pad is larger than the part. The Coilcraft footprint is the size of the part. If you butt that up to the capacitor pad there is not enough room for any assembly tolerances. I switched to the AVX0603L footprint which has a larger pad which will allow for some tolerances. The holes in the solder mask layer are in the same place but the copper is larger. There are 31 mils of copper beyond the part outline on the AVX part which are not there on the

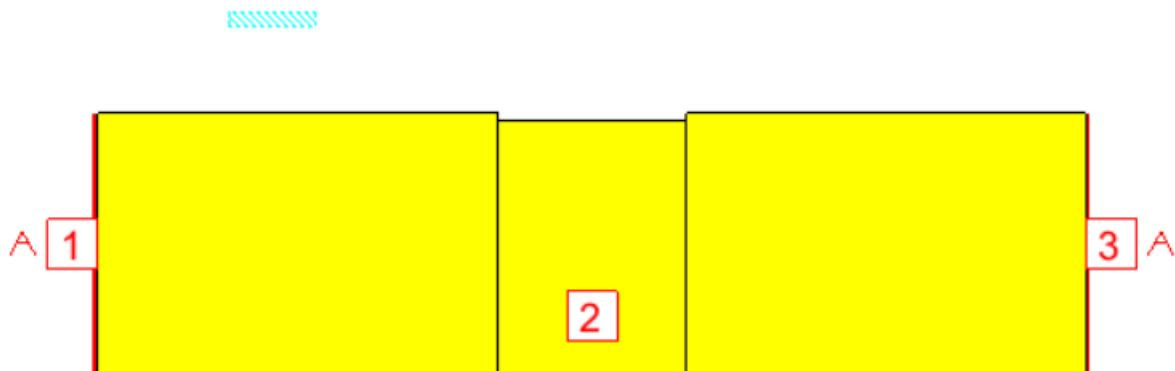




Layout view of the interior pads.

Please ignore the rat lines. I later switched from this layout, of 1 up and 2 vias down to 2 up and 1 down. I also used an edge pad instead of a point pad to get closer to a totally passive response. A negative real part isn't happening in the real world.

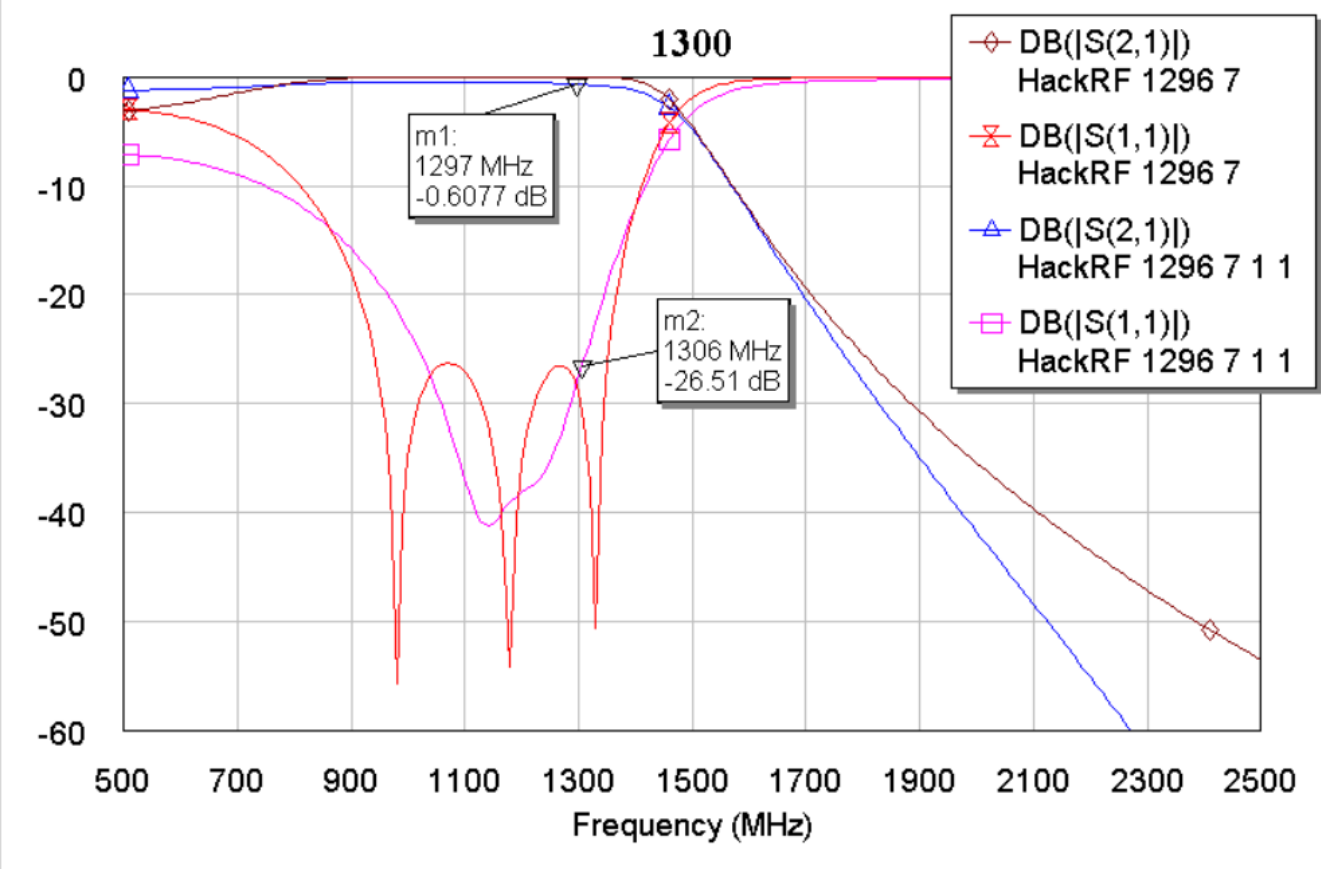
The interior model



I used two edge ports for the inductor connection and an interior port for the capacitor.

When you add the interior pads the corner frequency is pulled in a bit due to the capacitance of the metal... Going to the next size smaller capacitor moves the corner back out.

At this point our filter is nearly complete. The response meets requirements.



Filter Responses.

The HackRF 1296 7 is the original 7 pole Zolotarev design. The HackRF 1296 1 1 has the distributed elements built in and the updated schematic follows.

This filter looks like we have a functioning design. The loss is a bit high due to losses in the inductors. Right now it works if the parts meet the specifications.

Yield.

It would be nice to be finished now but we don't know how the filter will perform in production. Components have tolerances. For many reasons vendors cannot make the same value every time no matter how hard they try to control their process. This is a way to try to control our process to have an acceptable product.

Tolerances.

Capacitors	+/- .1pf	Shape: Uniform
Inductors	5% or 2%	Shape: Uniform
PCB Material	?	?

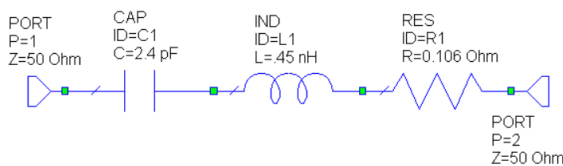
The tolerances may actually be Gaussian but uniform should be a more difficult condition. The distribution across the range in Gaussian tapers as we move away from the mean but is unbounded. Getting the standard deviation for most vendors is difficult to impossible. The uniform distribution has

a higher population near the tolerance limits but is bounded and has ideally no parts outside of the limits.

The measured values for a typical component are replaced with a model of the part with tolerances. The models are matched to the nominal performance. The models with tolerances enabled are placed in the circuit and evaluated in a Monte Carlo fashion to see what the production performance bounds are likely to be. Monte Carlo simulation gives you a fair idea of the extremes of production will be. Any one lot is liable to be near an extreme in performance giving the idea that the performance may be improved. This is a bad idea since yield is lumpy; production lots of components have tighter tolerances than expected, but the next lot will have a different distribution.

Capacitors

Capacitor Model



A quick evaluation of the model show that below 10 GHz the part is not very sensitive to the resistance. The performance is sensitive to the inductance and there is some expected positioning variation. I am going to use a WAG of 10% or .05 nh.

The series resonance of the 5.6 pf capacitor is higher than the 2.4 pf model would indicate, the series inductance is lower. That could be because of wider metalization but I don't really know. With a 0.4 nh series L the model matches to 20 GHz.

AVX Accu-P Tolerances

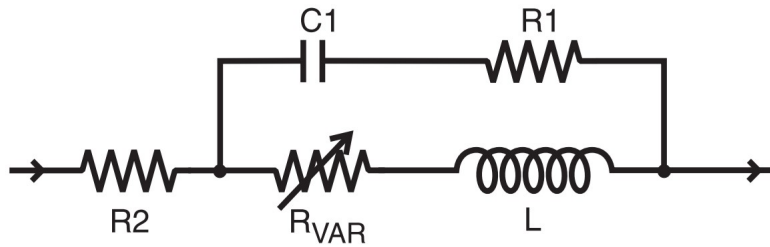
Label	Tolerance
Q	+/-0.03pF
A	+/-0.05pF
B	+/-0.1pF
C	+/-0.25pF
For C<5.6 pf	
A	+/-0.05pf
B	+/-0.1pF
C	+/-0.25pF

Inductor

Coilcraft provides an excellent discussion of their model. Using a spice model instead of measured S-parameter data will make it easy to vary the component.

$$R_{VAR} = k * \sqrt{f}$$

- k is shown for each value in the accompanying table.
- f is the frequency in Hz



Part number	R1 (Ω)	R2 (Ω)	C (pF)	L (nH)	k	Upper limit (MHz)
0603CS-1N6	2	0.001	0.030	1.6	6.50E-06	25300
0603CS-1N8	2	0.001	0.035	1.8	1.00E-05	22100
0603CS-2N2	23	0.070	0.024	2.2	11.22E-06	23000
0603CS-3N3	12	0.002	0.032	3.3	8.90E-06	17100
0603CS-3N6	9	0.002	0.040	3.6	1.25E-05	14600
0603CS-3N9	9	0.002	0.042	3.9	1.40E-05	13700
0603CS-4N3	9	0.003	0.043	4.3	1.31E-05	12900
0603CS-4N7	10	0.003	0.042	4.7	1.99E-05	12500

The inductors come in 2,3 and 5 percent tolerances. Only the inductance value will be varied for estimating the performance.

PCB Material

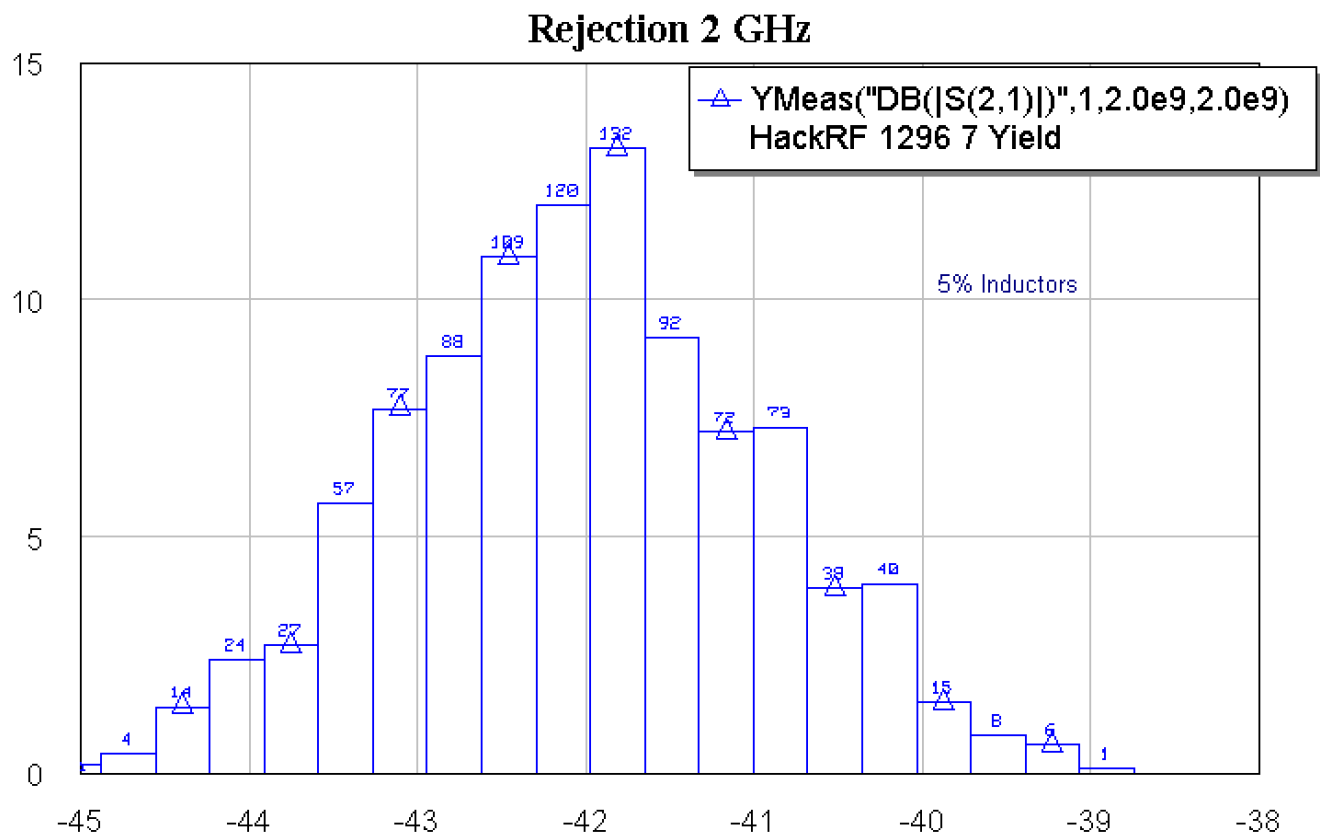
This is a difficult one. Even the good RF materials don't provide a tolerance to the dielectric constant or the dissipation factor. There is also some variation in thickness. Even though the width and length

are photolithographically defined there will be some variation. The etch tolerances are defined by the processor not the material manufacturer. The RF microstrip layers are usually done on rigid material, since prepreg has greater thickness variations. Using the outer two layers for microstrip requires that they be specified as rigid material for the PCB fab; otherwise they may be prepreg.

The PCB material summary is neither Isola, Panasonic or even Rogers(RO4350) specifies the thickness or dielectric constant variation. How about a guess of ± 0.5 mil. This change is across all the microstrip elements.

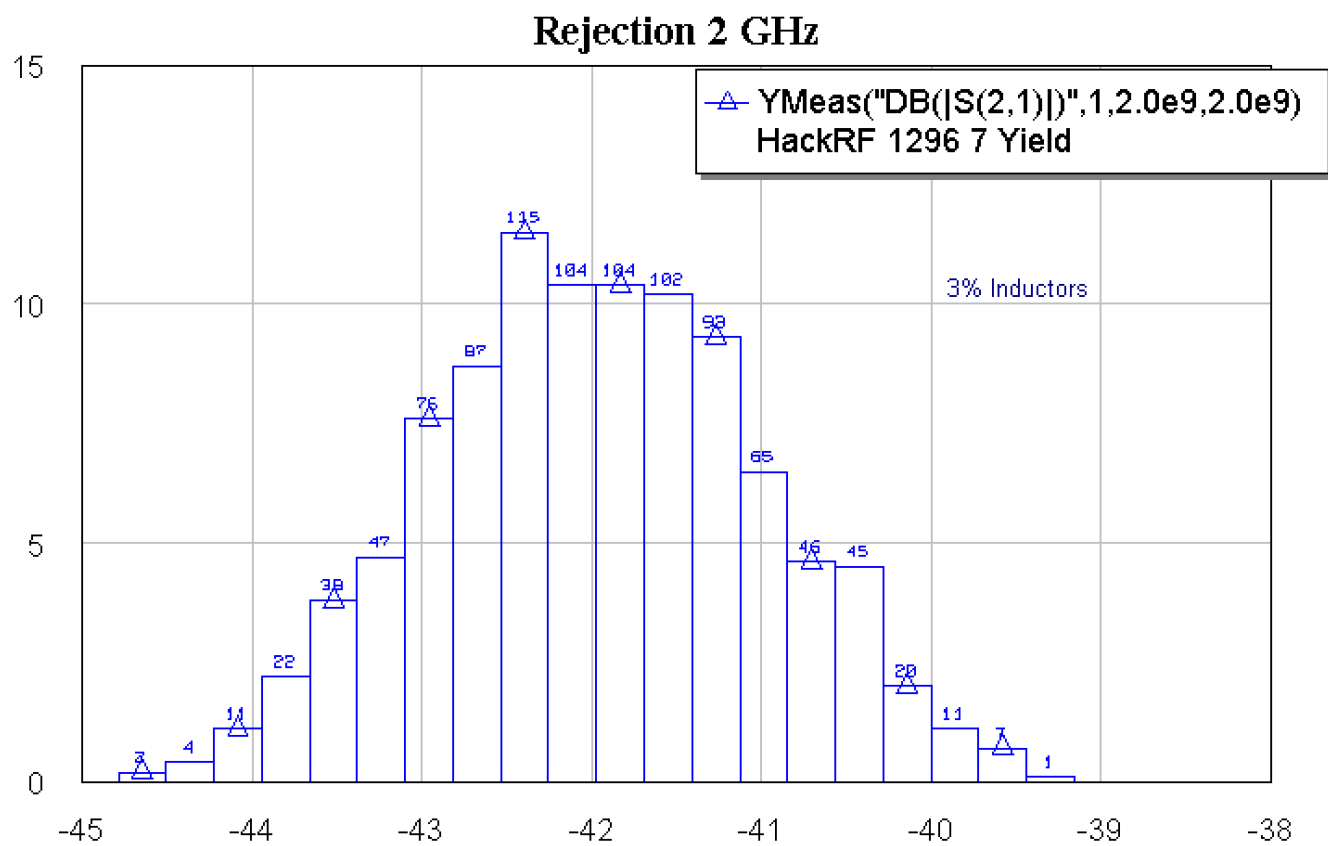
Yield Results

Using 5% inductors, and 0.25 pf tolerance caps:

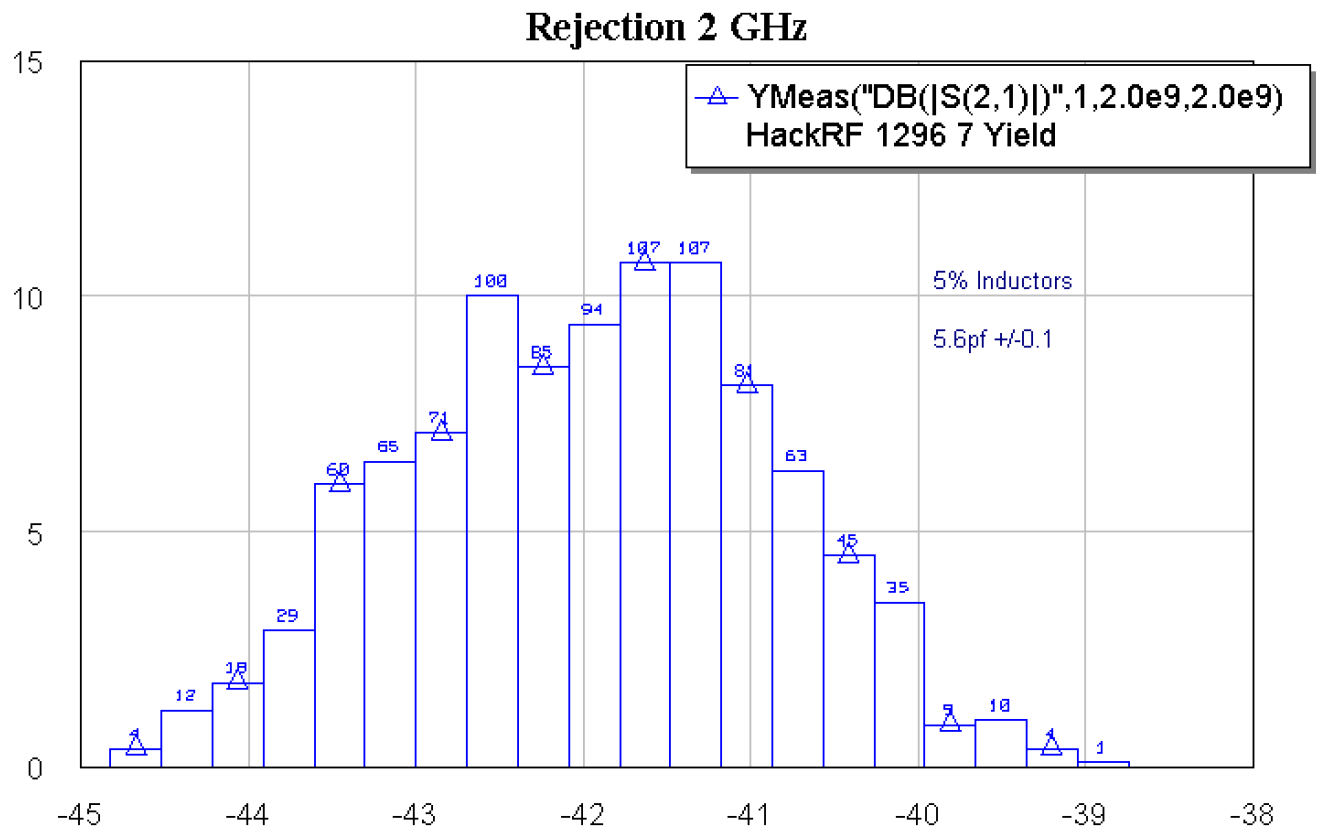


We fail to meet the 40 dB rejection about 2% of the time at 2 GHz.

Lets tighten up the inductor tolerance and see what improvement we get:



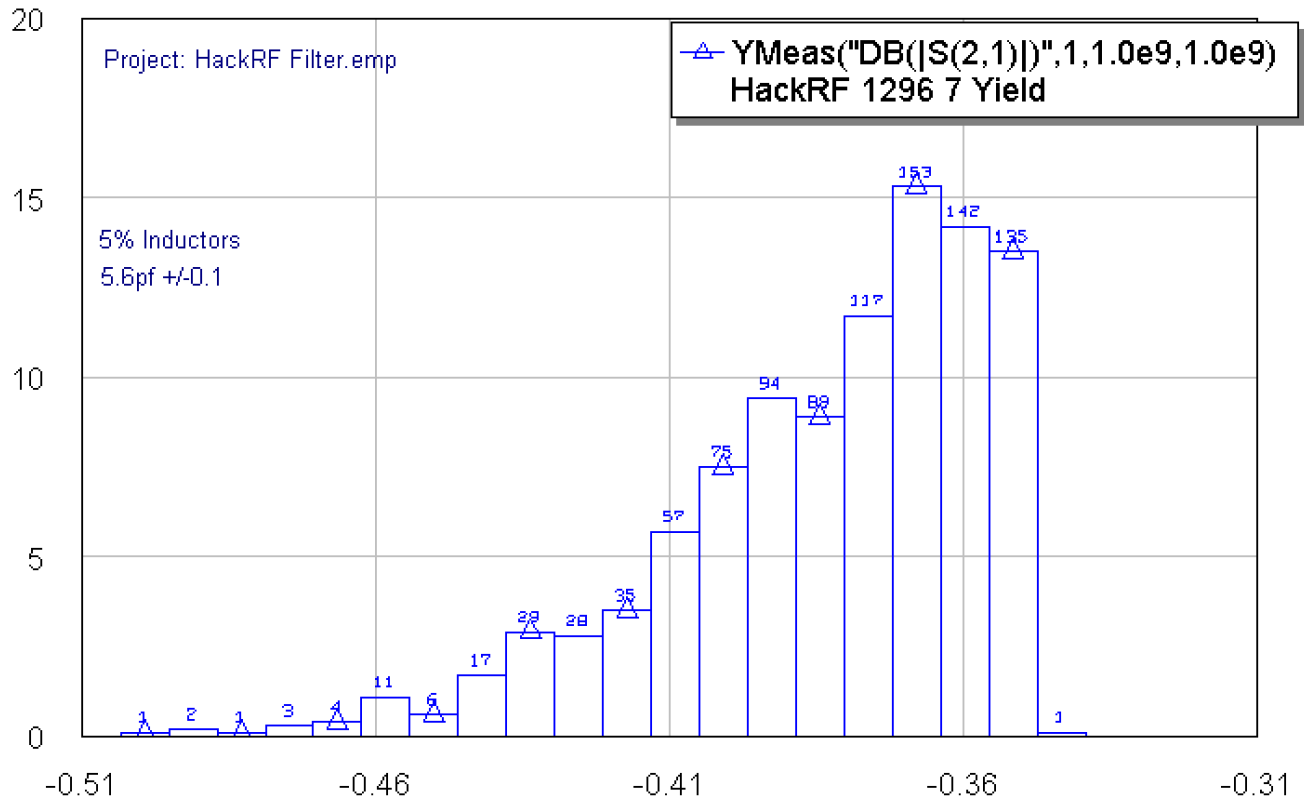
Tightening up the inductors from 5% to 3% only gained about 1% better. Applying a little be of the knowledge built by building in the parasitics the rejection slope was set by the resonance of the 5.6 pf caps. Tighten up those caps to 0.1pf from 0.25 pf.



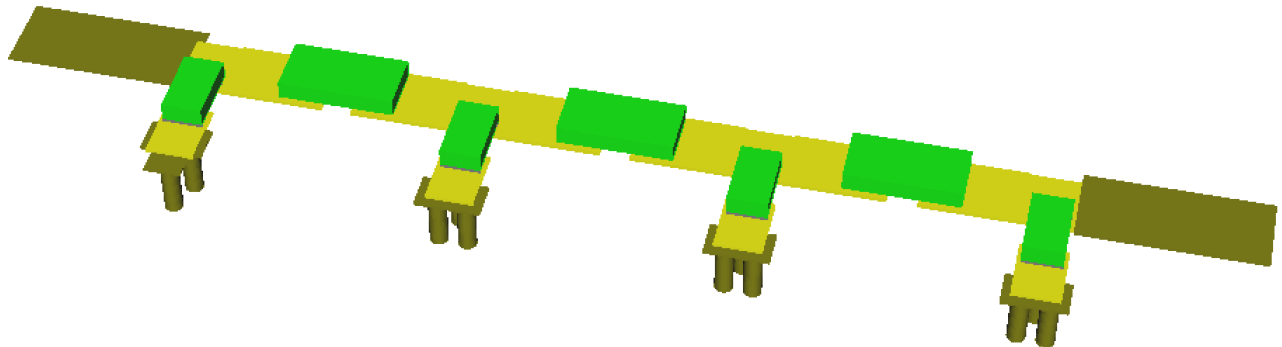
We still have a tail above 40 dB. At this point it may be less expensive to add another pair of poles and use the lower tolerance parts. Another alternative is to go to a higher frequency for optimum S11.

In band Response

Loss 1.0 GHz



3D layout view



Production

If this design is taken to production what should the test specifications be? Taking a VNA sweep looks great and gets as many data points as you want. But how do we store the data? Pictures sound great but seeing the results of a production lot to compare is quite difficult.

Now we have a new customer, the Quality Control Department. They need simple specifications, such that the inspector can see the production test data and see that the product is in control. Control means something special to them. Control means that the product is varying within expected levels. I will cover creating production specifications in a later article.