Electronic filters design tutorial

PREFACE

I designed my first low pass filter in 1975. It was a T filter, used to reduce the harmonics of a 5 W transmitter. The transmitter was already built in mass production, but it did not pass homologation tests. Inserting an external boxed filter on the antenna output, it passed all homologation tests, and I gained a solemn encomium.

Since then, in a third of a Century, I designed hundreds of filters, with different topologies and use. Starting from hand calculation the first years and going thru the various and more refined CAD systems, decade to decade: before SPICE simulators, and after EM ones.

In all these years I spent a lot of time to understand the concepts behind filters.
Mathematic approach is important, but only a number of filter approximations are available for the design. If your needs do not fall into someone of these classes your design will be non optimal.

In my opinion the best simulator available is our brain! All available simulators today are number crunching, but if you don't have in your mind the right model of the circuit, the simulated circuit could be anything from perfect to useless. It is not the case to came up with Murphy's law! Simply if the result is perfect you have chosen the right model/circuit for your application (maybe you are a clever engineer, or maybe you are a bit lucky!), but if the circuit do not operate properly this means only that the model is incorrect or incomplete in your application, or the circuit chosen do not fit the application.

In every case you have to increase your knowledge of the circuit/system before designing it. This seems very simple, and is the basis for every engineering work, but unfortunately the today culture is to consider circuit blocks as "black box" and to avoid the study inside them, merely connecting blocks one to the other, and hoping Mr Murphy will be far for holidays and anything will go well!

This is true for every circuit, system and situation in life.
A simple example? If you buy an electromagnetic simulator program which operate in "closed box" environment, it will be perfect for simulating filters and circuit matching, but it will be poor in antenna simulation, since the closed box will alter antenna impedance and radiation pattern. I'm not saying the simulator is not useful , but it will be more difficult to obtain good result.

If you have chosen a "closed box" electromagnetic simulator for the development of antennas, probably you do not have enough specific knowledge about electromagnetic simulators for the design of antennas, and you hope to compensate your loss of specific knowledge by means of an EM simulator, that will unfortunately be of scarce help.

Instead, if you read at least one book on EM simulator you will be able to choose the best program that fit your application!

This tutorial is divided in five sections:

1) **Band pass passive filters**

2) **Band pass filter with distributed components**

3) **High pass, low pass and notch passive filters**

4) **Active filters**

5) **Time domain filters**

It is expected from the reader a general electronic knowledge, but the use of mathematic is purposely reduced to a minimum, the scope of the tutorial is to improve the practical knowledge of filters, mathematic is demanded to the simulators used in many examples. In the bibliography, books are listed that investigate also mathematical aspect of filters.

Par 1) THE RESONAT CIRCUIT

The circuit of Fig.1 consist of a parallel resonant circuit at 100 MHz, loaded with a 50 Ω resistor, the generator is a perfect voltage source. Since both the inductor and the capacitor exhibit a reactance of 50 Ω the Q of the circuit is 1 for definition (Q = R/XC, or = R/XL).

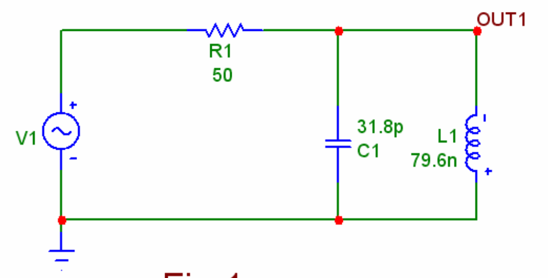


Fig.1

Per definition the BW of the circuit at 3dB point is equal to: **BW = 1/Q**

and hence the bandwidth is equal to the central frequency, 100 MHz.

Running the simulation (fig 2) it can be seen that

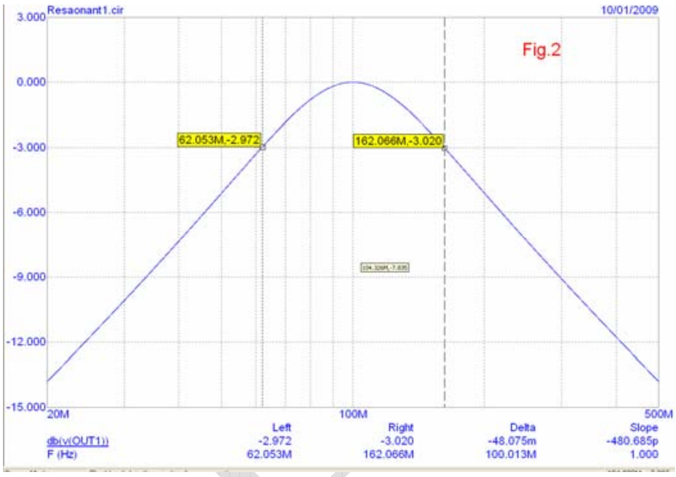


Fig.2

the BW goes from 62 MHz to 162MHz, or exactly the 100 MHz expected.
Running a subsequent simulation we step the resistor from 50 Ω to 250 Ω, enhancing the Q from 1 to 5. The BW should decrease in proportion, moving from 100 MHz to 20 MHz.

It can be seen in Fig3 that the simulation displays as expected. The same result can be obtained lowering the impedance of the reactive elements, for example increasing the capacitor value five times and decreasing the inductor value at 1/5 of its value will yield to an impedance of 10 Ω and the Q will be again 5.

Note that impedance of inductor and capacitor has to be changed simultaneously, otherwise a resonance shift in frequency will occur.

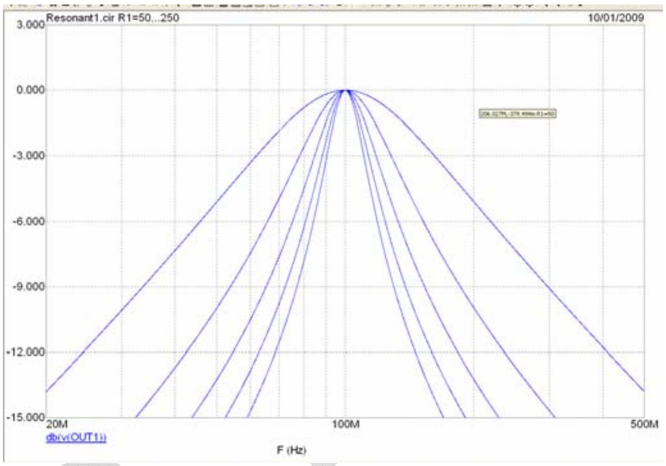


Fig.3

Now we have all the elements to realise a band pass filter; we have only to add two reactive elements in the circuit and place the load at output to 50Ω, in order to maximize the power transfer. The two reactance in series with generator and load determine the Q of the filter.
Here the relationship is more complex, since input and output coupling are now complex number, due to the sum of 50Ω resistors with series reactance. Doubling the reactance do not exactly halves the bandwidth.

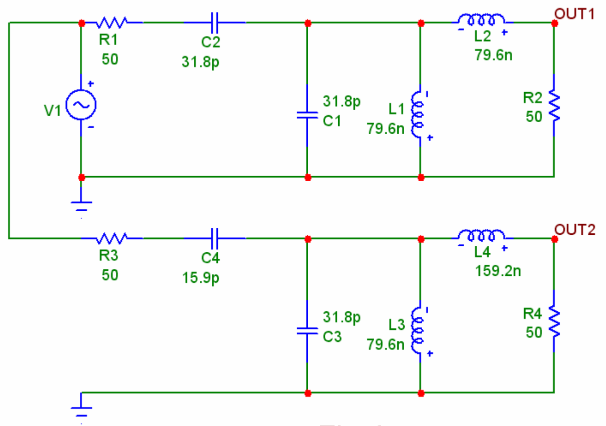


Fig.4

In Fig 4 appears such a filter where two different circuits are simulated with series reactance of 50Ω and 100Ω. The response in Fig.5 makes evidence of the two bandwidth.

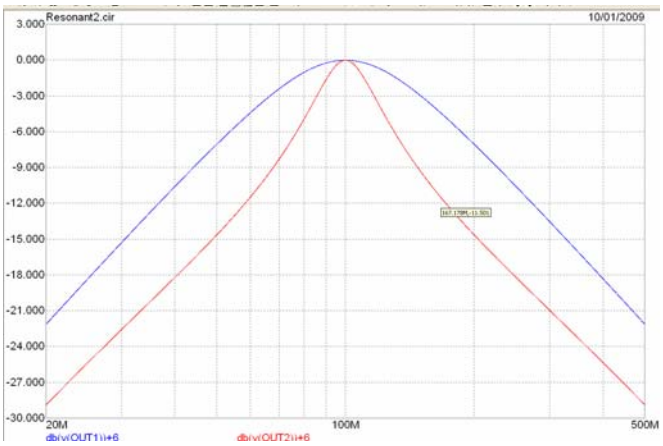


Fig.5

Moving the values of C2 and L2 we are able to set the bandwidth of the filter, note that the two reactance must be equal in amplitude (and opposite in sign, in this case) otherwise central frequency will shift and insertion loss won't be zero (simulation is made with lossless components)

Till now the circuit is made with two complementary series reactance.

What happens if the two series reactance are equal?

In Fig 6 three circuits appear simulating the different possibilities. It is very interesting to check the effects of this modifications on the filter response in Fig.7

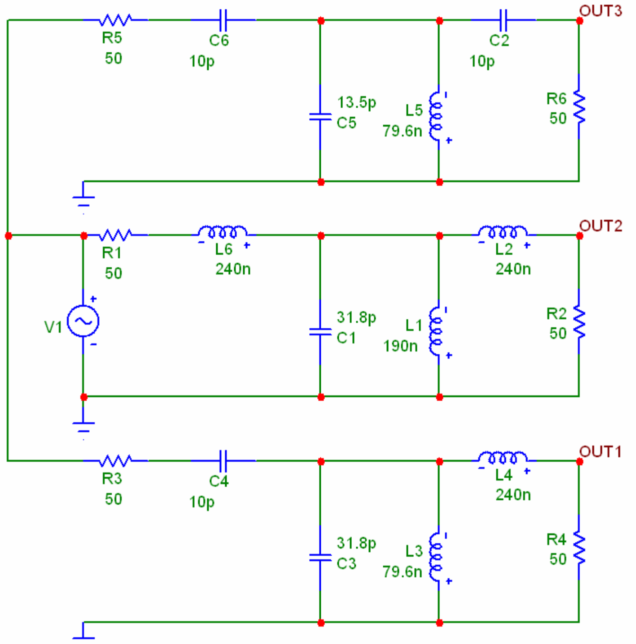


Fig.6

The response show that while the filter with series inductor and capacitor has a symmetrical response (blue trace), the two inductor filters has steeper response at higher frequencies (red trace), this network is normally referred as **low pass coupling**. The two capacitor filter has steeper response at low frequencies (green trace), this type of coupling is referred as **high pass coupling**.

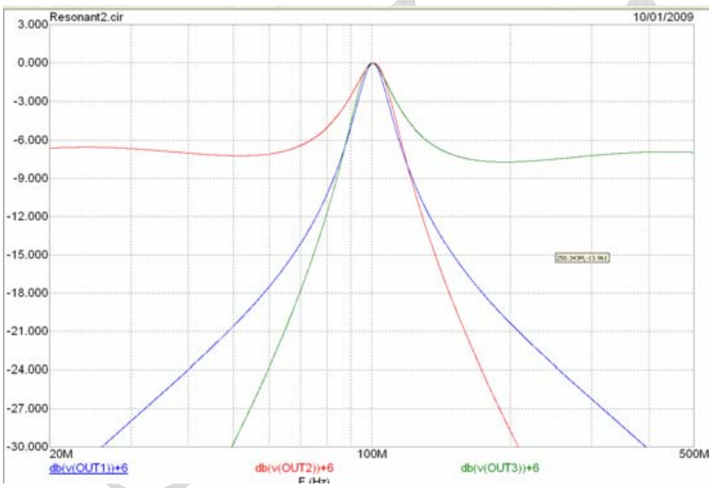


Fig. 7

Note that in the two coupling capacitor filter, the value of resonant capacitor C5 has been decreased in order to compensate the frequency shift caused by the two coupling capacitors, in the two inductor filter inductance L1 has been increased to compensate for the frequency shift caused by the two inductors.

* **these different frequency responses lead to a great flexibility in applications.**

For example, if you have to design a front end filter for a receiver with an image to reject <u>below</u> the operating frequency, a filter with capacitor coupling will maximize image rejection.
If the first local oscillator is operating beyond the operating frequency, the image frequency will appear also over the operating frequency; in this case the two inductor coupled filter will give the maximum image rejection.

* **To recapitulate:**

1) With a very simple calculation we are able to set a resonant circuit with a loaded Q of 1 to start the design of a filter
2) Placing the proper reactance in series with the generator and the load we are able to change the Q of the filter, and consequently the BW
3) Changing the type of reactance in series we are able to manipulate the response of the filter, enhancing cutoff at higher or

lower frequency, or making the filter symmetrical.

Spice simulators: all Spice based simulator are useful for this type of analysis, even a linear simulator can be used for this first section with passive filters.

If you do not have a Spice simulator you can freely download one of the following:

A) Spectrum Software: Microcap-9 http://www.spectrum-soft.com/index.shtm free demo version downloadable with reduced performances, but adequate for the trials with this tutorial. For learning purpose the evaluation version can be used indefinitely. All the simulation presented here are made whit the full version of Microcap-8

B) Texas Instrument: Tina-TI http://focus.ti.com/docs/toolsw/folders/print/tina-ti.html a free version of TINA developed for Texas Instrument. Actual version is 7.0

C) Simetrix: SIMetrix http://www.simetrix.co.uk/site/downloads/download.htm another free version of spice simulator

C) Linear Technology: LT spiceIV http://www.linear.com/designtools/software/#Spice spice simulator developed for Linear

Multiple resonators filters

Filters used in real life rarely are constructed with a single resonator, because the rejection is poor in most applications. Moreover the shape of the response is gaussian and again in many application the shape of the filter is requested more rectangular.

In Fig.8 two single resonators are connected together:

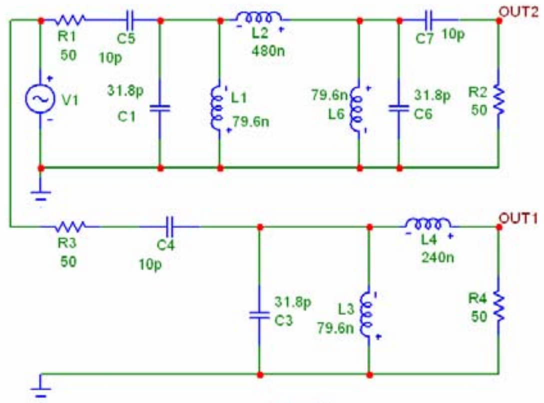


Fig.8

The two inductor are substituted by a single one, L2, with twice the value. Running the simulation we should expect a filter with a sharper response and a bandwidth slightly reduced with respect to the single pole. Running the simulation (Fig.9)

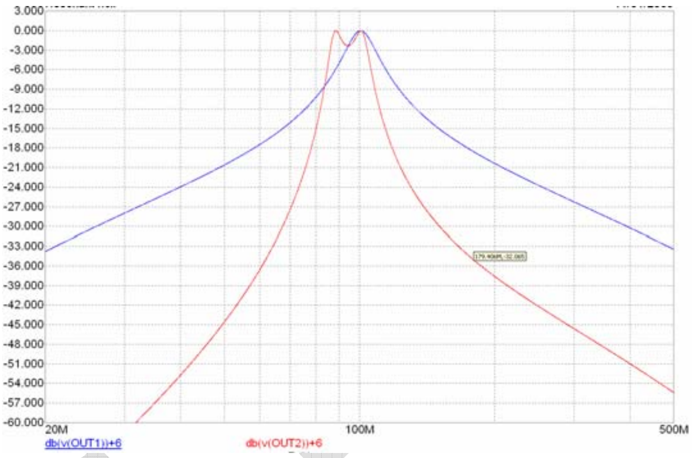


Fig.9

we can see that the filter has a ripple of 3 dB in the passband and a slight center frequency shift of approximately 7 MHz.

Both are caused by the termination between the two resonators that now differ from the resistive 50Ω seen in the single stage filter.

Editing the value of L2 we can correct the ripple in the BW. In Fig.10 the L2 value is stepped from 600 to 1000 nH in 100 nH steps.

The two tuning capacitors C1 and C6 has been reduced to 25 pF to compensate for frequency drift. The vertical divisions are set to 1dB.

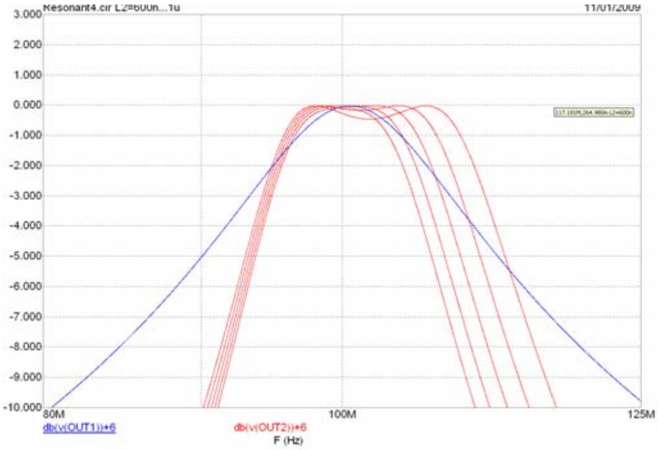


Fig.10

Stepping the inductor L2 has three effects simultaneously:

1) change the bandwidth of the filter
2) change the ripple in the bandwidth
3) shift slightly the center frequency

The point 3) can easily be overcame tuning capacitors C1 and C6 to the right frequency.

About the point 1) and 2) you can select L2 for the best ripple in band, and adjust the bandwidth of the filter as will be explained later, or if the

ripple is acceptable for a given BW, you can fix the filter as is.
In Fig.11 the response of the filter is show, with L2 set at 800nH for maximum in-BW flatness.

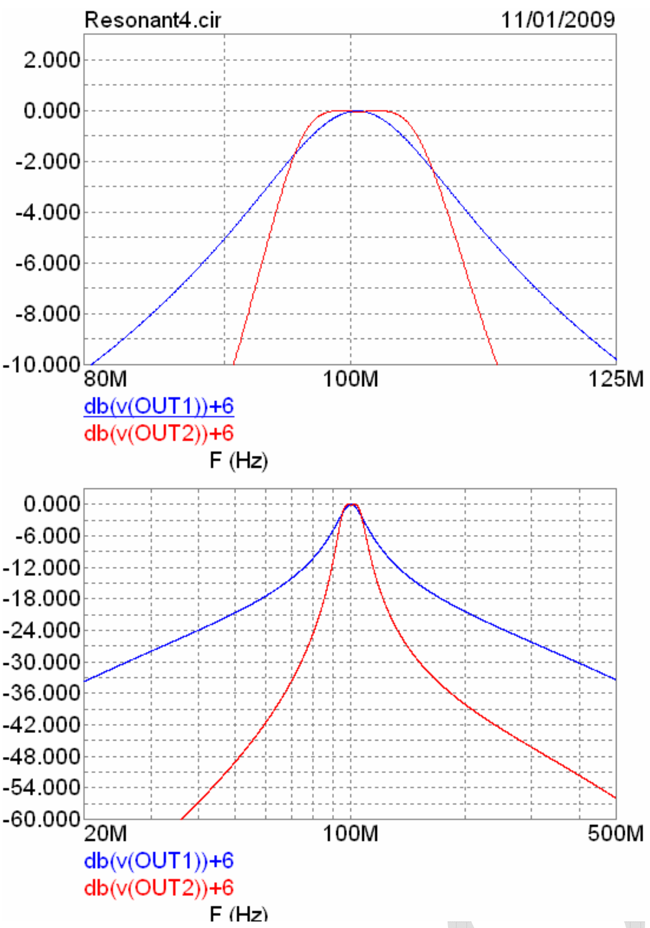


Fig.11

Both the close-in response and the broad band attenuation are shown. You can see the dramatic improvement in response compared with the single resonator filter (blue trace). The in- band response is more flat, and the attenuated BW is extremely sharp (red trace)

What happen now if we want to change the bandwidth of the filter? Simply you have to edit the parameter of the series components, C5–C7–L2, changes must be done together, decreasing the reactance of L2 (decreasing the inductance of L2) and simultaneously decreasing the reactance of C5 and C7 (increasing the value of the capacitors). Unfortunately there isn’t a perfectly linear relationship between the BW and the value of these components. In fig.12 we initially halves the value of L2 to 400 nH and increased C5 and C7 to 20 pF. As expected the BW did not double, and the filter exhibits a poor response. Changing C5 and C7 to 15 pF restore the shape of the filter and the BW move from 13 MHz to 25 MHz.
See the diagram in Fig. 12 and the relative responses.

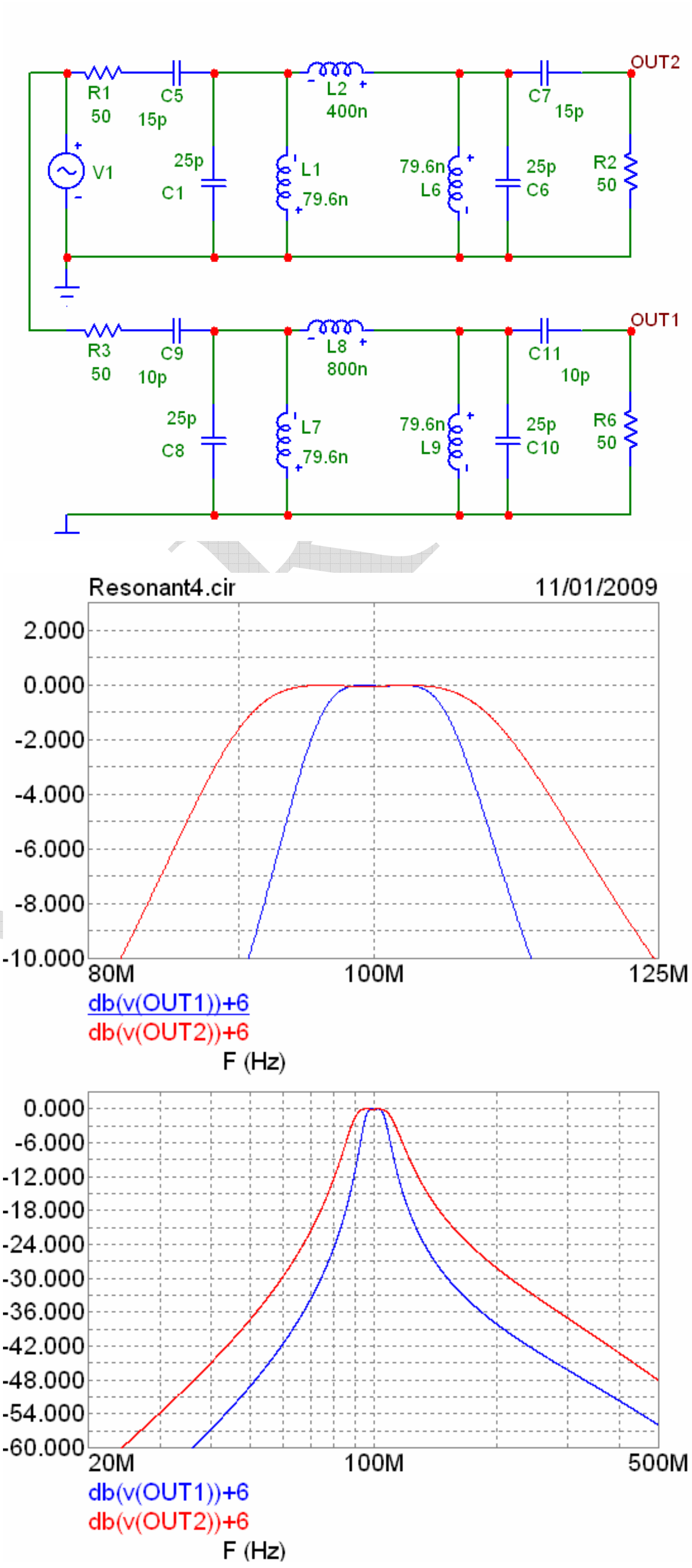


Fig.12

The blue trace is the original filter, the red one is the OUT2 of the drawing.

We can verify another parameter in order to check the proper fitting of the filter we are designing. This parameter is called in many ways: return loss, VSWR, source and load match, etc.

Return loss is an important indicator that the filter is matched to the generator. An alternate indicator is the insertion loss, that for a perfectly matched lossless filter must be zero.
But when we will use real component with its own loss it will be difficult to understand if the insertion loss of the filter will be generated by mismatch or by component loss.
Monitoring the return loss, we can distinguish between the two types of loss.
In Fig.13 the return loss in dB of the filter from Fig. 12 is simulated.

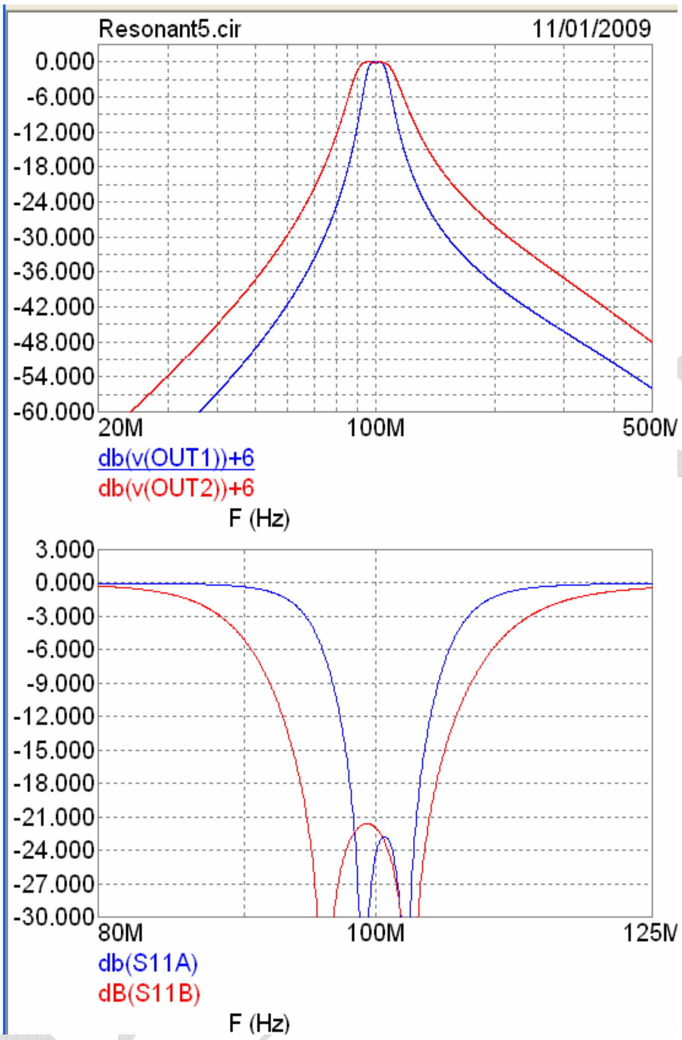


Fig.13

The return loss of the filter in the pass band is in excess of 20 dB, a good result for a lossless filter.
The loss in the filter generated by a return loss of 20 dB is less then 0,05 dB. As we will see later the loss induced by the parasitic of components is greater at least one order of magnitude.

* Q of the filter and Q of components

A common subject of misunderstanding is the definition of "Q" or quality factor, that is used both for the filter bandwidth and for the quality of components. When Q is used for filters we can use the term "loaded Q"; for components the quality factor is expressed simply as Q or "unloaded Q".

When Q is used for filters with a single resonator the relationship is Q= 1/BW, as explained before.
If the filter has many resonators the relationship became complex, because normally the various resonator do not exhibit the same loaded Q in the circuit. Usually in this case the loaded Q of the filter is referred as the highest Q present in the filter. This no longer gave information on the BW of the filter, but is useful for evaluating the minimum Q requested by the components used in the filter.

And many times the misunderstanding is here!
To build a filter with a loaded Q of 10 you don't need inductors an capacitors with an unloaded Q of 10, **but as a rule of thumb, you need at least a Q of 100**.

Many times people misunderstand loaded and unloaded Q and try to assembly a filter with components totally unsuited for that application.

* physical components

In a third of a century I have heard every type of story about component suited or not suited for filter design.
One example for all: it isn't possible to use ceramic capacitors in audio filters, they are not stable and they "sound" (makes acoustical noise) when used at relatively high power.
Evidently upholders of this theory makes their experiments and found that some capacitors are not stable with temperature, applied voltage, and aging. Moreover some of this capacitors exhibit a piezoelectric effect, and they "sound" when music is applied. Poor quality ceramics as Z5U and Y5V exhibit astonishing tolerances, and some of them has piezoelectric effect.

But on the other way exist ceramic materials that has very tight tolerance (5% to 1%), zero temperature, voltage and aging drift. Materials as COG or NPO has these characteristics and are perfectly suited for audio frequency filters.

My position is to avoid the suggestions of people who made a work or two in the field, but they are not expert. If you makes a design or two and find a solution, you are probably going to consider your circuit as the only possible one. But since

you aren't an expert in this field, maybe a dozen of circuit with best performances exist.
As before, my suggestion is to learn about the design you are going to made from many and different sources, and after verify experimentally if the knowledge is true.

A customer requested me to review a design of a system that has production yield problems. I found that each stage was buffered by an additional one, being each stage an operational amplifier or an RF amplifier, each one was followed by an additional buffering stage. In 70% of location these buffers were not only unnecessary but they spread the production parameters and gave a poor yield in production.
I asked to the designer why he used that design methodology and his answer was that his chief engineer many years ago teach him to design in that manner because " a redundancy in design is always advisable, so you have room for fixing bugs and problems".
I believe this is the perfect example to cite for indicating a person which do not work with his head!

* Capacitors:

Radio frequency filters employed in the majority of applications use multilayer ceramic capacitors of the SMD type; the dielectric used is usually COG or NPO which exhibit a zero drift over temperature.
Only in high power applications porcelain, teflon and mica capacitors are employed.
In microwaves circuits single layer types are used.

At lower frequency you can use ceramic capacitor, polypropylene film capacitors (especially at high voltages), mica capacitor in special cases (they are very expensive). The use of polyester film capacitors isn't recommended, since they are not stable with temperature.

Manufacturers in their WEB sites have useful tools to investigate the properties of their capacitors. Spice models are available also for many simulators brand. However in the first phase I'm suggesting you to avoid this models and to check manually the behaviour of capacitors in the environment. Manual insertion of the parasitic element in the simulator will improve your knowledge of components.
Of course when you will be skilled this approach will be no longer necessary, but if you gain this component knowledge, you will became able to see errors or problems in the models when they don't fit your application.

You can download SW evaluating the performance of capacitors from the following sites:

1) Murata: Mcsil
http://www.murata.com/designlib/mcsil/index.html a great SW for understanding the capacitor properties. There you can see series resistance, frequency of auto-resonance, Q of capacitors and so on. It contains also parameters for small SMD inductors

2) Kemet: Kemet spice
http://www.kemet.com/kemet/web/homepage/kechome.nsf/Weben/kemsoft#fit it contains data for ceramic and tantalum capacitors.

* Inductors:

The greatest part of inductor used today in filters are SMD wire-wounded. Monolithic type usually has lower Q and rarely are used in filters, where to keep losses low, maximum Q is needed.
Thru- hole inductors are generally used only for power applications.

Ferrite or air core Inductors?
Air inductor are used for higher frequency, where ferrite cores had unacceptable loss. On the contrary at moderate frequency (up to 10-50 MHz) the ferrite core could increase the Q of the inductor.

One of the best WEB site for information on inductor is Coilcraft:
http://www.coilcraft.com/prod_rf.cfm where you can find a lot of data on Q, inductance vs. frequency and frequency of auto-resonance of inductors.

Web sites represented here are only suggestion to consult, the directory would be very long, and the choose to present only these is personally.

* Other passive component used in filters:

1)Quartz crystal, SAW resonator and ceramic resonator are used in narrowband filters, the design of these filters are usually monolithic and they are outside the purpose of this tutorial and will not be treated here. However in the bibliography a book on this topic is listed.

2) strip-line, coaxial resonator and other distributed components: these components will be treated in the second part of this tutorial, since the design of filter with this class of component is quite different.

Starting the design with real components.

Using all the tools learned in the previous section, we can start the design of a practical filter. The example is a realization of a bandpass filter used to separate the FM audio from the video signal. The FM audio channels are from 6 to 7.5 MHz, where the maximum allowable loss is 3 dB, and the bandpass ripple allowed is +/-2 dB maximum. An attenuation of at least 50 dB is requested from DC to 4 MHz (the BW of the video signal), the attenuation requested in the upper band from 10 to 100 MHz is at least 15 dB, since strong signals are not present here, there is only to reject noise and some spurious frequencies.

Specifications of design also request that the filter will be produced with all SMD components and could be assembled with no post-production tuning.

Starting from the design of Fig.6 the “high pass” response (out 3) has been chosen, in order to increase cutoff at low frequency.

To obtain the required attenuation at least three resonator must be used. This is a rule of thumb, dictated by experience, but it is quite easy for the experimenter to start the design with a two poles filter and verifying that specifications are not met. (I believe a single pole filter, checking Fig.6, is discarded immediately)

The filter with ideal components has been simulated in Fig.14. Note that components with standard value has been used, in order to simplify the following steps.

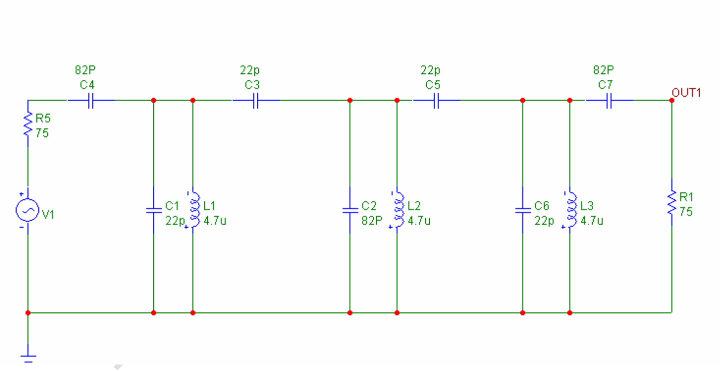


Fig.14

The input and output impedances are 75Ω, the value of components are all standardized, and luckily only two values of capacitors are needed, and the three inductors has the same value.

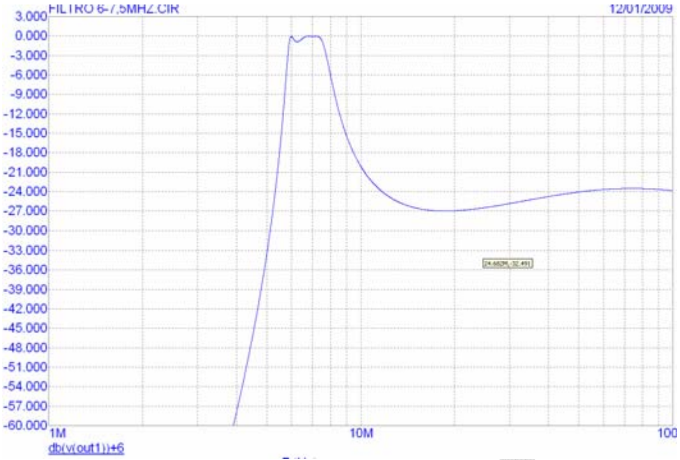


Fig.15

Response appear in fig 15, it can be seen that some ripple in bandwidth already exist, indicating that the design is not perfect, but the use of standard value in components is highly regarded in production.

Now we can start to insert the value of parasitic in components, since the design is at few MHz, we can insert the series resistance only, because at these frequencies self inductance is negligible. Capacitors has been chosen as 0603 size, COG ceramic with 2% tolerance, and inductors are in 1008 size and 5% tolerance. In Fig. 16 the parasitic resistances are indicated in red.

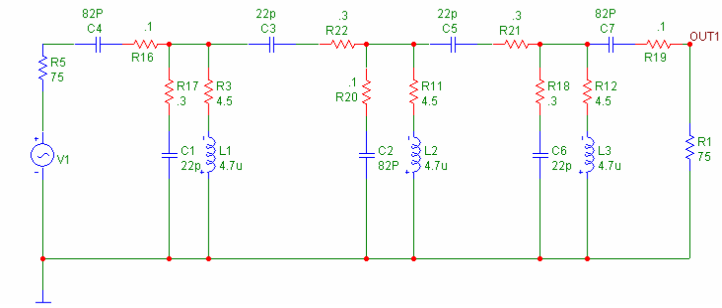


Fig.16

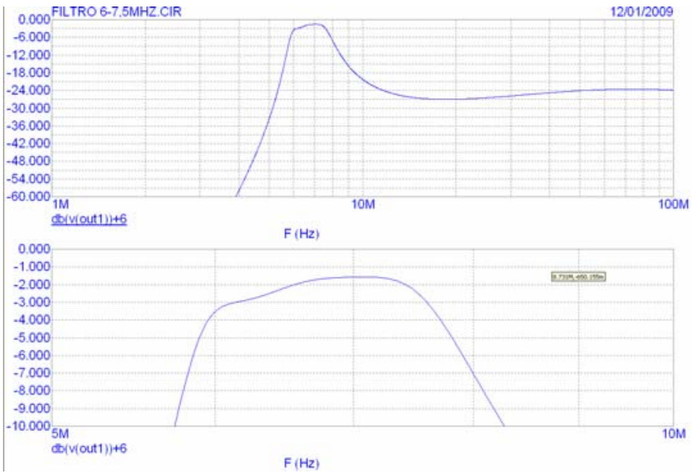


Fig.17

It can be seen that coil losses dominate all over the other losses. The response of the filter suffer from such a high losses, see Fig.17. It is important to decrease the loss of inductors, maybe using a larger case model.

Running the Monte-Carlo simulation at Fig.18 (100 iterations, worst case distribution) it can be seen that we are outside specification for BW loss at 6 MHz: some iterations show a 7.5 dB loss instead of 3+2 dB maximum allowed.

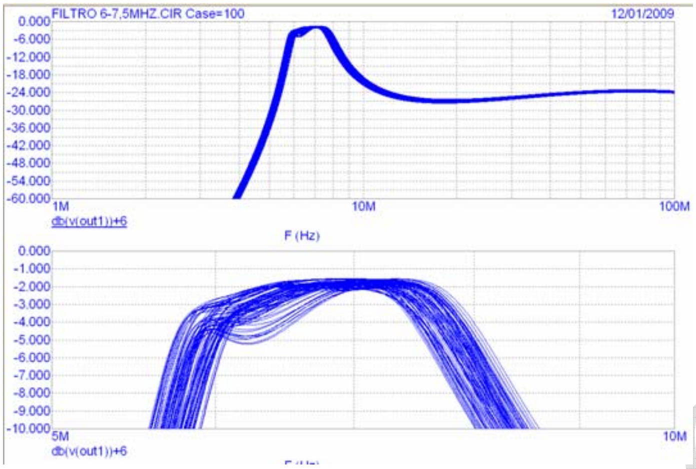


Fig.18

We made two more simulation with Monte-Carlo analysis, the first (Fig.19) with the same inductor but with a 2% tolerance, the second (Fig.20) with a 1210 size inductor which exhibit a 2Ω series resistance.

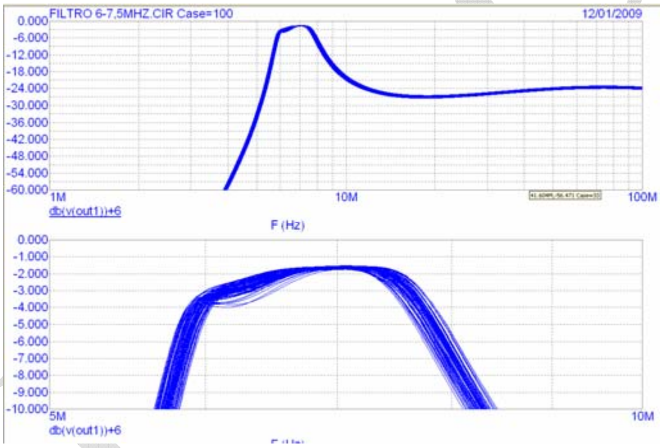


Fig.19

It can be seen that the best solution is to employ a 2% tolerance inductor, in this way we met the specification, while in Fig 20 you can see that few iterations are outside the specification.
It is to be noted however that we are using in the simulation the worst case distribution, using gaussian distribution instead, which is the normal spread in components (or at least it should be).

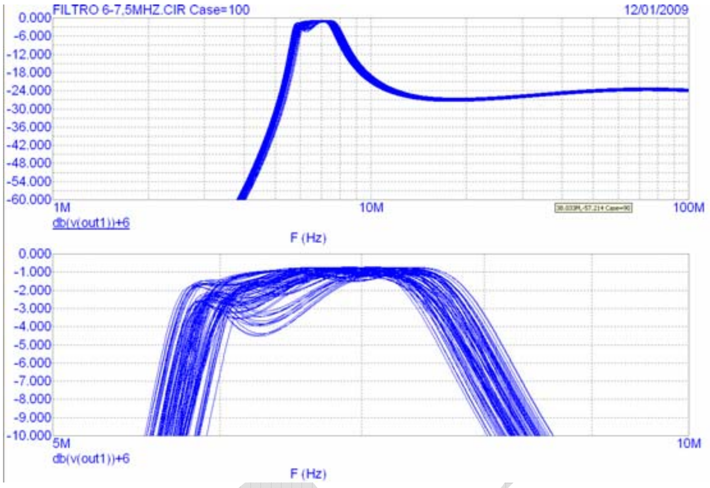


Fig.20

We can see in Fig.21 that a production lot of even 1000 pieces with Gaussian distribution meet the specification even if the worst case distribution is slightly outside specification (Fig.20)

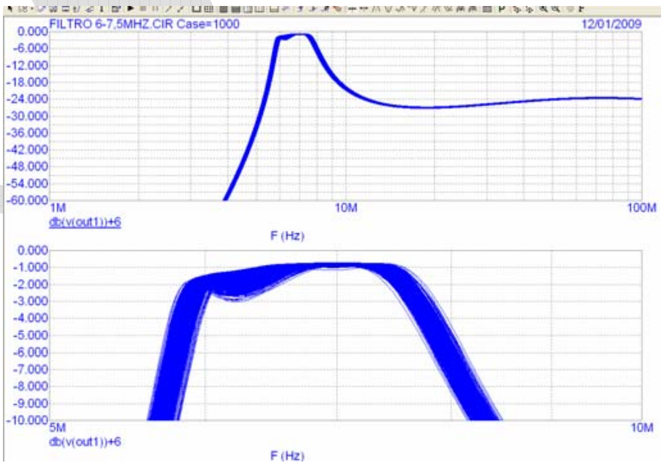


Fig.21

So the filter goes into production without problems with the original inductors, but with a 2% tolerance.
In that case I did not win a Solemn Encomium as in the previous case, but only a coffee from a colleague that bet with me that it was impossible to manufacture such a filter without tuning. I won only a coffee, but it was an Italian coffee, very strong and very thick!

Next step we will increase the operating frequency and will start to introduce the parasitic of the PCB and the surrounding environment.

Till now we have only inserted the parasitic resistance of capacitors and inductors. But capacitors exhibit also parasitic inductance, which is, for ceramic capacitors mainly function of the size.

* As a rule of thumb here are the parasitic inductance for capacitors in the following size:

size	0201	0402	0603	0805	1206	1812
Induct.	.4nH	.5 n	.63 n	.55 n	.74 n	.9 n

These value can change varying the value of the capacitor and its voltage, however in the table are listed the mean values and typically the difference is less than a +/- 15%. If you need a more precise value data are available in the Web sites of various manufacturer.

Inductors: of course they exhibit parasitic capacitance, but the value is not so case-dependent. The value of capacitance strongly depend on manufacturing technology, number of turns and inductor value, core material and so on.
The only way to have reliable data is to verify in the data sheet the resonating frequency and calculate the capacity from this data.
Usually if the frequency of resonance is 3-5 times higher than the operating one, this capacitance can be neglected.

Printed circuits: three main sources of parasitic exist on a PCB, first the capacitances of PADs which are functions of PCB thickness and material. Second vias to GND and vias to other layers of the PCB which behave like inductors. Third the tracks that connect the various component also behave like inductors.
In the following table a list of typical values for parasitic capacitances and inductances are shown

Thickness to GND Layer	PAD Capacity Epoxy	Via ∅ .5 mm	Via ∅ 1 mm	Track .5 mm 1cm long
.4mm	.7 pF	.25nH	.2nH	2.5nH
.8mm	.3 pF	.5nH	.4nH	3.3nH
1.2mm	.2 pF	.8nH	.6nH	3.6nH
1.6mm	.14pF	1.2nH	.9nH	4.1nH

A capacitance per square millimetre is indicated in the table, to obtain the value of capacitance for a given PAD you have to calculate the actual area. Please note that the thickness do not necessary correspond to the thickness of the PCB, in the case of a multilayer PCB the thickness correspond to the distance from the populated layer (normally the top layer) to the ground plane (normally the inner layer 1). The inductance of the track is indicated for a length of one centimetre, over a ground plane; it is measured on a 50Ω impedance.

* If in a certain point you have a 100 pF to GND you can ignore the parasitic capacitance of say 0.25 pF, but if you are operating at 1 GHz, the impedance in that point will be 1.6 Ω and if you have a connection to GND with a via hole its parasitic inductance of 1 nH will exhibit a reactance of 6.28 Ω. Of course your circuit will not operate properly, so you will have to choose a different grounding mechanism, or change the working impedance of 1.6 Ω, or even better do both the remedies.

Till now the filter presented here are all poles type, they have zero response at DC or at infinity. A different class of filters exist that places zero responses at some frequencies of interest, they are called elliptic filters.

In the next example we will insert not only parasitic but we will use an elliptic filter.

One of our customers gave us specifications of an input filter for a receiver operating from 250 to 300 MHz. The loss of the filter should be less than 1dB in order to keep the receiver sensitive. Cutoff at low frequencies should be at least 25 db at 100 MHz and 50 dB at 30 MHz.
The local oscillator was beating in the upper mode and the fist IF was at 110.6 MHz. Therefore the image was 250 + 2 X IF = 471,2 MHz as minimum and 300 + 2 X IF = 521,2 MHz. The required cutoff in the image band was 45 dB, since the tuned filter between the low noise amplifier and the mixer cut the rest. The receiver must have a maximum height of 7 mm and hence coils must be miniaturized.
A three resonators filter was chosen, and the three resonant inductors were chosen of the Mini-spring air core inductor, from Coilcraft, which exhibit a Q of 150 at the frequency of operation. The inductors has a nominal value of 18.5 nH, but the value inserted in the simulator is 20,5 nH to keep count of parasitic inductance.
All the parasitic resistances are indicated in red.
The parasitic capacitances are not inserted in the simulation since all them are referred to ground and hence can be inserted in the value of the three resonating capacitors C1,C2 and C3.

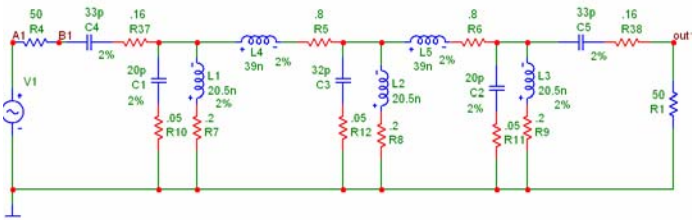


Fig.22

The tolerances of components are set to 2%, except for the central resonator, where tolerances are set to zero (the capacitor C3 will be tuned), otherwise the spread of parameters will be too high. In Fig 23a appear the analysis of the filter, where you can see that the loss in the passband meet the specification even with Monte-Carlo analysis (500 run) Fig 23b.

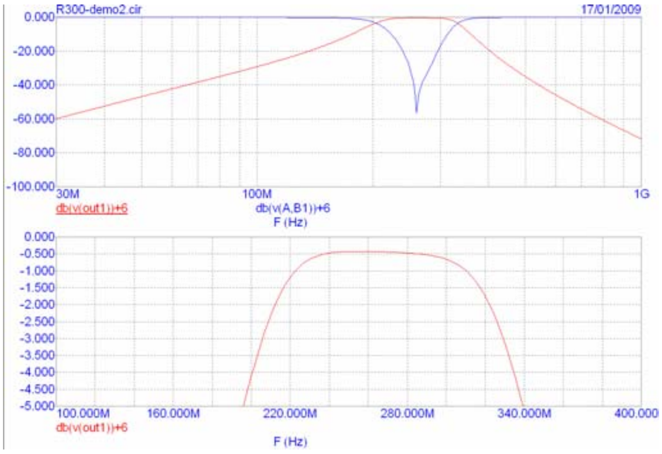


Fig. 23a

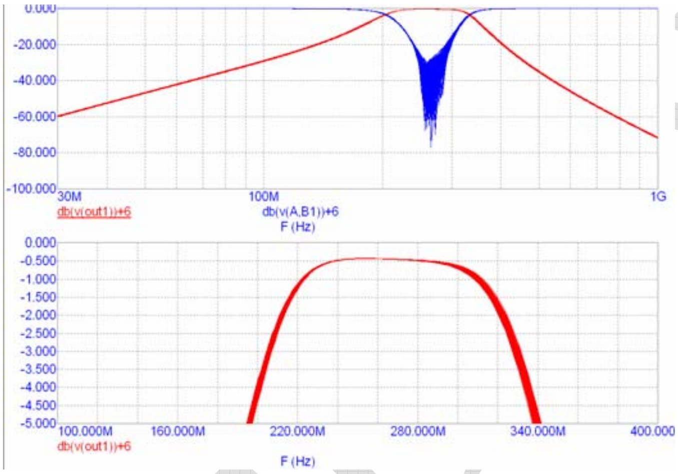


Fig.23b

At low frequencies the specification is also met, but in the image band the attenuation is 15 dB out of specification.

The solution, instead of redesign the filter with a higher resonator number, is to replace the two coupling inductors L4 and L5 with two parallel resonant circuits, in this case the filter became elliptical and two zeros are placed near the image band, increasing the attenuation. In Fig.24 you can see the modified circuit, with the two capacitors added. The capacitor C8 has been decreased to retune the central resonator.

The two inductors L9 and L10 has been decreased in value to compensate for the fictitious increasing of the apparent inductance in the passband caused by the two 2.4 pF capacitors.

In Fig. 25 you can see the simulation results.

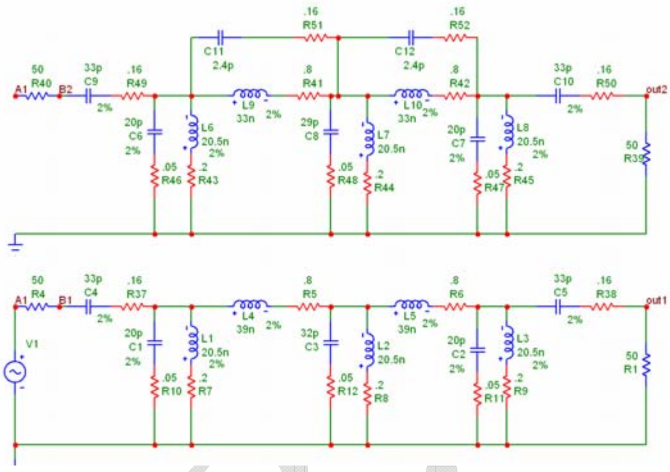


Fig 24

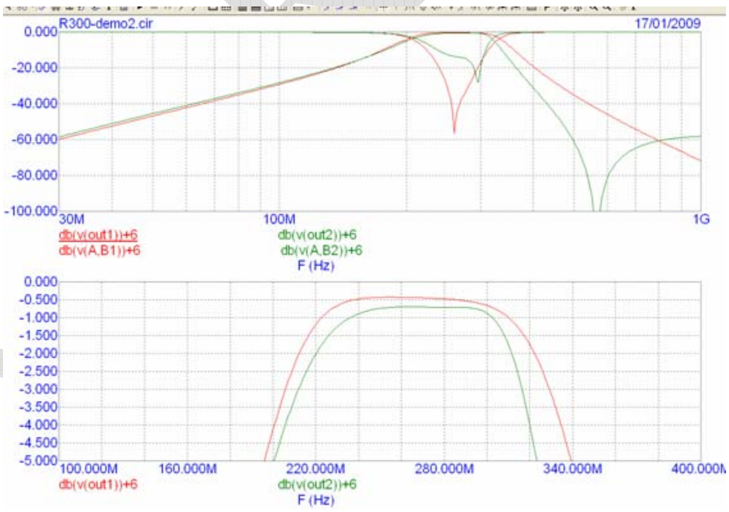


Fig.25

It appears that the cutoff at higher frequencies has greatly increased (green trace versus red trace). Now the specification of 45 dB cutoff at frequencies over 471.2 MHz is easily met. The insertion loss instead is slightly increased, but it is still inside specification.

Many people believe my way to design filter is a loss of time. Usually you can synthesize the filter directly with a program or from a data table.

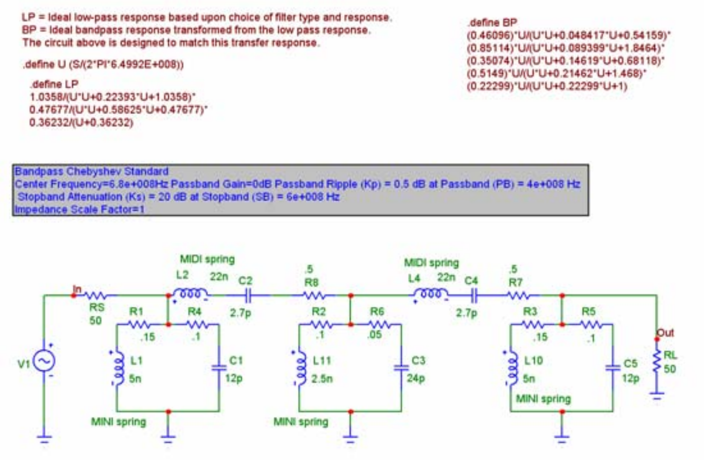
I don't use only this method to design filters, the approach is useful for teaching people how to learn about filters, and is the only method useful when the design needs fall outside the class of filters available in the programs of synthesis or in the data tables.

In the following 2 example we will synthesize two filters, one perfectly fitting real components, the other impossible to manufacture with that design.

If you will understand immediately why the second design couldn't be manufactured, I hope this tutorial has increased your knowledge.

The first design is a filter with a bandpass of 480 to 880 MHz with a maximum of 3dB loss and rejection of at least 20 db from 100 MHz to both band edges.

The spice simulator (Micro cap) directly made the synthesis, (Fig. 26) the ripple in BW is set to 0,5 dB as a design parameter.



```
LP = Ideal low-pass response based upon choice of filter type and response.
BP = Ideal bandpass response transformed from the low pass response.
The circuit above is designed to match this transfer response.

.define U (S/(2*PI*6.4992E+008))

 .define LP
 1.0358/(U*U+0.22393*U+1.0358)*
 0.47677/(U*U+0.58625*U+0.47677)*
 0.36232/(U+0.36232)

.define BP
(0.46096)*U/(U*U+0.048417*U+0.54159)*
(0.85114)*U/(U*U+0.089399*U+1.8464)*
(0.35074)*U/(U*U+0.14619*U+0.68118)*
(0.5149)*U/(U*U+0.21462*U+1.468)*
(0.22299)*U/(U*U+0.22299*U+1)
```

```
Bandpass Chebyshev Standard
Center Frequency=6.8e+008Hz Passband Gain=0dB Passband Ripple (Kp) = 0.5 dB at Passband (PB) = 4e+008 Hz
 Stopband Attenuation (Ks) = 20 dB at Stopband (SB) = 6e+008 Hz
Impedance Scale Factor=1
```

Fig.26

The design is reasonable, parallel inductors are of acceptable values, even if great care must be taken in the central resonator, to keep parasitic low, due to the 2,5 nH inductor. Series resonators also has acceptable values and the filter works with real value components: Mini-spring high Q inductors and low loss ceramic capacitors. Tolerances are 2% for inductors, 1% for 12 and 24 pF capacitors and 5% for 2.7 pF type.

The simulation appears in Fig.27.

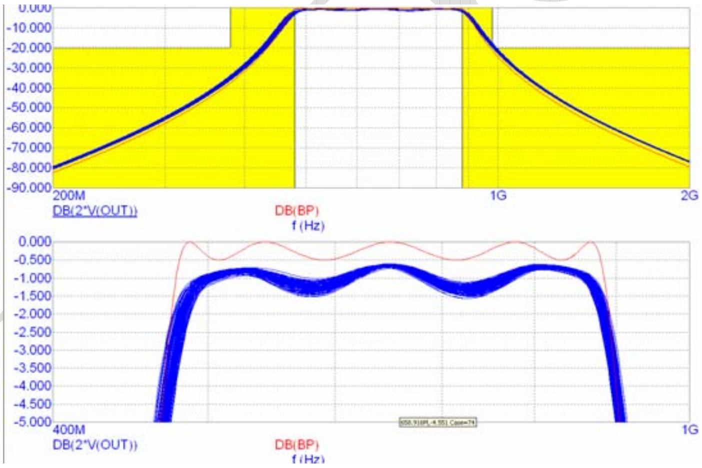


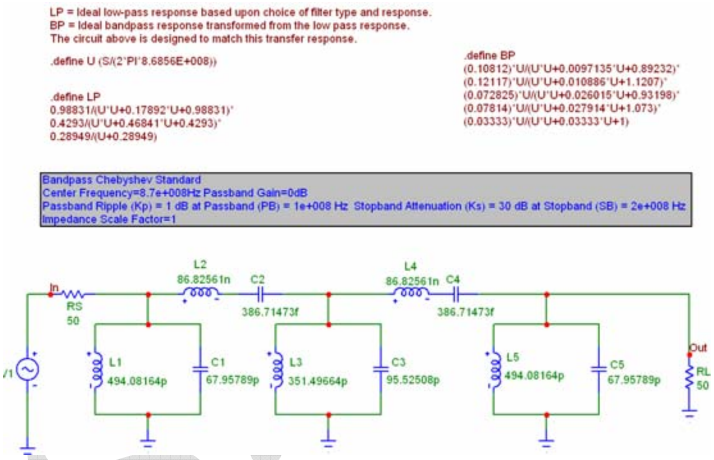
Fig.27

Red trace is the theoretical simulation and the blue one is a Monte-Carlo analysis of 100 runs. It can be seen that at the band edge at 480 MHz the trace is slightly outside the specification, but this will be correct in the physical sample.

We can see that in this design the synthesis tool work properly and the result is directly applicable to a prototype.

Now instead we will design an 870 MHz filter with a 100 MHz BW and a 3 dB maximum loss. Rejection of at least 25 dB is requested from 50 MHz to both edges.

The design is in Fig 28 and the ideal response in Fig.29.



```
LP = Ideal low-pass response based upon choice of filter type and response.
BP = Ideal bandpass response transformed from the low pass response.
The circuit above is designed to match this transfer response.

.define U (S/(2*PI*8.6856E+008))

.define LP
0.98831/(U*U+0.17892*U+0.98831)*
0.4293/(U*U+0.46841*U+0.4293)*
0.28949/(U+0.28949)

.define BP
(0.10812)*U/(U*U+0.0097135*U+0.89232)*
(0.12117)*U/(U*U+0.010886*U+1.1207)*
(0.072825)*U/(U*U+0.026015*U+0.93198)*
(0.07814)*U/(U*U+0.027914*U+1.073)*
(0.03333)*U/(U*U+0.03333*U+1)
```

```
Bandpass Chebyshev Standard
Center Frequency=8.7e+008Hz Passband Gain=0dB
Passband Ripple (Kp) = 1 dB at Passband (PB) = 1e+008 Hz  Stopband Attenuation (Ks) = 30 dB at Stopband (SB) = 2e+008 Hz
Impedance Scale Factor=1
```

Fig.28

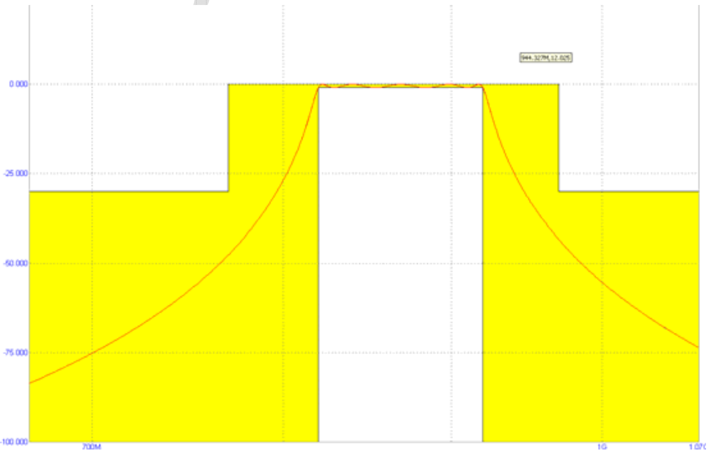


Fig.29

It can be seen that the simulated values are perfect, but the value of the components used in the filter are hard or impossible at all to find in the physical word. The three resonance inductors has sub-nH values (0.494 and 0.351 nH) even if you find them with the right value, it seems difficult to place them in a real word PCB where the mere distance between two components has nearly the same inductance value. The same is true for capacitor C3, which has a reactance of 1,7 Ω; its self inductance generate a reactance grater than this value. Moreover the series capacitors C2 and C4 has a value of 0.386 pF; the nearest commercial value available is 0.39 pF, the tightest available tolerance is +/- 0.1pF or a +/- 25%! Again the capacitance versus ground of the two pads in the hot points between L2-C2 and L4-C4 exhibit a parasitic capacitance nearly the same value of C2 and C4. It seems

clear that in this case the circuit synthesis is useless.
With this last simulation the first section of this tutorial ends.
The second part will threats of filters with distributed elements.

Bibliography

1) MICROWAVE FILTERS, IMPEDANCE-MATCHING NETWORKS, AND COUPLING STRUCTURES
G. Matthaei, L.Young, E.M.T. Jones
1st edition 1964 ISBN:0-89006-0991.
A Historical book, but still actual today

2) HANDBOOK OF FILTER SINTHESIS
Anatol I. Zverev ISBN:0-471-74942-7
1st edition 1967
A classical book on filter, full of tables for filter design

3) FILTERING IN THE TIME AND FREQUENCY DOMAIN A.I.Zverev, H.J. Blinchikoff
1st edition 1976 ISBN: 1-884932-17-7
Another classical book, perfectly useful today, you couldn’t miss it in your library

4) INTRODUCTION TO THE THEORY AND DESIGN OF ACTIVE FILTERS
L.P.Huelsman, P.E.Allen ISBN:0-07-030854-3
1st edition 1980
Another dated book, but still actual.

5) ELECTRONIC FILTER DESIGN HANDBOOK
A.B.Williams, F.J.Taylor ISBN:0-07-070434-1
1st edition 1981
A practical book, containing a lot of tables.

6) CRYSTAL FILTERS R.G.Kinsman
1st edition 1987 ISBN:0-471-88478-2
A useful book for the engineers interested in crystal filters design.

7) COMPUTER-AIDED CIRCUIT ANALYSIS USING SPICE, W.Banzhaf ISBN:0-13-162579-9
1st edition 1989
A useful book about Spice simulation

8) MICROWAVE CIRCUIT MODELING USING ELECTROMAGNETIC FIELD SIMULATION
D.G.Swanson, W.J.R.Hoefer
1st edition 2003 ISBN:1-58053-308-6
A great book on electromagnetic simulation.

9) STRIPLINE CIRCUIT DESIGN Harlan Hove, Jr.
1st edition 1974 ISBN:0-89006-020-7
A good book on stripline circuits and components.

Disclaimer