EE426 RF Circuits Design Techniques

Fall 22 Examination & Corrections

General information and system level specifications

We are designing a super-heterodyne receiver in the license-free, 26MHz wide, 915MHz-centered ISM (industrial, scientific, and medical) band with an IF frequency of 100MHz.

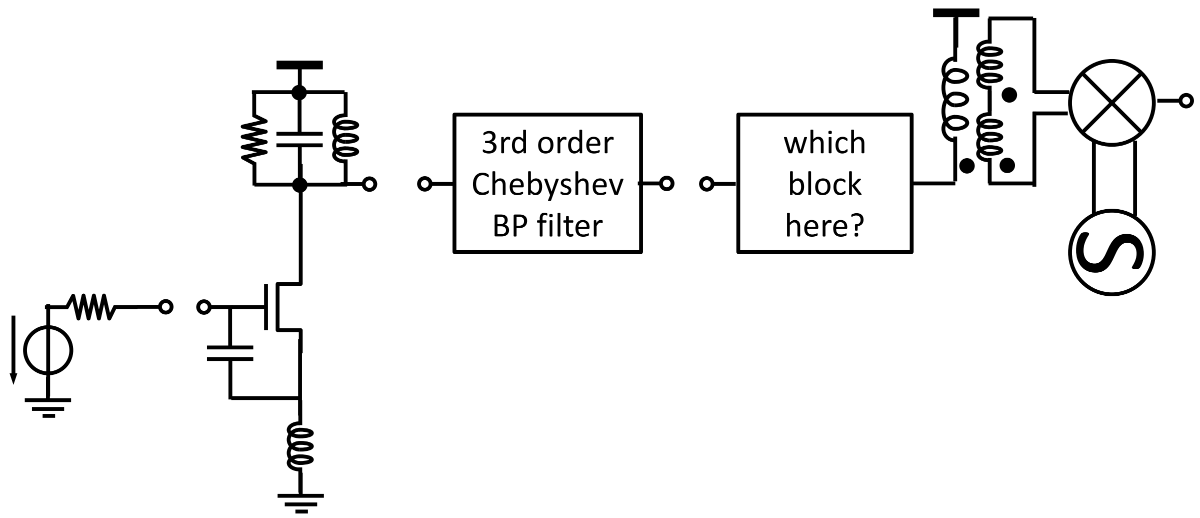
The LNA should be matched to the 50Ω antenna and be designed for maximum gain using the topology sketched below.

In order to implement image rejection, we have to design an LC BP 3rd order filter centered at 915MHz. The Q and input/output impedances (Rs=RL) should be chosen according to what is feasible according to available components values (see below).

The block following the BP filter should provide additional gain and be input impedance matched to implement the filter output impedance.

At the output of the preceding stage, we have an ideal transformer with a turn ratio of 1, coupling of 100% that is used to generate a differential RF input to the mixer (no design, nor consideration to be given for that block). The mixer should be of the double-balanced type (as designed during the class).

A 100MHz filter with a Q of 100 will perform channel selection (not shown). We assume an off the shelf component and do not consider interfacing it.



Question 1) Frequency planning (a few simple questions to warm-up!)

In our configuration, there are very strong interferers centered at 715 MHz (some strong LTE bands)

a) Calculate the Q related to the ISM band, why is this value important?

Q=fo/fBW=915/26=35

The Q or equivalently fractional BW (1/Q) of the desired band has strong impact on a transceiver system (TX & RX). For resilience to interferers we would need to sharply reject any frequency outside our bandwidth (high selectivity). However with a Q of 35 we would already attenuate the lower and upper channels significantly (3dB) while we are designing a filter with only 1dB ripple in the passband. Furthermore if you cascade several blocks with each a Q of 35, you get 6dB, 9dB attenuation for 2, 3 blocks at those frequencies near the band edge. Consequently, the sensitivity is severely degraded due to the increased NF rendering those channels unusable wasting frequency resources. This reasoning tells us that we should always take some margin!

Furthermore, we have to consider the component spread and the fact that they take discrete values if external parts are used. The resonance frequency is given by

$$\omega_O = \sqrt{\frac{1}{LC}}$$

We may thus write the following equation to characterize the variation of ωo

$$\frac{\Delta\omega}{\omega_O} = \frac{1}{2}\left(\frac{\Delta L}{L} + \frac{\Delta C}{C}\right)$$

with respect to the relative variation of L and C. The first term above is 1/Q ! and we should compare it with our fractional bandwidth of ±1.5% (±13MHz/915MHz).

Our coil components are rated at 5% for the small ones and down to 3 or 2% for the bigger values. We may assume similar value for capacitors. With ±5% for both components, we end up with ±5% frequency deviation. Our filter would thus be centered at offsets as large as ±45MHz! And our desired BW would fall out of band!

Adding the spread to the fractional BW, we should shoot for a fractional filtering BW of ±6.5%, thus a Q of ~8. For components at ±2%, we could go up to Q~15.

An alternative is to find a way to tune the resonance of our front-end with varactors as a function of the wanted channel (1MHz) inside the 26MHz band, but this is not that obvious!

b) Calculate the LO (local oscillator) frequency range and the image band

The IF is 100MHz, thus the LO should have an offset of either + or -100MHz from each channel; the image band is thus at 200MHz offset for each channel

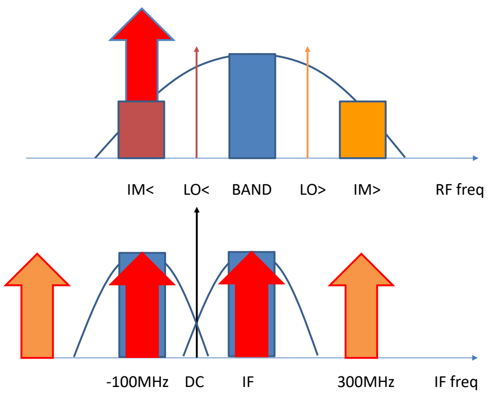
Case 1: LO centered at 815MHz +-13MHz, image band is at 715+-13MHz

Case 2: LO centered at 1015MHz+-13MHz, image band is at 1115MHz+-13MHz

Since we have strong 715MHz interferers, we pick high side injection thus the LO at 1015MHz

c) Have you made the best choice choosing the LO frequency? Why ? Illustrate this graphically on a chart with the X-axis being the frequency

The image reject filter will be symmetrical thus no effect, but the filtering at the mixer output will reject better 715MHz since it will be located at 200MHz from the center frequency providing thus additional rejection of the strong interferers. This is shown below, in blue the RF band with the two possible LOs and corresponding image bands. In red the interferers. At the IF frequency, the interferers with low side injection (LO<) are only attenuated by the RF filter selectivity and superimposed to our signal band. With high side injection (LO>), they reside at 200MHz from our wanted band and hence will be further attenuated by the IF filter (interferers in orange before being filter by the IF).



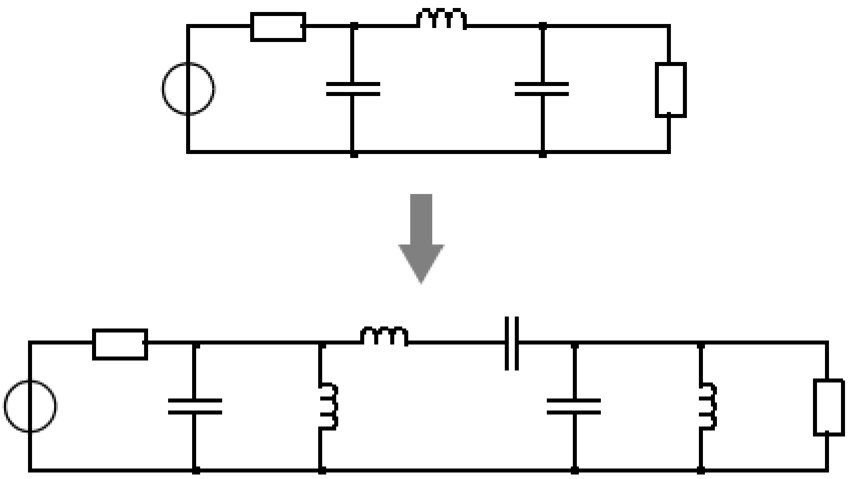
IM<
LO<
BAND
LO>
IM>
RF freq
-100MHz
DC
IF
300MHz
IF freq

Question 2) Image reject filter centered at 915MHz

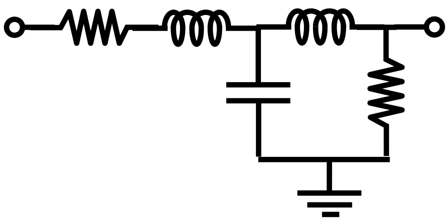
Using the proposed coilcraft components design a 3rd order bandpass filter with a 1dB ripple that will be used as image rejection in combination with the selectivity of our LNA.

a) we have already calculated the normalized prototype LP values in an exercise (l=.9941, c=2.0236) for the Π network with a single inductor or alternatively you may use the T network with two series inductors between input/output and a single cap in between (in which case l and c values given above should be swapped)

This is the normalized LP 3rd order Π prototype and how it is transformed in a BP



The T topology has 2 coils in series between in and out and one shunt capacitor to ground. It is transformed similarly into a BP.



After the BP transformation we thus have either one series resonating element (Π) or two (T)

Thus for Π l=.9941, c=2.0236 for both caps, for T l=2.0236 for both L and c=.9941

b) calculate the BP filter components in a parametric way using fo, Q and the input/output impedance which have to be equal in this filter design

Many of you forgot to include Rs in the equations. Compared to the prototype, all Z have to be multiplied by Rs hence C∝1/Rs, L∝Rs. Here is an intuitive reasoning to convince you: if all components in a transfer function are multiplited by a constant, nothing changes !

When you are asked to define the components parametrically, do not jump in with Q=35 now, because you clearly bias and invalidate your results!

Π network (l=.9941, c=2.0236)

Cp1 and Cp3 are in parallel with LP1 and LP3, the components in the two branches are equal and given by

$$Cp1=Cp3=c\cdot Q/(2\pi fo)/Rs,\ Lp1=Lp3=1/c/Q/(2\pi fo)\cdot Rs$$

Ls2 and Cs2 the series components are given by

$$Ls2=l\cdot Q/(2\pi fo)\cdot Rs,\ Cs2=1/l/Q/(2\pi fo)/Rs$$

T network (l=2.0236 for both L and c=.9941)

$$Ls1=Ls3=l\cdot Q/(2\pi fo)\cdot Rs,\ Cs1=Cs3=1/l/Q/(2\pi fo)/Rs$$

$$Cp2=c\cdot Q/(2\pi fo)/Rs,\ Lp2=1/c/Q/(2\pi fo)\cdot Rs$$

c) what limits the achievable Q of the filters, please justify with a formula

It is mostly the ratio of the coil components as discussed in the course (only 1 person mentioned it ☹) , then their spread in frequency (see 1a), then their own Q factor (mostly cited answer in this exam but considering we have Qs in the 50, that's not the first limit). The ratio of the L components is given by

$$\Pi \text{ network: } Ls2/Lp1 = l\cdot Q/(2\pi fo)\cdot Rs/(1/c/Q/(2\pi fo)\cdot Rs)= l\cdot c\cdot Q^2$$

$$\text{T network: } Ls1/Lp2 = l\cdot c\cdot Q^2$$

The lc product is ~2, the component range is from 0.8nH to 120nH, with a Q of 8, our ratio is 128. This is somewhat the upper limit. Let's pick 0.8nH for Lmin and ~100nH (or 0.9/120). Note that this is conservative with our ±5% component spread!

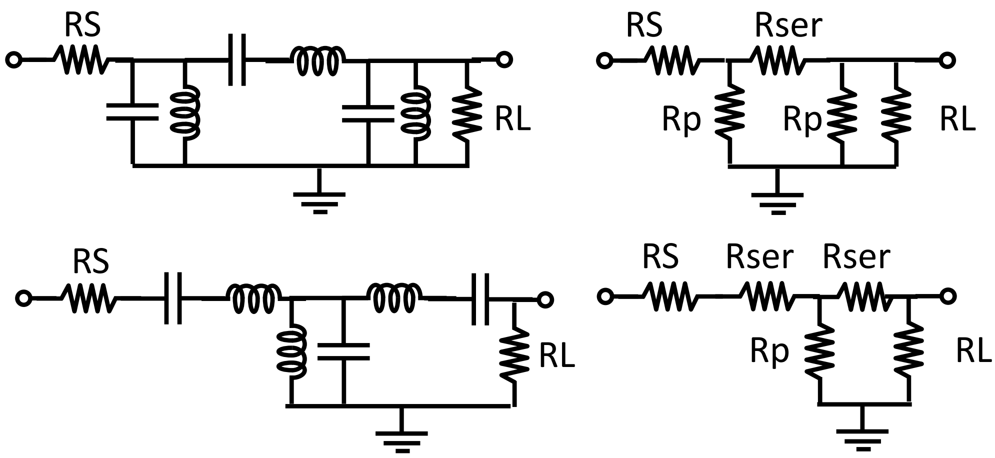
With a Q of 35 (popular choice in the exam but a nice trap !, you require a component ratio of 2500 !!).

d) compare the T and Π topologies in terms of inductor loss (Q is given in the inductor table at 900MHz) and select the best one

See below the sketch of the lossy filter at resonance on the right. The big L are in series thus intuitively, the T is worst since we have twice the equivalent resistor Rser = $\omega L_{Max}/Q$ in series (this is a sufficient answer). But on the other hand, the shunt elements have a much lower impedance and Rp = $\omega L_{Min}\cdot Q$, which is seen at resonance is not that high. Note indeed that we have 2 such elements for the Π topology vs 1 for the T one. So we will get a final answer later below!

$$Rser = \omega L_{Max}/Q;$$

$$Rp = \omega L_{Min}\cdot Q$$



e) select the input/output resistor and Q value so that the filter could be designed with the available components

The shunt and series component values are the same for Π and T but the input output resistance is different since l and c values are swapped for both topologies.

For Π network

Q of 8, Rs=RL of 75 Ω

Cp1=Cp3=37.8pF, Lp1=Lp3=0.8nH

Ls2=104nH, Cs2=292fF

For T network

Q of 8, Rs=RL=37 Ω

Cp2=37.8pF, Lp2=0.8nH

Ls1=Ls3=104nH, Cs1=Cs3=292fF

We may now calculate the Rp and Rser values

$R_{ser} = \omega L_{Max} / Q = 11.3\ \Omega$; $R_p = \omega L_{Min} \cdot Q = 248.4\ \Omega$

which again are the same for both topologies.

Our intuition is reinforced since for the T network, we would have 22.6 Ω in between 2x 37 Ω input output resistors but we have one vs two ~250 Ω shunt elements! We will get the final answer in the next section.

f) with ideal components, our filter would have a loss of 6dB - and a 1dB ripple - resulting from matched input/output, determine at the resonance frequency the additional insertion loss when considering the effective Q of your inductors at that frequency.

At resonance, we are left with only resistors (see drawing below). Some current will be lost in the shunt element(s) while the series resistance(s) would account for additional voltage drop. In the no-loss case, we have VOUT=VIN/2.

We have to calculate VOUT/VIN in the lossy case and then compute their ratio to get the insertion loss.

Considering only series losses, we may write

$$\frac{V_O}{V_I} = \frac{R_L}{R_S + \sum R_{SER} + R_L}$$

The additional attenuation is the ratio of 1/2 (lossless matched case) divided by this equation. One gets

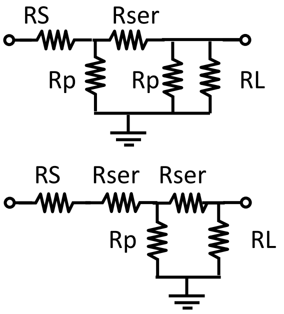
$$A_{dB} = 20 \cdot log\left(\frac{1}{2} \cdot \frac{R_S + \sum R_{SER} + R_L}{R_L}\right) = 20 \cdot log\left(1 + \sum \frac{R_{SER}}{2 \cdot R_S}\right)$$

We get 0.6dB and 2.3 dB for the Π and T networks (sufficient for the exam).

Let's now include the shunt branches !

Method: reduce the resistors from the load towards the source and calculate the intermediate voltage gains to get the total attenuation

From the sketch below we may write:



<u>Π network</u>

Req=((RL//Rp)+Rser)//Rp for the Π network from which we get the voltage after the input Rs:

V12=Vs·Req/(Rs+Req). Then we get Vout=V12·(RL//Rp)/(Rser+(RL//Rp))

Req=58 Ω; V12/VIN=0.437, VOUT/VIN=0.363

<u>T network</u>

Req23= Rp//(Rser+RL) for the T network from which we get the voltage at the intermediate node V123=Vs·Req23/(Rs+Rser+Req23). Then we get VOUT=V123·RL/(Rser+RL))

Req23=40.7 Ω; V123/VIN=0.344; VOUT/VIN=0.26

Conclusions :

The T network is indeed more lossy. In the ideal case with lossless components the voltage at the filter output would be 0.5·Vin corresponding to an attenuation of 6dB, for the T network, the attenuation increases to 11.7dB while it reaches 8.8dB for the Π network. The insertion loss is thus 2.8dB for the Π filter compared to 5.6dB for the T one. Comparing with 0.6 and 2.3dB (series loss only), we see that shunt branches are important too. While intuitions are nice, as engineers we should go for facts!

g) calculate the approximate noise factor including the main inductors Q limitations

in the lossless case, we have Rs=RL thus F is 2 and NF 3dB (see calculation made during the class).

For the Π network, we have the input current PSD given by 4kT/Rs in parallel to that of Req given by 4kT/Req. The noise factor F is thus given by (1/Rs+1/Req)/(1/Rs)=1+Rs/Req = 1.77. The noise figure

NF is thus 2.5dB. It's the same derivation that we did in the course. It is indeed smaller than the lossless case as Req is smaller than RL due to the shunt branches.

For the T network, the added noise PSD is 4kT/(Rser+Req23), F is thus 1+Rs/(Rser+Req23) thus 1.71 while NF is 2.3dB. It's even lower than for the Π network.

You would be right to argue that it was told during the lecture that the NF of a filter is given by its insertion loss. It's pretty much the case for the Π network with NF=2.5dB vs IL=2.8dB, but it does not hold for the T network: NF=2.3dB vs IL=5.6dB. That's something that would be worth digging into!

As a demonstration: the series parallel reduction method for resistor also applies for noise. Consider R1 and R2 in series, we may simply add the voltage noise sources 4kTR1+4kTR2=4kT(R1+R2) while for the parallel case we add the current noise PSD 4kT/R1+4kT/R2=4kT(1/R1+1/R2). We know that 1/Req=1/R1+1/R2. The above NF calculation is thus valid!

h) we still have a little problem to resolve due to the non-idealities of the components: how is the self-resonance of the series inductor(s) affecting our design? Compute the L impedance vs L & ω_O, ω_{SR} (center and self-resonance radian frequency), calculate how the component is affected. How can you resolve this issue?

we may write the non-ideal reactance as

$$Z_L(s) = \frac{1}{sC + \frac{1}{sL}} = \frac{sL}{s^2LC + 1} = \frac{sL}{1 - \left(\frac{\omega_o}{\omega_{SR}}\right)^2}$$

The 100nH coil has a self-resonance, ω_{SR} = 2.38GHz. The ratio of ω_O/ω_{SR} is thus 0.385 and our inductor value is increased by 17% (hey this explains why the L(f) curves increases vs f on our datasheet!, now you are really reaching the expert level !!!), we could thus pick a smaller coil and iterate if needed; the self-resonance will increase the filter rejection at ω_{SR}. However any spread in ω_{SR} will affect the L value as well !

Question 3) Smith Chart matching

i) Pick any pairs of source and destination purely resistive points on your Smith chart located on each side of 50Ω.

In the pdf correction attached, the starting point (source) is chosen at 0.5 and the destination (load) at 5 corresponding to Rs=25 Ω and RL=250 Ω if normalized to 50 Ω.

j) Sketch the two 2-componentsz HP & LP match and put next to the lines which and how the components are used (vert or hor C or L symbol for shunt, series); calculate the Q at the intermediate points

From $RL/Rs-1=Q^2$, we should get a Q of 3. To join the two points, we follow a constant resistance circle from (1) using a series element and a constant conductance circle from (2) using a shunt element. The intersection of those two circles gives the two solutions which reside symmetrically compared to the X-axis. To get the Q from the intersection points, you follow the same color line towards the outer circle that is perpendicular to the one corresponding to the added reactance or susceptance to find the imaginary magnitude. You get 0.6/0.2 for the susceptance/conductance or 1.5/0.5 for the reactance/resistance thus indeed a Q of 3. Since the constant Q lines all pass to the points -1 and +1 (on the X-axis outer bounds), you may draw the Q=3 line easily !

k) Add a possible 3 elements π network with a higher Q and label as above

Higher Q circles will be closer to the outer region of the Smith chart. With a Π network, on both side there is a shunt element that decreases the impedance seen at the node it is connected to (the trace gets closer to the left of the Smith chart which is a short for an infinite added cap or a zero H inductor). We thus start from the leftmost point (1) on a constant conductance circle so as to reach a lower constant resistance circle point from which we draw the circle to intercept with our initial constant conductance circle linked to point (2). A higher Q is indeed obtained as the RL/Rvirt ratio is increased as evidenced from a bigger distance between the two outer points on the X-axis. Note that there are several solutions and the elements do not need to be opposite when placed in series or shunt. From the Smith chart, the Q is ~1/.2=5

l) Add a possible 3 elements T network with a higher Q and label as above

Here it is the opposite, a series element will always increase the impedance since, thus we start from the highest resistor (2) towards the right (open circuit) and then proceed as above but with a shunt and series element reaching again the initial impedance circle starting from point (1). From the Smith chart, the Q is given by 2.2/.5 and is thus 4.4

m) Add a low-Q 4 elements network

A lower-Q matching is obtained by passing through an intermediate purely real point placed in-between (1) and (2) - or getting closer to the X-axis. The lowest Q is obtained when RL/Rvirt-1=Rvirt/Rs-1. Solving for Rvirt yield sqrt(RL·Rs)=sqrt(5·0.5)=1.58. Starting from the solution of a) you just break the path to remain closer to a lower-Q center part of the Smith chart. From the chart we extract approximately the following reactance/resistance and susceptance/conductance ratios of 0.75/.5 and .3/.2 both corresponding to a Q=1.5.

We may also write

$$Q_{LOW} = \sqrt{\frac{R_{VIRT}}{R_S} - 1} = \sqrt{\frac{\sqrt{R_L \cdot R_S}}{R_S} - 1} = \sqrt{\sqrt{1 + Q^2} - 1}$$

which is 1.47 when starting with a Q=3 and thus close to sqrt(Q).

Question 4) Derivation of the LNA input Z and gain

We want to maximise the gain of our LNA and achieve 50Ω input matching to the antenna. We will use the structure sketched above with inductive degeneration and a capacitor placed between the gate and source of the gain transistor. Neglect the substrate body effect (gm=gms) and all components which are not shown on the schematic. We choose to have a resonant load. All elements are discrete components; capacitors could be ideal; inductors should be chosen from the Coilcraft 0402dc series [0402dc.pdf (coilcraft.com)](0402dc.pdf). Self-resonance and Q @900MHz are given so no need to do a lot of calculations.

In calculations we consider ideal L components unless noted explicitly!

a) Calculate the input impedance of the circuit Vin/Iin after writing the Kirchhoff equations to determine the condition for impedance matching to Rs=50Ω (observe the equations first to avoid unnecessary and complex calculations !). Write the conditions clearly and re-use them later for simplification.

The following set of equations could be written for all three nodes (VIN, VMOS_SOURCE and VOUT) that we call V1, V2 and V3 respectively (components are named according to the node(s) number). The antenna source node is VS but not used in the Kirchhoff set. We assume R3 as a load (not RLC)!

$$\begin{pmatrix} sC12 & -sC12 & 0 \\ -sC12 - gm & sC12 + gm + 1/sL2 & 0 \\ gm & -gm & 1/R3 \end{pmatrix} \cdot \begin{pmatrix} V1 \\ V2 \\ V3 \end{pmatrix} = \begin{pmatrix} Iin \\ 0 \\ 0 \end{pmatrix}$$

Before solving anything, it's worth looking a bit closer at our equations. The first one gives us,

$$sC12 \cdot (V1-V2)=Iin$$

indeed, all current goes through the capacitor and generates some voltage on the transistor!

Dividing by V1 on each side yields after rearranging terms,

$$V2/V1=1-1/(Zin \cdot sC12),$$

which relates the gain from V1 to V2 as a function of the input impedance V1/Iin.

The third equation gives us,

$$gm \cdot (V1-V2)=-V3/R3$$

thus, the output voltage as a function of the transistor VGS=V1-V2

Reusing the above first result we get

$$gm \cdot Iin/sC12=-V3/R3,$$

thus

$$V3/Iin=-R3 \cdot gm/sC12,$$

which is our transimpedance gain!

If we are input matched, Iin will just be Vs/2Rs ! (Vs = antenna voltage or input voltage source).

We already have the LNA gain (none of you were able to get it properly ☹) but not yet the input impedance without the V2/V1 ratio. We must use the 2nd equation too to find

$$\frac{V2}{V1} = \frac{sC12 + gm}{sC12 + \frac{1}{sL2} + gm} = 1 - \frac{1}{Zin \cdot sC12}$$

and equate it with our above result to find

$$Zin = \frac{\frac{1}{sC12}}{1 - \frac{sC12 + gm}{sC12 + \frac{1}{sL2} + gm}} = \left(sC12 + \frac{1}{sL2} + gm\right) \cdot \frac{sL2}{sC12} = sL2 + 1/sC12 + gm \cdot \frac{L2}{C12}$$

Substitution as done above and as many of you have carried out, is cumbersome, prone to making mistakes (I had a hard time validating your calculation and many of you missed the simplifications).

Using matrix solving

We note that the equations for nodes 1 and 2 do not depend on V3, thus we may solve the reduced set of equations

$$\begin{pmatrix} sC12 & -sC12 \\ -sC12-gm & sC12+gm+1/sL2 \end{pmatrix} \cdot \begin{pmatrix} V1 \\ V2 \end{pmatrix} = \begin{pmatrix} Iin \\ 0 \end{pmatrix}$$

We have

$$V1 = \frac{\begin{vmatrix} Iin & -sC12 \\ 0 & sC12+gm+1/sL2 \end{vmatrix}}{\begin{vmatrix} sC12 & -sC12 \\ -sC12-gm & sC12+gm+1/sL2 \end{vmatrix}} = \frac{(sC12+1/sL2+gm)\cdot Iin}{\frac{C12}{L2}}$$

From which we get

$$Zin(s) = \frac{V1}{Iin} = sL2 + 1/sC12 + gm \cdot \frac{L2}{C12}$$

This is ultra fast isn't it ?

The first term is real and should be equated to Rs=50 Ohm; the reactance should be cancelled thus L2 and C12 must resonate at 915MHz. There is thus a single degree of freedom. Picking any of L2, C12 or gm determines all 3 parameters.

b) Sketch on a Smith chart normalized to 50Ω the input impedance when varying slightly the frequency (this will be re-used later).

At ωo, we are at the center; around the resonance we follow the cst R curve with series cap (lower freq) or series coil (higher freq)

c) Calculate the transistor source voltage vs Iin, this will be useful for the overall gain calculation; now you may get the VGS voltage as a function of Iin; what do you conclude?

$$V2 = \frac{\begin{vmatrix} sC12 & Iin \\ -sC12-gm & 0 \end{vmatrix}}{\begin{vmatrix} sC12 & -sC12 \\ -sC12-gm & sC12+gm+1/sL2 \end{vmatrix}} = \frac{(sC12+gm)\cdot Iin}{\frac{C12}{L2}} = \left(sL2 + gm\cdot\frac{L2}{C12}\right)\cdot Iin$$

Computing VGS/Iin=(V1-V2)/Iin we get what we have already derived above using only the first equation. Indeed VGS is defined by Iin/sC12.

We may also get V2/V1 as

$$\frac{V2}{V1} = \frac{\begin{vmatrix} sC12 & Iin \\ -sC12-gm & 0 \end{vmatrix}}{\begin{vmatrix} Iin & -sC12 \\ 0 & sC12+gm+1/sL2 \end{vmatrix}} = \left.\frac{sC12+gm}{sC12+gm+1/sL2}\right|_{\omega o} = 1+\frac{sC12}{gm} = 1+\frac{sL2}{Rs}$$

with the rightmost terms, the voltage gain at resonance and then using the matched condition. You may notice that sL2/Rs is the Q of the unloaded RLC series network. We thus have a voltage multiplication by 1+jQ from node 1 to 2. V1 and V2 are both positive when the circuit is fed with a current Iin but outphased by 90° (this result is consistent with the above calculated V2/V1 ratio with the minus sign and 1/sC12/Zin). The gain in magnitude is sqrt(1+Q^2).

d) Calculate the maximum in-band gain of the LNA from the antenna to the output; simplify your result and express it as parameters of the components that are used when it determines the performances

The current flowing through the MOST is given by gm·V12 and is fed into R3 at resonance (this is intuitively derived but could be obtained from the 3rd equation in our set, we may also replace R3 by the complete RLC impedance but let's keep things as simple as possible since we are anyway interested in the gain at the resonance)

V12/Iin was found to be 1/sC12. Iin is simply given by Vs/(Rs+Zin)=Vs/2/Rs when matched

We thus get

$$\frac{V3}{Vs} = \frac{V3}{V12} \cdot \frac{V12}{Iin} \cdot \frac{Iin}{Vs}\bigg|_{\omega o} = -\frac{gm \cdot R3}{2Rs \cdot sC12}$$

R3, the load resistor (either limited by the own Q of the inductor or lowered with the addition of a resistor) is now a 2nd degree of freedom.

We could express R3 as ωL3·Q3 and gm/sC12/2Rs as 1/2ωL2 using the Zin matching condition, thus the voltage gain becomes

$$\left|\frac{V3}{Vs}\right|_{\omega o} = \frac{1}{2} \cdot Q3 \cdot \frac{L3}{L2}$$

Counter-intuitively the gain is now maximized when L2 is made small meaning that C12 is large since their product is constant.

That equation indicates that whatever the load Q-factor, which could be limited with an additional resistor, we should pick a large load coil and a small degeneration one to get the maximum gain. This is counter-intuitive with the fact that a large C12 - resulting from the resonance condition on L2C12 - reduces the VGS voltage at the transistor as it is given by 1/sC12·Iin!

e) The input matching condition is making things less intuitive as it is binding some parameters; assuming sub-threshold operation calculate the upper and lower gm and current that would be needed in our LNA when using the smallest or maximum coil among the inductor choice

Since ωL2=1/ωC12, the matching condition gm·L2/C12=Rs could be rewritten as a function of gm and C12 or L2 instead of their ratio, one gets

$$gm/(\omega \cdot C12)^2 = Rs \text{ or } gm \cdot (\omega \cdot L2)^2 = Rs$$

Changing L2 or equivalently C12 affects gm quadratically. Indeed, decreasing L2 increases the gain because our matching conditions imposes that gm increases quadratically leading in turn to an increased current into the load.

From our component list 0.8nH<L2<120nH thus 2.36 S >gm>1.04e-4 S and 83mA>I>3.7uA assuming nUT=35mV and sub-T operation. The equivalent C12 range to get a purely real Zin is 38pF>C12>0.3pF

f) Discuss shortly the main results and how voltage and power gains are affected over the design space; select the two sets of L and C components

The voltage gain is Av=-gm·R3/sC12/2Rs

which reduces to Av=-R3·sC12/2=-R3/sL2/2 when using the input matching condition

The condition on gm and thus the circuit consumption is hidden in this relation as analyzed above. Let's look at the power gain instead:

The power gain is $G=Av^2 \cdot Rs/R3=1/4 \cdot (gm/(\omega C12))^2 \cdot R3/Rs$

which reduces to G=1/4·gm·R3 when using the input matching condition

It's thus always beneficial to maximize the output resistance. The output voltage swing is indeed proportional to R3 so as the output power.

The other parameter is the transistor gm, and indeed the higher the circuit current consumption, the higher the power gain which will matter when computing the cumulated NF among blocks.

To get the maximum voltage and power gain, one must pick a small L2 and a large high-Q L3. However, we are not willing to spend 83mA in our LNA!

If we allocate 1.6mA or 160uA at IC=1 (gm=ID/nUt·0.62), we get L2 = 7.3 or 23nH. Concerning L3, going for 120nH would give the highest gain even if we limit the Q of the load with a resistor (R3=ωL3·Q). However, with 120nH, the total capacitance of the node should not exceed 250fF, which would be too low if it includes a circuit pad with some ESD (electro-static discharge) protection. To resonate with 1pF, which is deemed more conservative, L3 should be ~30nH. We could try to go as high as 60nH provided we achieve 0.5pF of total stray capacitance on the output node including any loading from succeeding stages.

Limiting the Q to 15 assuming 2% spread on the components (see question 1) would yield the following:

L3=120nH, L2=7.3nH, R3=10kΩ -> Av=120/7.3·15=246.5 or 47.8dB, G=24.7dB

L3=30nH, L2=7.3nH, R3=2.5 kΩ -> Av=61.6 or 35.8dB, G=18.7dB

when consuming 1.6mA.

With 160uA current and thus L2=23nH, all above gains are reduced by 10dB.

Now we have a bit narrowed down the design space! But being outside that range does not mean the design is wrong!

g) Calculate the Q of the matching network and the Q of the load (the output of the block is not yet loaded but compare to the 915MHz BW requirement

The matching network including the source resistance has a 100 Ω input resistance while its reactance is 1/ωC12 or ωL2. Over the L2 range, the Q= ωL2/100 Ω is thus between 0.046 and 6.9. With 7.3 and 23nH, corresponding to our 0.16-1.6mA range, it is about 0.4 and 1.2. We have thus a wide impedance matching that is quite accurate as we may write

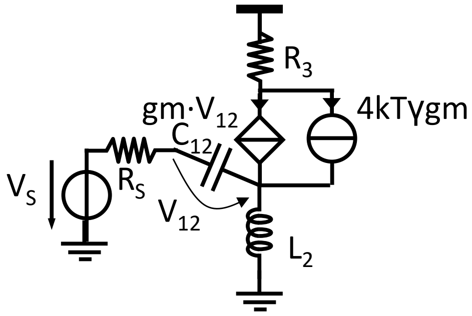
$$\frac{\Delta Z_{IN}}{Z_{IN}} = \frac{\Delta gm}{gm} + \frac{\Delta L2}{L2} + \frac{\Delta C12}{C12}$$

with 2% spread on each parameter (current may be trimmed), we have 50 ±3Ω

According to the coil table, without degradation, the Q of the load would reach about 50 which is too high for our BW and considering the component spread. We have thus to degrade the load Q-factor with the addition of a resistor. However, since we will load the LNA, we could shoot for an unloaded Q of 30 yielding a loaded Q of ~15 as used for the above calculation.

h) Sketch the small circuit equivalent of the LNA adding controlled current sources modeling the transistor and its main noise source (1/f noise is neglected)

The input current must flow through the C12 capacitor generating a voltage V12. That voltage determines the current flowing through the main transistor (gm·V12) and the load. Both currents must flow through L2. The MOS current noise $in^2=4kT\gamma gm$ flows in parallel to the MOS current.



Question 5) Output impedance and noise factor calculations

i) Now you need to connect Rs to the input!

The Kirchhoff equation should be modified as follow

$$\begin{pmatrix} \frac{1}{Rs}+sC12 & -sC12 & 0 \\ -sC12-gm & sC12+gm+1/sL2 & 0 \\ gm & -gm & 1/R3 \end{pmatrix} \cdot \begin{pmatrix} V1 \\ V2 \\ V3 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \end{pmatrix}$$

j) Determine the output impedance of the LNA (why is it so simple ? justify with the equations (you do not need to solve the system!)

$$\begin{pmatrix} \frac{1}{Rs}+sC12 & -sC12 & 0 \\ -sC12-gm & sC12+gm+1/sL2 & 0 \\ gm & -gm & 1/R3 \end{pmatrix} \cdot \begin{pmatrix} V1 \\ V2 \\ V3 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ i_{OUT} \end{pmatrix}$$

Injecting some current on V3 modifies indeed V3. However V3 does not influence the current flowing on nodes 1 and 2. Thus the only solution to our equations is that V1=V2=0 (Vsource=0 due to superposition).

The output impedance is thus simply R3.

k) The noise factor calculation of the LNA is a bit tricky; we will consider only the MOS transistor noise and neglect that of the output loss first; the relevant noise source is between the output node and the intermediate L-degenerated one and is thus 100% correlated on the way it acts on both nodes. We would need to write all 3 equations and use "-in" as the output node noise current and "+in" at the other node before solving Vout("in") noise. Rather we consider only the first two nodes equations and calculate the transfer function VIN("in") before computing the equivalent voltage noise source PSD $vn^2(in^2)$; with impedance matching it is straightforward to compute the equivalent noise of the input resistor and get F; show your calculation and then do not forget to do all the simplifications and you will be surprised how simple is the result!

Our system should be rewritten as

$$\begin{pmatrix} \frac{1}{Rs}+sC12 & -sC12 & 0 \\ -sC12-gm & sC12+gm+1/sL2 & 0 \\ gm & -gm & 1/R3 \end{pmatrix} \cdot \begin{pmatrix} V1 \\ V2 \\ V3 \end{pmatrix} = \begin{pmatrix} 0 \\ i_n \\ -i_n \end{pmatrix}$$

Solving for V1(In) the reduced sub-set of equations one gets

$$V1 = \frac{\begin{vmatrix} 0 & -sC12 \\ i_n & sC12+gm+1/sL2 \end{vmatrix}}{\begin{vmatrix} \frac{1}{Rs}+sC12 & -sC12 \\ -sc12-gm & sC12+gm+1/sL2 \end{vmatrix}} = \frac{sC12\cdot i_n}{\frac{1}{Rs}\cdot(sC12+gm+1/sL2)+\frac{C12}{L2}}$$

At resonance, we may simplify as

$$V1|_{\omega_o} = \frac{\omega_O C12\cdot i_n}{\frac{gm}{Rs}+\frac{C12}{L2}} = \frac{\omega_O L2\cdot i_n}{2}$$

From which we get

$$v_{n,1}^2 = \frac{(\omega_O L2)^2\cdot 4kT\gamma gm}{4} = kT\gamma gm\cdot(\omega_O L2)^2 = kT\gamma Rs$$

At node 1 with impedance matching, the noise due to Rs is simply kTRs. We may write

$$F = 1+\frac{kT\gamma Rs}{kTRs} = 1+\gamma$$

l) Using the proper voltage gain from VIN to VOUT get the equivalent output noise including the noise due to the load loss (Rp) and comment your result

$$v_{n,3}^2 = kT(1+\gamma)Rs\cdot\left(\frac{\mathrm{R3}}{\omega\mathrm{L2}}\right)^2 + 4kTR3 = kT(1+\gamma)gm\cdot\mathrm{R3}^2 + 4kTR3$$

We get

$$F = (1+\gamma)+\frac{4R3}{Rs\cdot\left(\frac{\mathrm{R3}}{\omega\mathrm{L2}}\right)^2} = 1+\gamma+4\frac{(\omega\mathrm{L2})^2}{Rs\cdot R3} = 1+\gamma+\frac{4}{gm\cdot R3}$$

The gm·R3 product is 280 and 28 for the 1.6mA and 160uA cases with L3=120nH, Q=15 and 70 and 7 when L3=30nH, Q=15. The noise of R3 is thus negligible except for the last case (160uA, L3=30nH).

Some of you tried to solve directly the noise on node V3 with equation 3:

$$gm(V1-V2)+1/R3\cdot V3 = i_n$$

but using the first equation

$$\left(\frac{1}{Rs}+sC12\right)\cdot V1 - sC12\cdot V2 = 0$$

we conclude that V1-V2 is no longer zero when Rs is included !!!

You would have to solve the following:

$$V3 = \frac{\begin{vmatrix} \frac{1}{Rs} + sC12 & -sC12 & 0 \\ -sc12 - gm & sC12 + gm + 1/sL2 & i_n \\ gm & -gm & -i_n \end{vmatrix}}{\begin{vmatrix} \frac{1}{Rs} + sC12 & -sC12 & 0 \\ -sC12 - gm & sC12 + gm + 1/sL2 & 0 \\ gm & -gm & 1/R3 \end{vmatrix}}$$

from which we get

$$\left.\frac{\frac{-i_n}{Rs} \cdot \left(sC12 + gm + \frac{1}{sL2}\right) + -i_n \left(\frac{C12}{L2} + \left(\frac{gm}{Rs}\right)\right)}{\frac{1}{Rs \cdot R3} \cdot (sC12 + gm + 1/sL2) + \frac{C12}{L2 \cdot R3}}\right|_{\omega o} = \frac{-i_n R3 \left(\frac{C12}{L2}\right)}{\frac{gm}{Rs} + \frac{C12}{L2}} = \frac{-i_n R3}{2}$$

At resonance, indeed the MOS noise source sees R3/2 as impedance due to some feedback from the gain transistor. Replacing i_n by its formula, we get for the output noise due to the MOS transistor

$$v_{n,3,MOS}^2 = kT\gamma gm \cdot \mathrm{R3}^2$$

which is indeed equivalent to what we computed above with the reduced subset of equations.

Question 6) Circumventing LNA imperfections

a) Now we have a little issue, we have a strong parasitic capacitance to GND at the input of the LNA Cpar (e.g. pad capacitance), sketch how this affects the impedance matching on the Smith chart of 4b)

We are adding a shunt element, thus we move on a cst conductance curve. See attached drawings.

b) Propose graphically and with a short text two different ways to solve the problem to get back to 50Ω using one additional component (for one of the solutions you have to change something to Zin !)

We could add a shunt inductor to compensate for the shunt capacitor and resonate it out

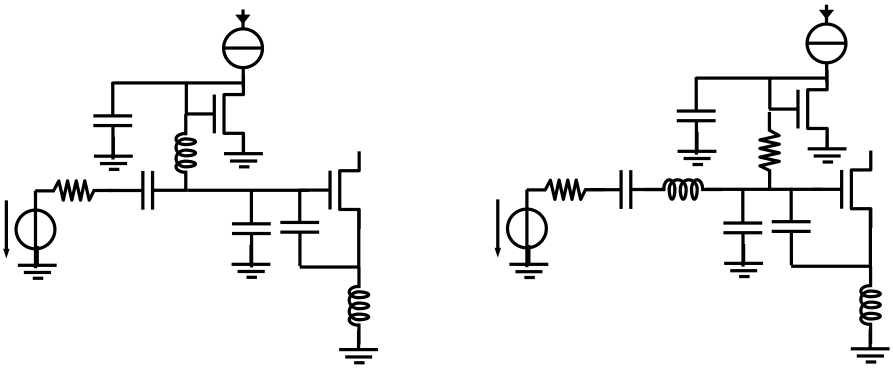
The common source LNA is often used with an additional external series inductor. Moving on the constant resistance line, we would reach a too low purely real resistance value. We thus modify our initial matching condition to start from a higher impedance. The addition of the series inductor gives us an extra degree of freedom breaking the $C12 \cdot L2 = 1/\omega o^2$ relationship. See attached Smith chart (note that drawing still assumes $C12 \cdot L2 = 1/\omega o^2$)

Note that we could also have lowered the resistance of our initial matching condition including a residual positive reactance (ωL) so that adding the shunt C element bring us back right on 50 Ω without any additional component.

c) How could you do simply the DC biasing of the transistor using an input mirror? sketch your solutions for the two cases of 4b); any impact on Zin (reasoning only)?

When the shunt C is neutralized with a parallel inductor, the input mirror with a decoupling cap could be attached to the coil other node (left drawing). Note that a DC blocking cap should be added if an external generator is used to test our circuit (and thus in simulation). The antenna itself could be isolated at DC and reach 50 Ω only in its designed bandwidth.

With the series L compensation, a resistive biasing may be used as we did for the mixer. This element is in parallel with the input impedance thus it must be made rather high (>1 KΩ) so that it marginally influences Zin and F.



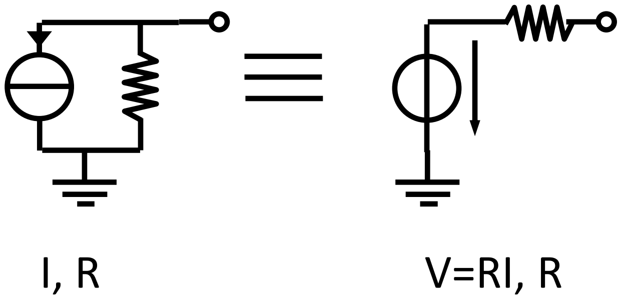
d) Another issue is that we have neglected CGD; with high LNA gain it is multiplied by the so-called Miller effect (1-Av); we could add a cascode transistor between the load and the first one with e.g. its gate connected to VDD (same sizing as former MOS is perfectly fine)

This is used in most LNAs. In addition, it improves the LNA reverse isolation (coupling from LNA output node to antenna) and thus its stability.

e) Calculate the voltage gain from the main transistor VGS to the source of the cascode and show that the current injected into the load does not vary (we neglect the cascode source capacitor); what is now our Miller gain? Our gain equations are thus still valid !

The current to the cascode is given by the gain transistor and equal to gmV12, neglecting the capacitance, it creates a voltage at the source of the cascode VS_CAS=-gmV12/gm_cas. If both gm are equal (usually we simply replicate the same transistor), the gain is -1. The CGD cap has thus a voltage of V1- -(V1-V2)=2V1-V2. We could thus add the CGD cap in parallel to that of CGS (V1-V2) and again at node V1 and GND (V1) reducing our Miller cap. The current injected towards the load is gm·VS_CAS and indeed still -gm·V12 with our assumptions. The capacitor at the intermediate node should be limited with a careful layout (gm/Cpar>ωo if possible)

n) knowing that Thevenin, Norton equivalents hold, match the LNA output impedance so that it may be used as the filter input impedance, what is now the Q of the LNA output stage? Does it fit with the 915MHz ISM band 26MHz BW requirement? How is the power gain affected?



Our LNA output is a current source (MOS) with a loading resistor. It is equivalent to a voltage source RI with a series resistor R. We may thus match it so that the filter sees 75 Ω with an LC L network.

This is an interesting reasoning! We could have placed the BP filter before the LNA and suffer from the added NF of the filter. Placing it after the LNA is off course worst as per the linearity but the filter NF is masked by the LNA gain. We know that when matched, the same power is dissipated in both the source and load resistances. Thus, our conclusions on the power gain of the LNA (maximise output R for max power gain) still hold when we convert the impedance back to ~50 Ω (75 Ω in our case). This, as long as the added losses due to the new components remain reasonable.

We should now match the LNA output impedance to 75 Ω with the Π filter. The Q of our matching network will thus be Q=sqrt(R3/75 Ω-1) and we chose an LC L-network. L is in parallel with L3 and the two components could be replaced by their equivalent and we only need a series capacitor towards the filter.

C is given by Q=1/ωC/75 Ω thus C=1/ω/Q/75 Ω while L is given by Q=R3/ωL or L=R3/ω/Q

Should we keep the LNA unloaded Q limit of 15 as discussed in 4e), we would reduce the voltage and power gains by 6 and 3dB with a loaded-Q halved

Rather, we shoot for a LNA loaded Q of 15, thus R3 could be doubled compared to our initial calculation so that our voltage and power gain remain valid with the filter load. We have R3=20 or 5kΩ with L3=120 or 30nH respectively (component Q is >50 hence ok). The matching network Q that we should use for the calculation of the elements is thus 16.3 or 8.1 (it is also halved when considering both input/output resistors in the end).

We find the following components values Cm=142 or 286fF, Lm=213 or 107nH for L3=120 or 30nH. After reduction of the two parallel inductors, we find L3_eq=77nH or 23.5nH respectively instead of 120 or 30nH. A 150fF capacitor from Kyocera has a tolerance of ±20fF (±15%) and a Q of 550 @900MHz. Up to 2pF, the absolute tolerance remains unchanged. This large tolerance might force us to reduce the Q of the LNA and thus of our coupling network.

We should also pay attention to the fact that our filter requires a wideband input/output resistance to guarantee its transfer function. As the Q of the loaded matching network we shoot for is 8 or 4 and thus quite close to that of the filter, we will experience significant ripple. Our matching network is of the HP type and will add some selectivity. It will be interesting to simulate the behavior and selectivity of the combined blocks and compare it with ideal results of individual blocks.

The power at the -lossless- filter output is equal to that of the loaded LNA and halved compared to the unloaded case. We may write:

G_{LOADED}=G/2=1/8·gm·R3

The voltage gain from the antenna to the lossless filter output is thus

AV=sqrt(1/8·gm·R3·75 Ω/50 Ω)

As we have doubled R3 compared to 4e) to account for the filter loading, we get the same results:

L3=120nH, L2=7.3nH, R3=20kΩ -> Av=47.8dB, G=24.7dB

L3=30nH, L2=7.3nH, R3=5 kΩ -> Av=35.8dB, G=18.7dB

when consuming 1.6mA.

With 160uA current and thus L2=23nH, all above gains are reduced by 10dB.

o) We need additional gain in our system, what block could you place at the output of the filter to implement the output impedance of the filter, how do you modify that block?

We simply reuse our LNA after modifying the input matching condition to 75 Ω.

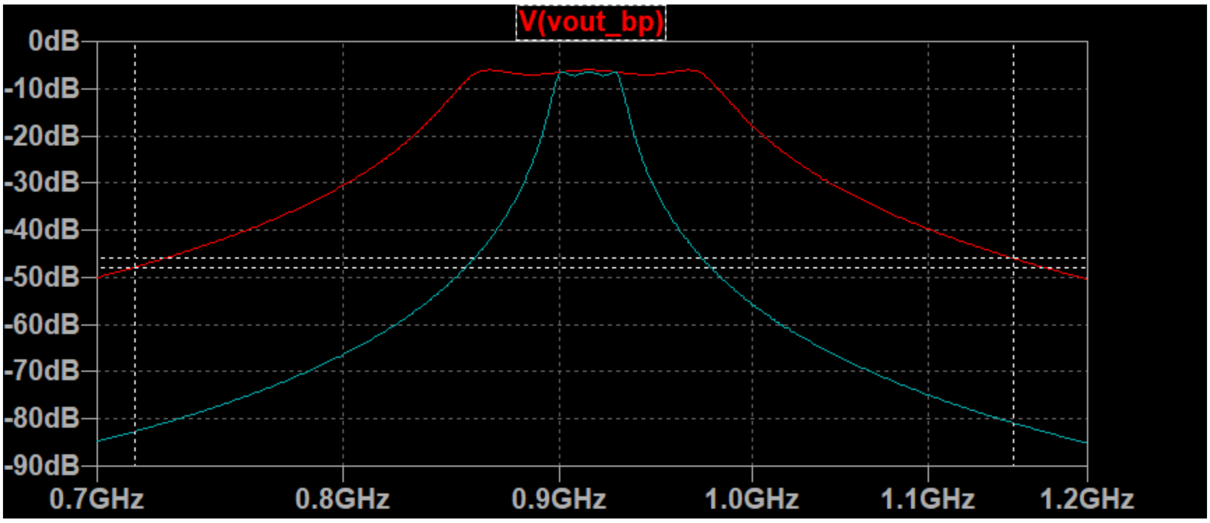
p) We make the assumption a differential output is available to drive the mixer

We would typically load the second LNA with a transformer or an RF balun to do that conversion. Another alternative would be to leave the double-balanced mixer second RF port only DC-biased.

Congratulations, now you have designed a pretty good front-end ! It's time to run some LTI spice simulation using ideal components with the addition of the cascode transistor !

Question 6) AC simulations of the front-end and non-linearity NF discussions

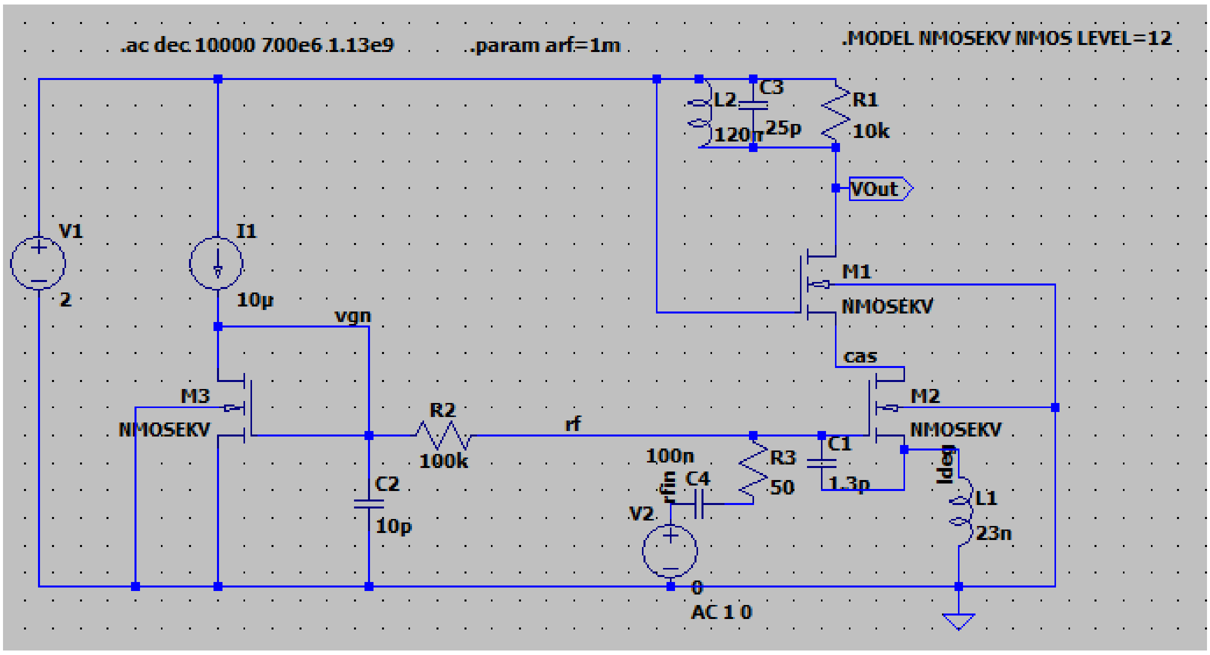
a) Make an AC sim of the filter alone with ideal components to determine the rejection in the image band



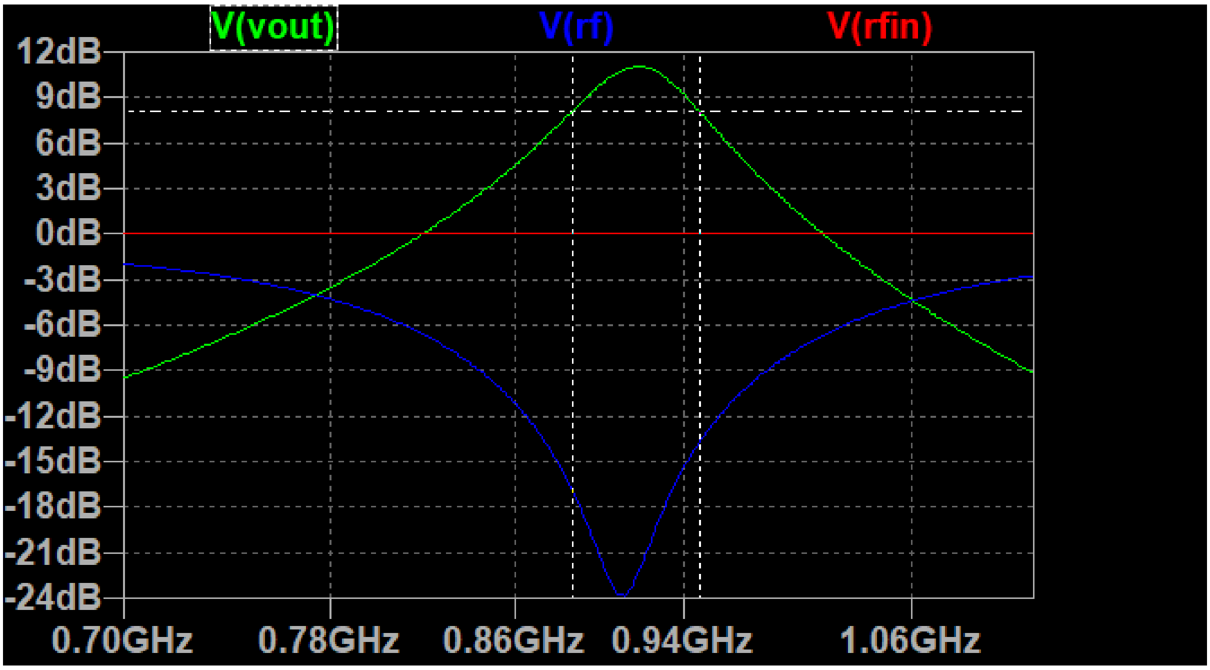
Simulation of the 3rd order Chebyshev filter with a Q of 8 (red) and with a Q of 30 (cyan) to highlight our RF BW. You see the margin that is kept to account for component mismatch. Again an implementation of a filter with a Q of 30 is not feasible with LC components due to the large ratio required (>2000) and the component spread that shifts the curves laterally. Even with a Q of 8, the image band attenuation is >45dB (cursors centered at 715 and 1115 MHz.

One obtains the same result when driving the circuit with a current source of 1/75 and a shunt 75Ω resistor.

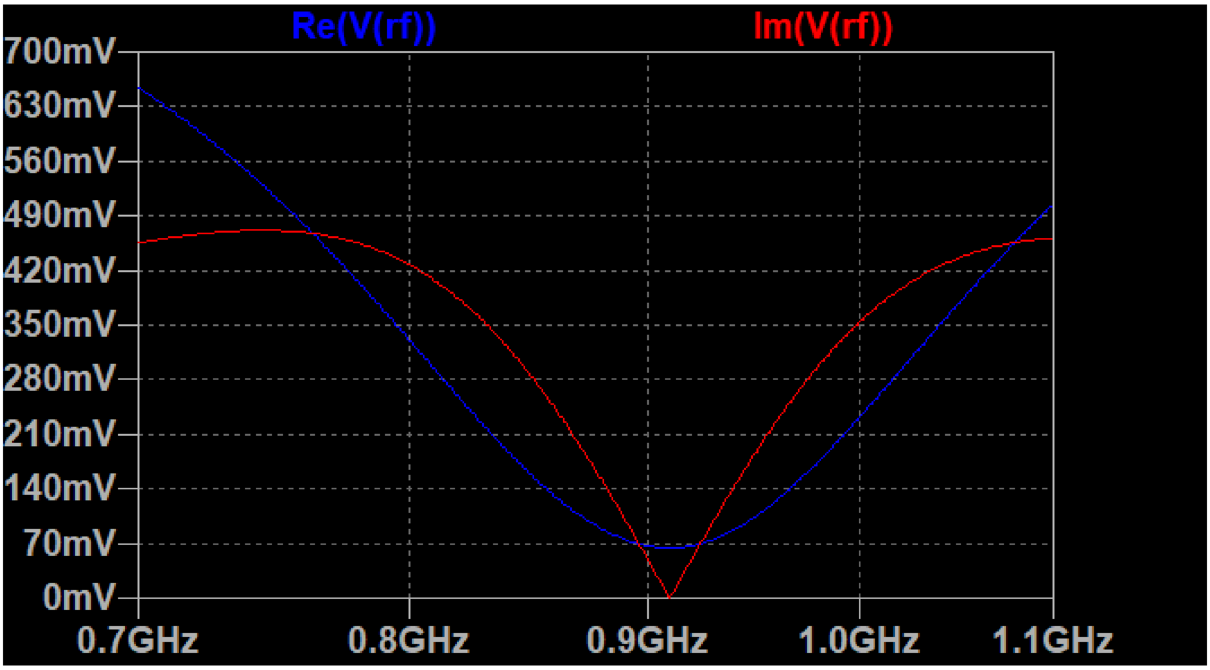
b) Make an AC sim of the LNA alone to determine the selectivity, compare with that of the filter



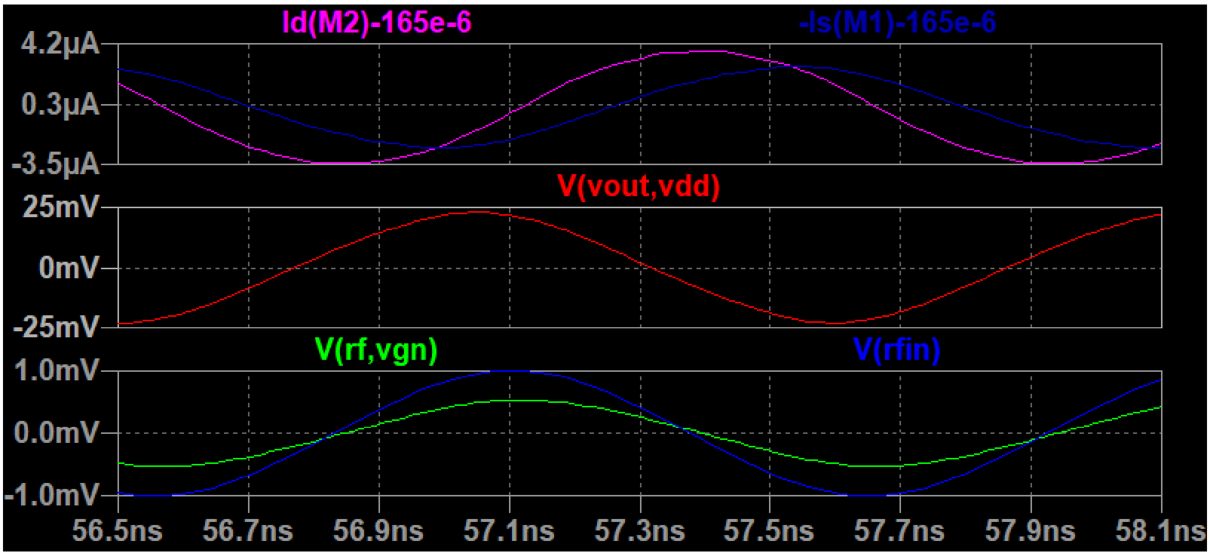
Schematic above with 160uA in M2 (m=16 using same MOS as mixer, cascode identical), note the bulk connections to GND (body effect accounted for). An ideal AC-coupling cap (C4) added to isolate the input source from the LNA DC bias.



Plot of the LNA gain (11dB). The BW is 61MHz indeed corresponding to a Q of 15. The input voltage should be at -6dB due to input match (it is 18dB lower, factor 8!). This is again a problem of LT spice!

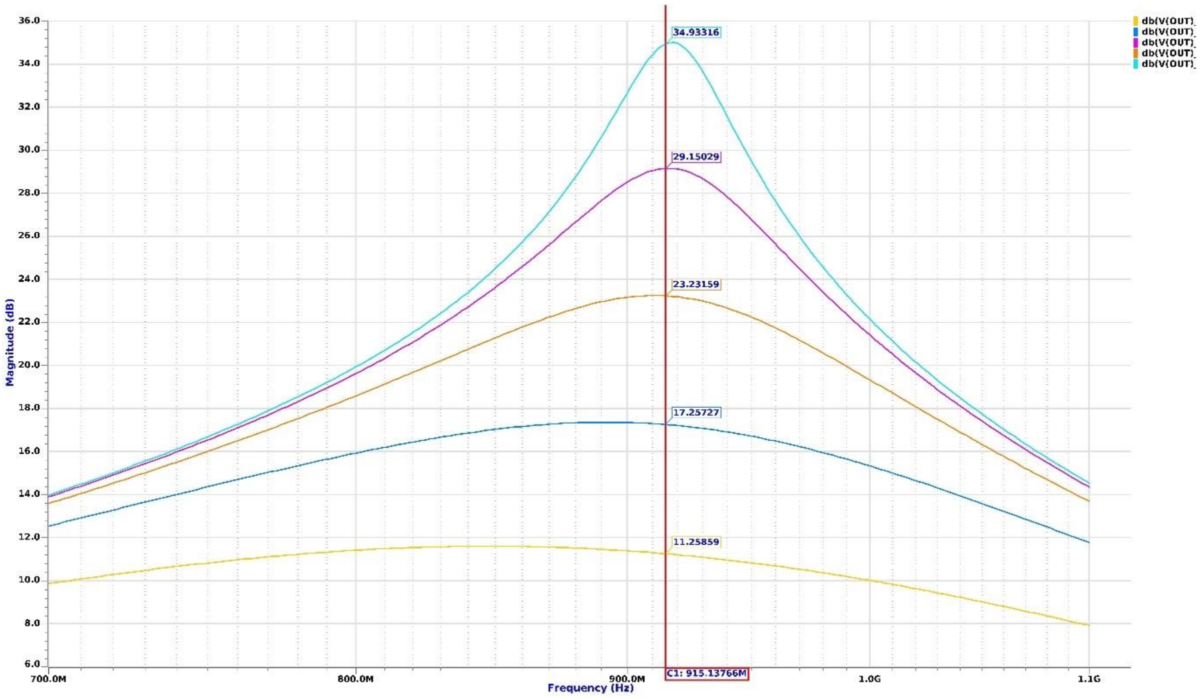


Plot of the real and imaginary part of node 1 (LNA input, matched to 50 Ω). We should get 0.5V instead of 70mV ! The imaginary part indeed goes to 0 at 915MHz.

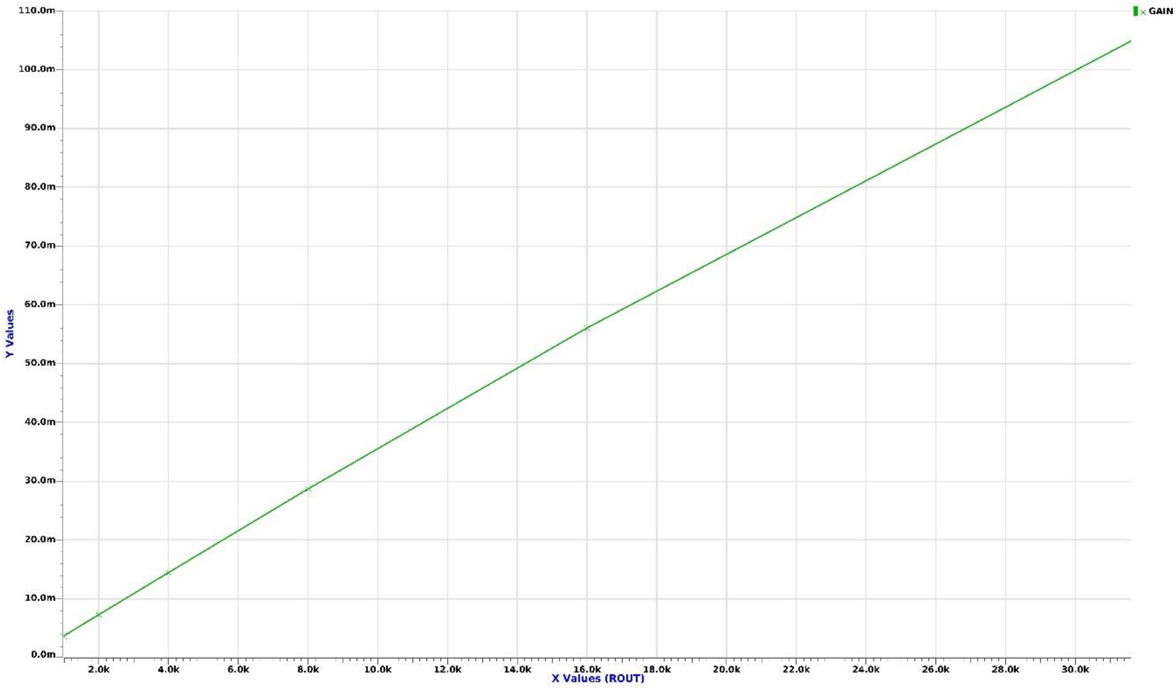


Transient simulation of the exact same circuit yields different results. With a 1mV RF signal (rfin, blue), we do get a 0.5mV in-phase one on V1 (green), input is thus matched to 50 Ω ; some current is lost in the cascode capacitor (top, blue cascode current, magenta, gain MOS one). The gain is 25 from input to output, thus 28dB while we would expect 38dB from our calculations (n-factor, cascode loss neglected). Note that the CGS capacitor was reduced iteratively to 0.8pF in the transient simulation to correct for the MOS one. Doing the same correction on the AC sim, shifts the result towards higher frequencies. The AC cap is thus not properly extracted.

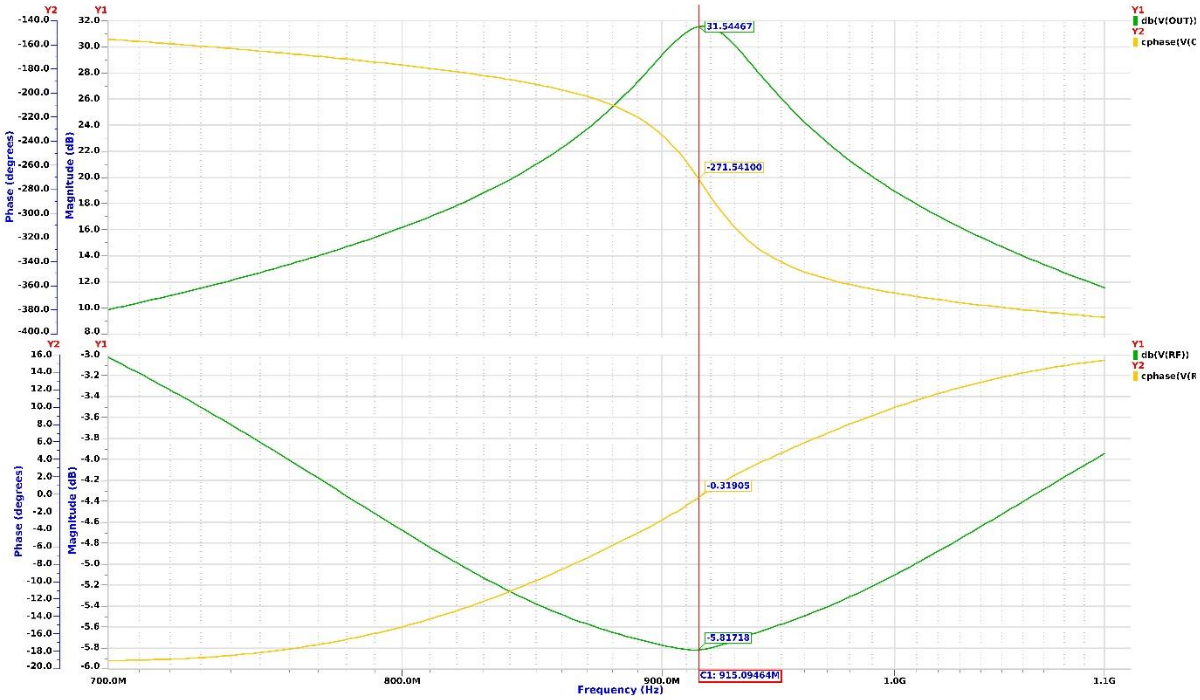
To compare the accuracy of LTSpice versus a professional simulator, the same circuit was implemented in a TSMC 180nm node. The MOS are 3um wide and 0.18um long. A single one is used for the bias while 10 parallel instances are used for both the gain and cascode transistors. An IC-factor close to 1 is obtained with 10uA input bias corresponding to an LNA current of 140uA (the mirror ratio is 14, - instead of 10 - due to gds effect as VD=1.2V for the gain MOS vs 0.5V for the input diode connected one).



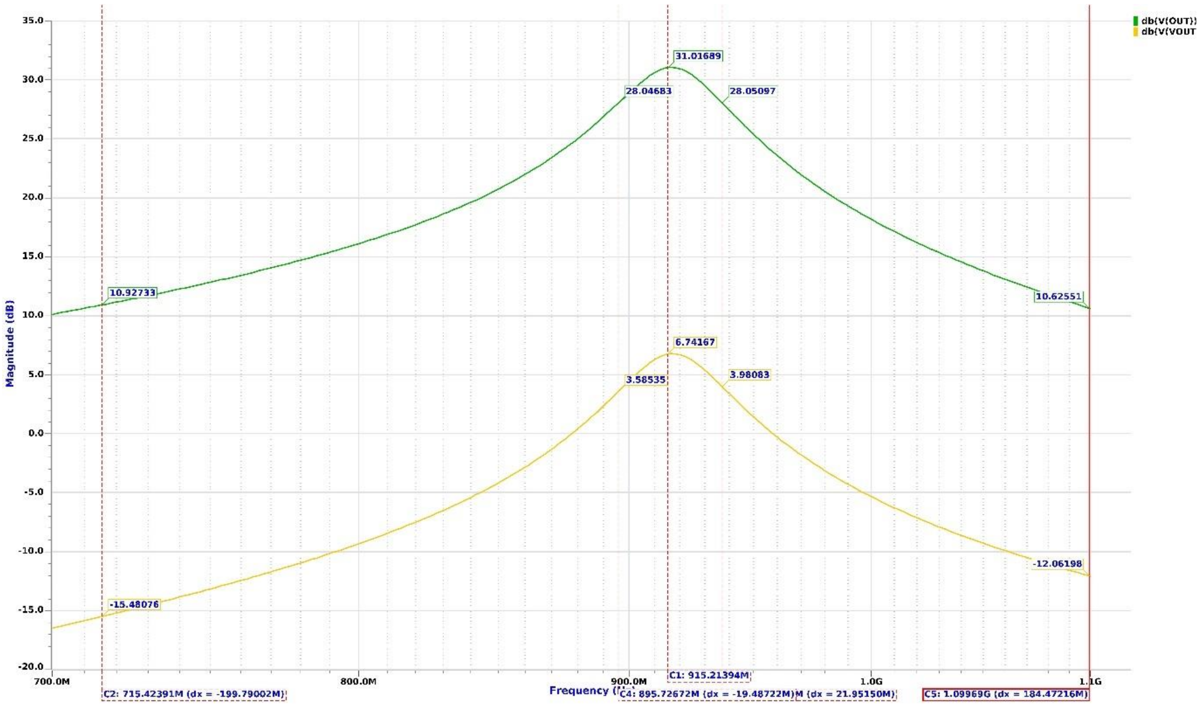
The voltage gain is plotted above for various output resistances (1, 2, 4, 8, 16 kΩ). The gain indeed increases by ~6dB each time R3 is doubled. Note that C3 was adapted to center the curves at 915MHz.



In a transient simulation, the gain is extracted again vs the output resistance and shown in the above plot. Rout=10k corresponds to a Q of 15. With L3/L2=120/23, the voltage gain is expected to reach 39 according to our calculation (L3/L2*Q/2). We read 35, this is much closer. You may notice the saturation of the gain at large Rout (likely due to output conductance and/or frequency selectivity and slight resonance offset).

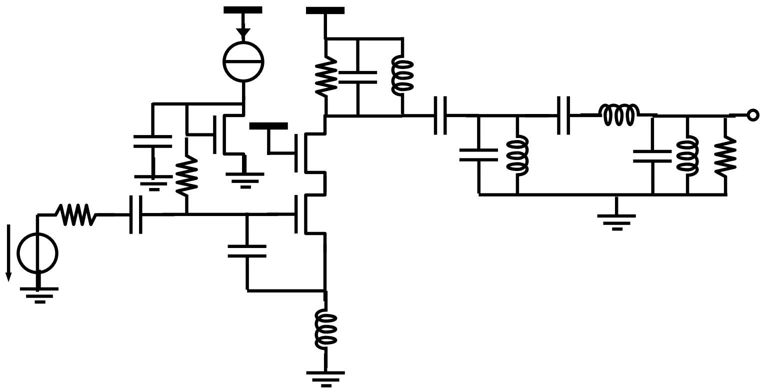


The circuit with R3=20 kΩ is now loaded with a 75 Ω resistor and impedance matched so that the loaded R is still 10 kΩ. The top above plot shows the V3 node output magnitude (31.5dB of gain) and phase. The bottom one shows that the input node is impedance matched (6dB attenuation, 0 phase near 915 MHz).

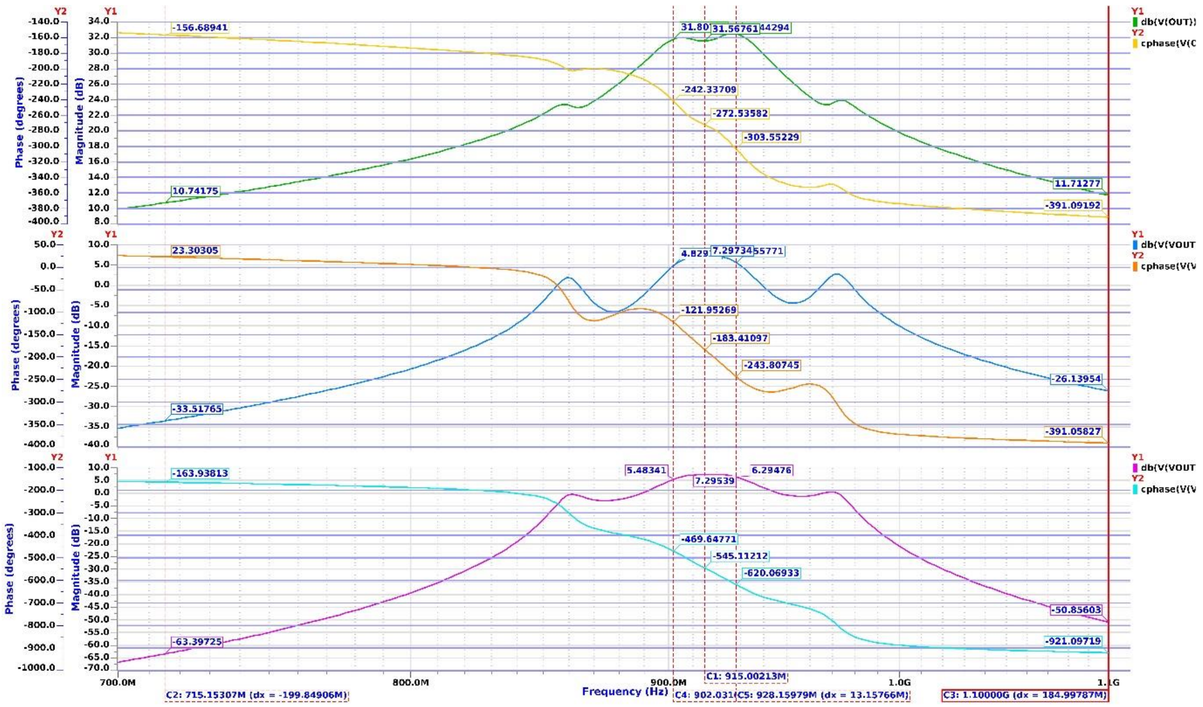


The above plot shows the voltage gain at node V3 and at the 75 Ω node. The resistor ratio is 10k/75 corresponding to 21.2dB difference (24dB seen on plot, thus we could optimize a bit the output match). With 6.7dB voltage gain on 75 Ω, the power gain from the input is 1.75dB lower at 5dB. From the LNA input (PG=-3dB), we thus have ~8dB of gain. At 200MHz from the RF band, we have 21dB attenuation at the LNA output. As our matching network is HP, we see some asymmetry at HF.

c) Make an AC sim of our complete front-end including LNA, filter and 2nd amplifier, what is the gain and selectivity in the image band?



The simulated circuit, whose results are presented hereafter, is sketched above. Mind the LNA output impedance which is transformed to appear as the required 75 Ω filter input impedance. The filter output impedance could be implemented with a slightly modified second presenting a 75 Ω input impedance. As the LNA output resistance R3 was increased to 20 kΩ (unloaded Q of 30), there is no gain penalty when it is loaded with the filter since the loaded Q is more or less halved again close to our Q limit of 15 imposed by the ±2% component spread (neglecting the filter losses of course !).



When loading the LNA with our 3rd order BP filter and a 75 Ω resistor output, we get the above results. The top plot is the LNA output node gain, the middle one, that at the filter input and the bottom one that at the filter output. As the Q of our LNA (15) and that of the output matching network (16) are greater than that of the filter (8), we see quite some ripple. But within our band (markers), the gain is rather constant 5.4-7.3dB! The attenuation at 1.1GHz is >50dB and at 715MHz >63dB.

Well, this is not too bad for a preliminary design! One would increase the consumption by 3x to increase the gain figures. It's rather simple to modify our LNA so that it presents a 75 Ω input and use it as a second gain stage.

d) Discuss shortly the linearity and noise-factor trade-offs, which are the most critical blocks

LNA is critical regarding the NF so that is masks that of the following blocks. Second gain stage is more sensitive to linearity but consider that now we have a much better selectivity.

e) How would you simulate the LNA or amplifier non-linearity efficiently? What parameter of which component would you monitor and how would you predict the IM3 product and IP3? Explain why it's much more difficult when looking at the LNA output voltage?

A single tone test produces H3 near 3GHz. Attenuation at the output is thus quite high. Looking at the MOS current help us quantify H3 nonetheless. We then know how H3 and IM3 products are related (factor 3) and could predict two tones test results and determine the SFDR of our front-end.

Question 7) Mixer transient simulations

a) Using the double-balanced mixer designed in the exercise, duplicate it and drive it with quadrature LO signals (0,180 and 90,270, in-phase and quadrature); only use ideal signal sources, you may keep the initial frequency (no need to run it at 915MHz).
b) Observe the I & Q steady state transient outputs of the mixer when driving it in the signal and image band (change the RF input to have ±Δf from the LO), what is the difference ? how do you distinguish between the two frequencies? This is the basis of a zero-if frequency shift keying (FSK) transceiver where the "image" represents '0' and the "signal" the '1' or vice-versa!
c) Design a RC-CR 90° phase shifter (use the same R value as that of the mixer load, size C so that the gain of the RC and CR are equal at the mixer output frequency); one mixer has two RC loads and the second one two CR loads which are equal but together perform a 90° phase shift of the mixer output signals.
d) Connect the outputs of the RC CR together in a differential way to a 0V voltage source and plot the voltage source current (this would be equivalent to a differential TIA input) for the two input tones; what do you observe?
e) You have just implemented a different way of performing image rejection
f) ! However due to components imperfection, the rejection will be limited; try to mismatch the quadrature LO by +-5° in a parametric simulation (80, 85, 90, 95,100 and 260 .. 280) and then separately the gain of the RC CR by +-10% by varying the cap. What do you observe on the rejection ?