Systematic Design Procedure for Q-Enhanced Integrated LC Filters

Florent D’halluin\textsuperscript{1,2}, Nicolas Beilleau\textsuperscript{1} and Hassan Aboushady\textsuperscript{1}
\textsuperscript{1}University of Paris VI, Pierre & Marie Curie, LIP6-SoC laboratory, 75252 Paris, France
\textsuperscript{2}Ecole Polytechnique, 91128 Palaiseau, France

Abstract—In this paper, we present a systematic design procedure for Q-enhanced integrated LC filters. It does not require any simulations and is thus suitable for design automation. The design procedure has been described in the CAIRO+ analog design environment, containing the BSIM3v3 models of the MOS transistor. Precise estimations of the quality factor and the resonance frequency were made possible by adding the integrated inductance $\pi$ model into the design environment. Several design examples of 2.442GHz Q-enhanced LC filters are given in a 0.13 $\mu$m CMOS process.

I. INTRODUCTION

Advances in highly integrated wireless communication transceivers provides potential applications for integrated RF bandpass filters. Active filters can achieve a high quality factor but with a poor dynamic range when operating at gigahertz frequencies. Passive LC filters can achieve high dynamic range at very low power consumption but requires high Q on-chip inductors.

Q-enhanced LC filters are a good compromise between these two types of filters [1]. Q-enhanced LC filters are not only used to realize integrated RF bandpass filters [2] but they are also used in the design of RF bandpass $\Sigma\Delta$ modulators [3]. Fig.1 shows a simple and popular realization of a Q-enhanced LC filter using a differential negative resistor. The design of such a circuit usually requires a significant amount of iterations with a SPICE-like circuit simulator. This is due to the fact that manual design equations are based on the parallel RLC inductor model, shown in Fig.1, while on-chip inductors models in circuit simulators are usually based on the more elaborate $\pi$ model [4], shown in Fig.2. In this work, we propose a systematic design procedure for Q-enhanced LC filter that does not require any iterations with a circuit simulator. The idea is to calculate, in the design phase, the quality factor of the original lossy LC tank. The design procedure is described in the analog design environment CAIRO+ [5].

II. QUALITY FACTOR ENHANCEMENT

Due to its nature, an integrated inductor presents some parasitics resistances and capacitances. Resistances are due to intrinsic resistance of the employed metal and interconnect and skin effect at high frequencies, and lossy substrate. The inductor with these parasitics can be modeled by the $\pi$ model Fig.2. As shown in Fig.3, we approximate this $\pi$ model with a RLC model for a certain range of frequency, for example the desired central frequency of the filter. In this case we take:

\[ R_{pl} \approx \frac{R_s}{(\omega_0 L)^2} \quad R_{pc} \approx \frac{1}{R_{sub}(C_{sub}\omega_0)^2} \]

\[ R_p = R_{pl} / R_{pc} \]  

(1)

Then we can replace the $\pi$ model by a RLC bridge with a $R_p$ resistor, a $C_{sub}$ capacitor and $L$ inductor. Once we have a value for a parallel resistor, we can obtain a value for the quality factor of the inductor by:

\[ Q_0 = \frac{L\omega_0^2}{R_p} \]
Using the differential negative resistance circuit shown in Fig.1, the quality factor is now equal to:

$$Q_{\text{enh}} = \frac{Q_0}{1 - G_{m_0}R_p}$$  \hspace{1cm} (2)

So choosing an appropriate value for $G_{m_0}$ can improve the value of the quality factor of the original lossy LC tank from $Q_0$ to a desired quality factor $Q_{\text{enh}}$. $G_{m_0}$ is the equivalent transconductance of the differential negative resistance, and is equal to:

$$G_{m_0} = G_m - G_{ds}$$  \hspace{1cm} (3)

where $G_m$ and $G_{ds}$ are respectively the transconductance and the output conductance of transistor M1. In the CAIRO+ analog design environment, it is possible to perform an accurate sizing of a transistor in order to obtain a certain transconductance $G_m$. Once we have the dimensions of the transistor, it is also possible to extract the precise value of $G_{ds}$. A few iterations are then required to size transistors M1 and M2 according to the required overall $G_{m_0}$ of the differential negative resistance.

## III. Resonance Frequency

As will be shown later in section V, the approximation by a parallel resistor is sufficiently accurate for the quality factor calculations. Unfortunately, it is not the case for the resonance frequency calculations. In Fig.4, we plot the total impedance calculated with the parallel and the series resistor versus frequency. We can see with the series resistor, the resonance frequency is shifted by 10 MHz from the standard $\omega_0 = 1/\sqrt{LC}$ value. Using a series resistor model, the total impedance is:

$$Z(j\omega) = \frac{(-1/G_{m_0})(R_s + j\omega C)}{-j\omega C(R_s + j\omega L)/G_{m_0} + R_s + j\omega L - 1/G_{m_0}}$$  \hspace{1cm} (4)

It is difficult to obtain an analytical expression for resonance frequency from this formula. A numerical method is used to compute the required capacitor value for a desired resonance frequency. This procedure gives us a value: $C_{\text{total}}$ for the total capacitance at the input node. In order to adjust the resonance frequency of the filter, the total parasitic capacitances, $C_{\text{par}}$ due to the inductor and the transistors have to be taken into account:

$$C_{\text{par}} = C_{\text{sub}} + C_{dg} + C_{gd} + C_{gs} + C_{ds}$$  \hspace{1cm} (5)

The MOS parasitic capacitances are easily obtained, using the BSIM3v3 model. $C_{\text{sub}}$ is obtained from the inductor $\pi$ model. Both models are embedded in the CAIRO+ design environment.

## IV. Design Automation and Characterisation

### A. CAIRO+ Design Environment

CAIRO+\(^1\), is a framework, developed at the LIP6 laboratory, which aims at helping analog circuit designers to automate the design of their circuits [6]. It provides a library of functions to describe the netlist template, layout template, specification template, design space exploration procedure and layout generation. The general method in CAIRO is to design Modules using devices. Each module has a list of defined parameters, and one or more procedure.

In our case, the goal is to design a bandpass Q-enhanced LC filter. The input parameters are: the desired quality factor, the resonance frequency, the input bias, and the value of the inductor with its geometrical parameters. The procedure will calculate and returns the sizes of the transistors, and the value of the capacitor of the resonator. A flowchart of the proposed design procedure, described in section II and III, is illustrated in Fig.5.

\(^1\)CAIRO stands for: “Optimized and Reusable Integrated Analog Circuits” in French.
B. Noise Characterization

Noise performance is an important characteristic of the designed Q-enhanced LC filter. We have also implemented a method to compute the noise generated by the filter. The noise sources in the circuit are represented in Fig.6. The thermal noise generated by transistors \( I_{n}^{2} \) is computed using Cairo’s transistor device, which include a procedure to compute thermal noise. The thermal noise of the resonator is computed using the total parallel resistor as approximated in section II so:

\[
I_{n}^{2} = 4k_{B}T \frac{R_{p}}{R_{n}} \tag{6}
\]

We have two current noise sources, we add the two spectral noise density to obtain the total current noise spectral density at the input. We can compute the equivalent voltage noise at the output by multiplying it by the total gain (the total impedance at the given frequency of the filter) and integrate it over the useful bandwidth (BW) to have the output noise power:

\[
P_{n} = \int_{BW} |Z(\omega)|^{2} \left( I_{n}^{2} + I_{n}^{2} \right) df \tag{7}
\]

V. SIMULATION RESULTS

Once the filter is sized, we can export a netlist file, which can be simulated. Here are the results of those simulations.

A. Quality factor and resonance frequency

In Fig.7 we plot the impedance versus input frequency for several values of Q. this way we could measures effective Q and resonance frequency. The parameters are: resonance frequency : 2.442 GHz, 5 nH inductor (4 turns, 12 µm width and 10 µm spacing, which give a \( Q_{0} = 15 \) ) for a 0.13 µm technology. The bias conditions are : \( V_{\text{in}} = 0.6V \), \( V_{\text{ds}} = 0.3V \). As we can see, the automatic design procedure is precise for the effective quality factor and the frequency.

B. Linearity

One fact this work has permit to underline is the importance of input bias voltage on linearity. As it is easy to generate filter sized to work with different input bias, we studied the linearity of those filters by plotting the output power versus input power in Fig.8 (\( V_{\text{ds}} \) stays at 0.3V). We also studied the intermodulation (IM3) with a two-tones signal. In Fig.9, we can see that the input voltage bias has a strong influence on the 1dB compression point.
C. Comparison with other works

A figure of merit to characterize the Dynamic Range of a filter, taking power consumption into account, is proposed in [8]:

\[ FOM = \frac{P_{\text{dB,1Hz}}}{P_{\text{noise,1Hz}} P_{\text{dc}}} \]  

(8)

where \( P_{\text{noise,1Hz}} \) is the output noise power spectral density in 1 Hz bandwidth. In our case we take the power density at the central frequency of the filter, and \( P_{\text{dc}} \) is the power consumption of the filter. In table II, we compare the performance of a filter generated using our design procedure with some other recent designs.

### VI. Conclusion

In this paper, we have presented a systematic design procedure for Q-enhanced LC filters. The procedure is based upon the inductor \( \pi \) model and the BSIM3v3 transistors’ model. It has been shown that a parallel RLC model is sufficient to calculate the required conductance of the differential negative resistance. On the other hand, a series model for the inductances losses is necessary for precise calculation of the resonance frequency. Several design examples have been presented to demonstrate the validity of the approach. Very little difference has been observed between the estimated performances of the circuits generated using the proposed design procedure and the performances measured from simulation.

### REFERENCES


