An inductor modeling and optimization toolbox for RF Circuit Design

F. Passos, E. Roca, R. Castro-López, F. V. Fernández

Abstract—This paper describes the SIDe-O toolbox and the support it can provide to the radio-frequency designer. SIDe-O is a computer-aided design toolbox developed for the design of integrated inductors based on surrogate modeling techniques and the usage of evolutionary optimization algorithms. The models used feature less than 1% error when compared to electromagnetic simulations while reducing the simulation time by several orders of magnitude. Furthermore, the tool allows the creation of S-parameter files that accurately describe the behavior of inductors for a given range of frequencies, which can later be used in SPICE-like simulations for circuit design in commercial environments. This toolbox provides a solution to the problem of accurately and efficiently optimizing inductors, which alleviates the bottleneck that these devices represent in the radio-frequency circuit design process.

Keywords—inductor design; single-objective optimization; multi-objective optimization; surrogate modeling; radio-frequency circuit design

1. Introduction

Designing a radio-frequency (RF) circuit is one of the most challenging tasks in nowadays electronics, due, partially, to its demanding specifications and convoluted trade-offs. To help managing this complexity and cut short the number of re-design cycles, an extensive effort is being made by the research community, over the past few years, to develop CAD tools that can efficiently support the RF circuit designer. However, with always increasing time-to-market demands, where time consuming re-design cycles are not desired and first-pass success is a design goal, new CAD tools have to be proposed to assist the designer in reducing the circuit design time while meeting the ever demanding RF specifications.

One of the most difficult RF components to design is the inductor, especially at gigahertz frequencies. The inductor is one of the most used components in RF circuits, vastly present in circuits such as low noise amplifiers (LNAs), voltage controlled oscillators (VCOs) or power amplifiers (PAs). When the use of integrated inductors is needed, RF designers usually rely on two different methods: either libraries provided by the foundry are used (which typically have limited options) or iterative electromagnetic (EM) simulations are performed until an inductor with desired performances is obtained (which is a computationally expensive method, critically slowing down the RF design cycle).

The complexity in designing inductors comes from the peculiar features that the RF designer has to control through the geometrical parameters that characterize the inductor (topology, number of turns, inner diameter, turn width, among others). When designing inductors, there are several alternatives to achieve a given inductance. Therefore, several design decisions have to be made in order to design the inductor that meets the circuit requirements with the highest quality factor and the smallest area for a given inductance. These decisions highly depend on the expertise of the RF designer and the technology process. To overcome these difficulties, optimization-based design methodologies can assist RF designers in order to design inductors while ensuring the best trade-offs for each design.

To carry out such assistance, several simulation tools for inductor design and optimization have been reported in the literature, which are intended to reduce the simulation time. Examples of these tools are ASITIC [4] and SISP [5]. Nevertheless, these tools are based on physical/analytical models, which typically present severe accuracy issues in some areas of the design space and at higher frequencies [6]. Therefore, these tools are not suitable for proper inductor design or optimization.

Currently, foundries and EDA companies provide tools for inductor design and optimization, but with some drawbacks: either analytical models are used to model inductors, presenting accuracy issues, or limited optimization options are provided to the user (e.g., allowing only the selection of the desired inductance, disregarding quality factor and inductor area). Therefore, a CAD tool with accurate and efficient inductor models and with extensive optimization options is still required. This is the scope of this work: to present a tool that allows the user to efficiently design inductors using accurate surrogate modeling techniques and also provides vast optimization options. Thus, the designer can either design the most suitable inductor for his application or explore the complex trade-offs inherent to inductors. In the presented toolbox, both single- and multi-objective optimization algorithms with different objectives are available, increasing therefore the options for the RF designer.
The paper is organized as follows. Section II provides detailed explanations of the toolbox features, modeling techniques, optimization algorithms, as well as its graphical user interface. Section III demonstrates the features of the toolbox by using it as a support in circuit design examples and, finally, in Section IV, conclusions are drawn.

2. SIDe-O: Surrogate Inductor Design and Optimization

This Section describes the SIDe-O toolbox. First, the toolbox features are presented. Second, the tailored strategy used in order to build accurate surrogate models is described and the optimization algorithms used are presented. The last part of this Section details the graphical user interface (GUI).

2.1 SIDe-O Features

One of the features of SIDe-O is its ability to simulate individual inductors. The toolbox allows the user to simulate three different inductor topologies for a given working frequency and also to draw the inductance and quality factor curve along a wide frequency range. To achieve efficient and accurate results, SIDe-O provides inductor models based on surrogate modeling techniques. Such techniques are recently being used to model complex structures that usually had to be electromagnetically simulated in order to obtain accurate results (e.g., filters, antennas, inductors, transformers, etc.) [7][8]. In the next subsection, the techniques and strategy used in this work will be described in detail.

SIDe-O also allows the user to create his/her own surrogate models for an easy migration to technologies and inductor topologies other than the ones already provided by the toolbox.

One of the main advantages of this toolbox is that it allows different and complex optimization options. The toolbox allows single-objective optimizations with different objectives, such as quality factor maximization, area minimization and the option of performing both the maximization of the quality factor and the minimization of the area by means of a weighted function.

Another improvement is the fact that SIDe-O allows multi-objective optimizations. By using multi-objective optimizations, the complex inductor performance trade-offs can be efficiently and automatically obtained by generating the so-called Pareto-optimal front (POF). Also, by using multi-objective optimizations, bottom-up design methodologies are enabled, which have proved to allow performance enhancements (e.g., a reduction in circuit area by 50%) [9]. The SIDe-O toolbox is, to the best of our knowledge, the first toolbox to ever allow multi-objective optimization of integrated inductors.

2.2 Modeling Strategy

In this section the modelling of integrated inductors is discussed and the proposed strategy used for an accurate modelling is described. A typical inductor layout is shown in Fig. 1. The geometry of this inductor is usually defined by four geometric parameters: number of turns \( N \), inner diameter \( D_{in} \), turn width \( w \) and spacing between turns \( s \).

Depending on the application, the designer may be interested in calculating inductance \( L \) and quality factor \( Q \) (e.g., for optimization purposes, where a given value of \( L \) must be achieved), or the S-parameters (e.g. for frequency-domain simulations).

The inductance \( L \) and quality factor \( Q \), are defined as:

\[
L(f) = \frac{\text{Im}[Z_{eq}(f)]}{2\pi f} \quad (1)
\]

\[
Q(f) = \frac{\text{Im}[Z_{eq}(f)]}{\text{Re}[Z_{eq}(f)]} \quad (2)
\]

where \( f \) is the frequency and \( Z_{eq} \) is the equivalent input impedance. The equivalent input impedance can be easily obtained from the S-parameters of the two-port structure representation of the inductor [10].

In Fig. 2, the inductance and quality factor as a function of frequency for three different inductors are shown. An important parameter is the self-resonance frequency \( SRF \), which is defined as the frequency at which the imaginary part of \( Z_{eq} \) is zero, or, equivalently, the frequency at which
the behaviour of the inductor changes from inductive to capacitive (see Eq. (1)).

Surrogate modeling is an engineering method used when an outcome of interest of a complex system cannot be easily (or cheaply) measured either by experiments or simulations. Therefore, an approximate model of the outcome is used instead. In this work a new tailored surrogate modeling strategy is used explicitly for integrated inductors [6]. Using this strategy, two sets of surrogate models were developed. The first set is used to model \( L \), \( Q \) and \( SRF \), which are the most important performance parameters for inductors. The second set of models is used to model the S-parameters of the inductors.

Generating a surrogate model usually involves the following four steps:

1) **Design of experiments:**
   The objective of surrogate models is to emulate the output response of a given system. Therefore, the model has to learn how the system responds to a given input. So, the first step in generating surrogate models is to select the input samples from which the model is going to learn. These samples should cover the design space evenly, so that it can be accurately modeled. To perform this sampling, different techniques are available, from classical Monte Carlo to Quasi-Monte Carlo or latin hypercube sampling (LHS) [11]. In this work, LHS is used.

2) **Accurate expensive evaluation:**
   Surrogate models learn from expensive evaluations. Therefore, it is important that these simulations have the best accuracy possible. In our work, these accurate evaluations are EM simulations, which are performed by using Keysight’s ADS Momentum simulator [12]. Depending on the size of the training set, these simulations could last one/two weeks, which can be considered as expensive. However, these simulations are only performed once for a given technology, therefore being useful for several years, as technology nodes do not become obsolete immediately after release. Any given technique can later be used in order to build a new model using the same training set. This means that these accurate simulations should not be accounted as time to build a given model as they are only performed once and can be used to build several different models using different techniques.

3) **Model construction:**
   This concerns the core functions used to build a surrogate model. Literature reports approaches based on artificial neural networks, support vector machines, parametric macromodels, Gaussian-process models, etc. Ordinary Kriging models, available in Matlab toolbox DACE [13], are used in this work.

4) **Model validation:**
   Many different techniques may be used in order to validate the model and assess its accuracy e.g. cross-validation, bootstrapping and subsampling techniques [14]. In this work, in order to validate the model, a set of points was generated independently of the training samples. These samples will be referred to as test samples and were also generated using LHS.

   To build the models used in SIdE-O, 800 inductors (for each topology) were generated as a training set using LHS and electromagnetically simulated using ADS Momentum. The technology selected was a 0.35-μm CMOS technology, for which the process information required for EM simulation was available. The inductor geometric variables were allowed to vary in the following ranges: \( N \in [1, 8] \), \( D_{in} \in [10, 300] \) μm, \( w \in [5, 25] \) μm, under the constraint that the outer diameter, \( D_{out} \), is below 400 μm. The spacing between turns, \( s \), was kept fixed at the minimum value \( s \approx 2.5 \) μm as no performance improvement is obtained for larger values [15]. Afterwards, Kriging functions using DACE were used in order to build the surrogate models following a tailored strategy.

   Concerning the strategy, the idea is to build several different surrogate models, one for each number of turns (e.g. one model for inductors with two turns, another for inductors with three turns, etc.), and one for each frequency point. This strategy increases the overall accuracy and efficiency of the model [6]. Surrogate models based on Gaussian processes assume continuity: if an input variable changes by a small quantity, the output varies smoothly. However, this is not the case for inductance if inductors have their self-resonance frequency close to the frequency of operation (see how sharply \( L \) changes around 2-3 GHz in Fig. 2). Some inductors with high number of turns may have their self-resonance frequency below or around the desired frequency of operation, therefore the modeling may still show some inaccuracy due the abrupt change in \( L \) around this frequency. Therefore, in order to solve this problem, the modeling strategy is based on a two-step method:

   1) Surrogate models for the self resonance-frequency (for each number of turns) are created using all training inductors.
   2) Only those inductors from the training set whose self-resonance frequency is sufficiently above the working frequency are used for the creation of \( L \) and \( Q \) (or S-Parameter) models.

   For example, if the working frequency \( WF \) is 2.5 GHz, only inductors with \( SRF \geq 3 \) GHz are used to generate \( L \) and \( Q \) (or S-Parameter) models.

   However, if a new model has to be generated for higher working frequencies, the same training set can be used and the generation of a new model takes less than one second (per frequency point).

   Using this strategy, the toolbox allows a very accurate
modeling of inductor performances. The models provide less than 1% error when compared with EM simulations. The modeling technique and the accuracy of one of the models used in this toolbox (for one of the inductor topologies) were presented in [6]. Likewise, the models for the other topologies present similar errors.

2.3 Optimization Algorithms

In this section the optimization algorithms used in SIDe-O are presented. In the toolbox, single-objective optimizations are performed with the selection-based differential evolution algorithm (SBDE) [16] and the objectives of the optimization can be freely changed. As previously said, the toolbox allows the user to maximize the quality factor, minimize the area, or both (by means of a weighted function where the weights of the function can be selected by the user) while achieving a given target inductance. The multi-objective optimizations are performed with NSGA-II [17] implemented in the NGPM toolbox [18]. Two different multi-objective optimizations can be performed: a two-objective optimization, maximizing quality factor and inductance, or a three-objective optimization, where quality factor and inductance are maximized and area is minimized.

In both optimization algorithms (SBDE and NSGA-II), constraints are applied in order to guarantee that the selected inductors can operate at the chosen working frequency. These constraints are specified in the following set of equations.

\[
\begin{align*}
\text{area} &< 400\mu m \times 400\mu m \\
\frac{L@WF - L@WF+0.05GHz}{L@WF} &< 0.01 \\
\frac{L@WF - L@WF-0.05GHz}{L@WF} &< 0.01 \\
\frac{L@WF - L@WF\text{ at }0.1GHz}{L@WF} &< 0.05 \\
Q@WF @0.05GHz - Q@WF &> 0
\end{align*}
\]

where, \(L_{@WF}\) and \(Q_{@WF}\) are the inductors’ inductance and quality factor at the working frequency (WF) and \(L_{@WF@0.05GHz}\) and \(Q_{@WF@0.05GHz}\) are the inductance and quality factor at WF±0.05 GHz.

These constraints are used in order to ensure that the inductance is sufficiently flat from around DC to slightly above the working frequency, and that the self-resonance frequency is sufficiently above this frequency [15].

2.4 SIDe-O Graphical User Interface

In this section, a brief description of the SIDe-O graphical user interface (GUI) is given. The complete interface can be observed in Fig. 3. Since the models were developed in Matlab, for a straightforward integration, the GUI was also developed in Matlab. The GUI is multi-tabbed, with each tab suited for a different operation.

The first tab in Fig. 3 a), Inductor Simulation, allows the user to simulate individual inductors with different topologies and geometries. The user has to select a given working frequency, the geometric parameters of the inductor and the frequency range up to which the performances are desired to be drawn. In this tab, the user can also generate a S-Parameter file for any given inductor. This S-Parameter file can afterwards be used in a modern circuit simulator, such as HspiceRF [19] or SpectreRF [20] for an accurate description of the inductor behavior. For example, the analoglib library of Cadence has a device particularly for this purpose, the nport. Since the models are so accurate, by using the S-parameter file generated by the toolbox, the inductor behavior is modeled with less than 1% error when compared with EM simulations.

The second tab in Fig. 3 b), inductor optimization, allows the user to perform single- and multi-objective optimization of inductors. For the single-objective optimization, besides the inductor topology, the user has to select the desired inductance, working frequency and optimization objective. After the optimization is completed, the geometry of the optimized inductor immediately appears in the same tab. For the sake of simplicity, the toolbox also allows the geometry of the optimized inductor to be loaded directly into the inductor simulation tab, so that the performances of the

![Fig. 3. SIDe-O graphical user interface shown in separate tabs, a) inductor simulation, b) inductor optimization and c) build models.](image-url)
optimized inductor can be observed plotted as a function of the frequency.

Regarding the multi-objective optimization, the user only needs to select the desired inductor topology, the working frequency and the set of objectives. When a multi-objective optimization begins, the NGPM toolbox launches another window showing the current information about the multi-objective optimization (e.g., elapsed time, current POF approximation, etc).

The third tab in Fig. 3 c), Build Model, allows the user to build his/her own models by providing a training set. By providing a test set it is also possible to validate the model automatically with the toolbox. The toolbox allows the user to build his/her own models for new technologies and different topologies from the ones already supported. Moreover, once the new model is built, it is automatically included as an option in the topologies popup menu of the other tabs and suitable for immediate simulation and optimization.

The toolbox provides a high level of automation with a fully automated inductor simulation, single- and multi-objective optimization (with different objectives) and the generation of new models for different technologies and inductor topologies. The toolbox allows all these operations with no necessary action from the user beyond the design requirements (e.g., inductor geometry, topology, working frequency, number of generations for a given optimization, etc.).

All tabs have a message board and a README file, which are appropriate for an easy toolbox usage.

3. Using SIDe-O in Circuit Design
In this Section, the different features of the toolbox are illustrated with real design problems, demonstrating the advantages that the toolbox presents against other methodologies (e.g., based on EM simulations). As previously seen in Section 2, the toolbox can be used for the simulation of an individual inductor and it can perform both single- and multi-objective optimizations. Therefore, these features will be explored in this section.

The computer used for all toolbox simulations was an Intel Core i7 @ 1.7GHz and all EM and circuit simulations were performed in a 6-core Intel® Xeon® E5-2630 v2 processors @ 2.60GHz.

3.1 Inductor simulation
The toolbox allows the user to simulate a single inductor for several different topologies (e.g. symmetrical octagonal, asymmetric octagonal, etc.). For the simulation of a given inductor the user has to provide the inductor geometry (N, D, and w), the working frequency, the frequency range (maximum frequency to which the inductor should be simulated) and the inductor topology. For this example, an asymmetric octagonal inductor with $N=5$, $D=130 \mu m$ and $w=13.65 \mu m$ was simulated for 500 frequency points between 100 kHz and 10 GHz. Afterwards, in order to compare accuracy and efficiency of the toolbox, the same inductor was simulated with the EM simulator for the same frequency points in the same range. The comparisons between both inductor curves obtained are shown in Fig. 4, where it is possible to observe the accuracy of the model in the entire frequency range. The inductor simulation with the toolbox lasted 21.71 seconds while the simulation with the EM simulator lasted 30.78 minutes (CPU time), proving therefore the efficiency of the toolbox.

3.2 Inductor single-objective optimization
This section illustrates the advantages of performing single-objective optimizations in the toolbox. One of the uses of single-objective optimizations is to assist the RF designer in top-down circuit design methodologies (see Fig. 5) [1]. In top-down design methodologies, the top level is first designed, resulting in the specifications for the sub-blocks. This process continues down to the lowest level (e.g., passive components).

When designing one of these sub-blocks where the usage of inductors is needed (e.g., VCOs) designers usually seek inductors with a given inductance value and the highest quality factor possible (e.g., to achieve a given oscillation frequency and increase the quality factor of the VCO tank) so that the desired performances are met.
Depending on the designers’ expertise, finding these performances can be challenging, therefore one of the options is to use single-objective optimization algorithms to obtain an inductor with a given inductance while maximizing the quality factor. The SIDe-O toolbox allows this operation.

Let us consider the design of the cross-coupled differential VCO shown in Fig. 6, with the desired specifications shown in Table 1. In the Inductor Optimization tab, the user can define a given inductor topology, the working frequency, the desired inductance (within a given margin) and optimization settings like the number of generations and individuals (see Fig. 7).

Henceforth, an optimization was made with SIDe-O with the objective of achieving a symmetrical octagonal inductor with $L=2$ nH ($\pm 0.05$ nH) at 2.5 GHz, while maximizing the quality factor. The geometrical parameters of the obtained inductor were $N=2$, $D_w=251$ µm, $w_w=22.2$ µm and its performance parameters are $L=1.98$ nH and $Q=15.22$. The entire optimization with 200 individuals and 40 generations takes around 10 seconds of CPU time. In order to compare accuracy and efficiency of the toolbox, the same optimization was performed with the EM simulator as a performance evaluator. The geometrical parameters of the inductor obtained with this method are $N=2$, $D_w=257$ µm, $w_w=25$ µm and its performance parameters are $L=2.00$ nH and $Q=15.57$ while the optimization lasted 108.11 hours of CPU time.

It is then possible to conclude that both methods achieve inductors with similar geometries and performances, however by using the toolbox the simulation time is outstandingly reduced (from 108.11 hours to 10 seconds). Hence, even if re-design cycles are needed, an optimization time of 10 seconds is an affordable time, something that does not occur with the EM simulator as a performance evaluator.

It may seem surprising that the optimization time and simulation time of only one inductor (using the toolbox, as shown in subsection 3.1) are not comparable. However, it has to be considered that the operation that takes longer is the loading of the models into MATLAB. For inductor simulation, 1000 models are loaded ($L$ and $Q$ models for 500 frequency points) and in optimization only 6 models are loaded (the six frequency models needed to ensure that constraints in Eq. (3) are met).

In order to inspect the behavior of the inductor in the entire frequency range it is possible to load the inductor variables into the Inductor Simulation tab and simulate the inductor (see Fig. 8). If the user is satisfied with the behavior of the inductor it is then possible to generate a $S$-parameter file in order to simulate the RF block. The

![Fig. 6. Cross-coupled differential VCO.](image)

![Table 1. VCO desired specifications and performances.](image)

<table>
<thead>
<tr>
<th>Performances</th>
<th>Specifications</th>
<th>SIDe-O</th>
<th>EM</th>
<th>Δ (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Phase Noise (dBc/Hz)</td>
<td>$&lt;-110$</td>
<td>$-126.38$</td>
<td>$-126.34$</td>
<td>0.04</td>
</tr>
<tr>
<td>Fosc (GHz)</td>
<td>2.50</td>
<td>2.50</td>
<td>2.50</td>
<td>0.12</td>
</tr>
<tr>
<td>Vout (mV)</td>
<td>$&gt;100$</td>
<td>410</td>
<td>408</td>
<td>0.48</td>
</tr>
</tbody>
</table>

![Fig. 7. Single-objective optimization inputs and results performed with the toolbox.](image)

![Fig. 8. Frequency behavior of the previously optimized inductor observed in the toolbox.](image)
Performances of the VCO designed using the optimized inductor can be observed in Table 1. In order to inspect the model accuracy, the obtained inductor (with the toolbox) was electromagnetically simulated and the circuit simulations were repeated in order to observe the performance shifts by using the model. The performance shifts can also be observed in Table 1, and it is possible to conclude that the shifts are negligible, confirming the model accuracy.

In Fig. 9 it is possible to observe comparisons between the performances of the VCO when simulated with the inductor obtained with the toolbox and the same inductor EM simulated. In Fig 9a) and 9c) the phase noise and transient simulation are shown. Since the performance shifts are almost negligible, in Fig. 9b) and 9d) a close up around the regions of interest is shown for the phase noise and transient simulations respectively. It can be concluded that the usage of the toolbox is very helpful to the RF designer since the optimizations are very efficient and the performance shifts are minimal when compared to EM simulations.

3.3 Inductor multi-objective optimization

The design example illustrated in this section, demonstrates the advantages of performing multi-objective optimizations with SIDe-O.

In multi-objective optimizations, the result of the optimization in not a single optimal inductor but a set of inductors called the Pareto-optimal front (POF). This POF characterizes the best trade-offs between previously selected objectives for a given inductor topology and working frequency. These trade-offs can be used by the designer for design space exploration. The POF generated in this optimization can also be used in the context of bottom-up design methodologies for circuit design (see Fig. 10) [9]. In bottom-up design methodologies the lowest level blocks are designed first, and then, the information of lower level sub-blocks is composed up the hierarchy.

Therefore, in this example, SIDe-O is going to be used in order to achieve a POF of inductors. Afterwards, using a single-objective optimization, a source-degenerated LNA (see Fig. 11) is going to be designed while the inductors will be automatically selected from the inductors’ POF, following a bottom-up design methodology.
The design methodology used can be observed in Fig. 12. The methodology consists on a two-step process: First, the inductor POF is generated using SIDe-O and second, the LNA is designed using a single-objective optimization algorithm. The work reported here does not exploit any specific characteristics of a given single-objective algorithm, hence, whichever algorithm can be used.

The first step is where the RF designer can be assisted by SIDe-O. For this step, a 3D optimization with the objective of maximizing $L$ and $Q$, while minimizing the inductor area was performed. SIDe-O window showing the multi-objective optimization is shown in Fig. 13. The 3D optimization in SIDe-O with 1000 individuals and 80 generations lasts 8.88 minutes while the same optimization using an EM simulator takes 1926.39 hours (roughly 80 days CPU time), proving once more the efficiency of the toolbox. In order to compare accuracy, the comparison between the POF obtained by the toolbox and the POF obtained with EM simulations can be observed in Fig. 14. It can be seen that the POFs are almost completely overlapped. As the POFs achieved by both methods are very similar, a more accurate comparison can be performed by using a well-known performance metric such as hypervolume [21]. The hypervolume is calculated as the sum of the hypervolumes determined by each point of the approximated POF and a reference point. As our goal is to compare the POFs generated with two different techniques, and the hypervolume metric depends on the selected reference point, the same reference point is used in both cases. The hypervolume of the POF generated with EM simulations is 9861.22 and the hypervolume of the POF generated with SIDe-O is 9736.84, which is very similar.

Through the methodology shown in Fig. 12, it is possible to understand that after obtaining the inductor POF the next step is to run the single-objective optimization to design the LNA with the desired specifications. The optimization was performed in a 0.35-µm CMOS technology and the LNA is intended to operate at the frequency band of 2.4-2.5 GHz, with a supply voltage $V_{dd}=1.5$ V. The design variables of the LNA can be observed in Table 2, where, $W_{1,2}$ are the transistor widths, $l_{1,2}$ are the channel lengths of the transistors, $V_b$ is the voltage used to bias the circuit, and $C_{1,2,3}$ are the three parallel-plate squared capacitors in the topology. For convenience and correspondence with the circuit layout, the design variable used is the side length of each capacitor. The lower and upper bounds defined in Table 2 correspond to 100fF and 5pF respectively.

In order to further compare the accuracy of the models used in the toolbox, two different LNA optimizations were performed: one where the inductors’ POF was used and another where the POF used was the one generated with the EM simulator as performance evaluator. Both LNA optimizations were performed in the

![Fig. 11. Source-degenerated LNA topology.](image1)

![Fig. 12. Flow diagram of the used methodology](image2)

![Fig. 13. 3D multi-objective optimization results performed with the toolbox.](image3)

<table>
<thead>
<tr>
<th>Variables</th>
<th>Min</th>
<th>Max</th>
<th>Step</th>
</tr>
</thead>
<tbody>
<tr>
<td>$l_{1,2}$</td>
<td>100</td>
<td>600</td>
<td>5</td>
</tr>
<tr>
<td>$V_b$ (V)</td>
<td>0</td>
<td>1.5</td>
<td>0.001</td>
</tr>
<tr>
<td>Inductors</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$C_{1,2,3}$ (pF)</td>
<td>10.6</td>
<td>76.05</td>
<td>0.05</td>
</tr>
</tbody>
</table>
same conditions using 40 particles and 1000 generations. The objective of the optimization was the minimization of the LNA area.

Table 3 presents the optimization constraints and the results obtained. The constraints imposed to the LNA are given in column 2 of Table 3. The inductor constraints are those in Eq. (3), which were imposed while generating the inductor POF. The performances of the obtained LNAs can be observed in column 3 (for the LNA obtained using the SIDe-O POF) and in column 6 (for the LNA obtained using the EM POF). By comparing these columns, it can be concluded that both optimizations achieved LNAs with similar performances and areas.

It has to be taken into account that the inductors obtained by SIDe-O may have some performance deviations due to the model errors. Therefore, the inductors were EM simulated and included in the LNA simulation in order to observe the performance shifts due to the model usage (see columns 4 and 5 in Table 3). It can be seen that the performance deviations are negligible, confirming once again the accuracy of the models. Some of the LNA performances are shown in Fig. 15, where it is possible to once more recognize the modeling accuracy of the toolbox.

<table>
<thead>
<tr>
<th>Specifications</th>
<th>Constraints</th>
<th>Performances using POF from SIDe-O</th>
<th>Performances using POF from SIDe-O (after EM)</th>
<th>( \Delta (%) )</th>
<th>Performances using POF from EM</th>
</tr>
</thead>
<tbody>
<tr>
<td>( S_{21} )</td>
<td>(&lt; -10 \text{ dB} )</td>
<td>-20.640 dB</td>
<td>-20.866 dB</td>
<td>1.085</td>
<td>-18.785 dB</td>
</tr>
<tr>
<td>( S_{22} )</td>
<td>(&lt; -10 \text{ dB} )</td>
<td>-20.580 dB</td>
<td>-20.3425 dB</td>
<td>1.167</td>
<td>-17.509 dB</td>
</tr>
<tr>
<td>( S_{21} )</td>
<td>( &gt; 17 \text{ dB} )</td>
<td>17.083 dB</td>
<td>17.001 dB</td>
<td>0.482</td>
<td>17.085 dB</td>
</tr>
<tr>
<td>( k )</td>
<td>( &gt; 1 )</td>
<td>9.801</td>
<td>9.942</td>
<td>1.418</td>
<td>9.530</td>
</tr>
<tr>
<td>NF</td>
<td>(&lt; 3.5 \text{ dB} )</td>
<td>3.448 dB</td>
<td>3.466 dB</td>
<td>0.536</td>
<td>3.448 dB</td>
</tr>
<tr>
<td>( P_{DC} )</td>
<td>(&lt; 10 \text{ mW} )</td>
<td>10 mW</td>
<td>10 mW</td>
<td>0</td>
<td>10 mW</td>
</tr>
<tr>
<td>IP3</td>
<td>(&lt; -10 \text{ dBm} )</td>
<td>-4.830 dBm</td>
<td>-4.815 dBm</td>
<td>1.350</td>
<td>-5.407 dBm</td>
</tr>
<tr>
<td>Inductors</td>
<td>Usable @ 2.4-2.5 GHz</td>
<td>yes</td>
<td>yes</td>
<td>-</td>
<td>yes</td>
</tr>
</tbody>
</table>

Fig. 14. Comparison between the POF obtained with EM simulations and the SIDe-O toolbox.

![Fig. 14. Comparison between the POF obtained with EM simulations and the SIDe-O toolbox.](image)

![Fig. 15. Comparisons of the LNA performances using the inductors obtained with the toolbox and the same inductors EM simulated. a) Gain (S21) comparisons. b) Input (S11) and output (S22) matching comparisons. c) Noise figure (NF).](image)
Table 4. Design variables for the LNA designs obtained from optimization using different inductor POFs.

<table>
<thead>
<tr>
<th>POF used</th>
<th>W₁ (µm)</th>
<th>W₂ (µm)</th>
<th>l₁ (µm)</th>
<th>Vₛ (V)</th>
<th>C₁ (µnF)</th>
<th>C₂ (µnF)</th>
<th>C₃ (µnF)</th>
<th>L₅ (µH)</th>
<th>L₆ (µH)</th>
<th>L₇ (µH)</th>
<th>L₈ (µH)</th>
<th>L₀ (µH)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIDe-O</td>
<td>600</td>
<td>455</td>
<td>0.35</td>
<td>0.757</td>
<td>23.95 (0.501pF)</td>
<td>33.35 (0.967pF)</td>
<td>11.35 (0.114pF)</td>
<td>D₅=10µm w=5µm</td>
<td>D₆=10µm w=5µm</td>
<td>D₇=26µm w=5µm</td>
<td>D₈=18µm w=5µm</td>
<td>N=1</td>
</tr>
<tr>
<td>EM</td>
<td>600</td>
<td>440</td>
<td>0.35</td>
<td>0.757</td>
<td>22.80 (0.454pF)</td>
<td>34.35 (1.026pF)</td>
<td>10.75 (0.103pF)</td>
<td>N=1</td>
<td>D₅=10µm w=5µm</td>
<td>N=1</td>
<td>D₆=25µm w=5µm</td>
<td>N=8</td>
</tr>
</tbody>
</table>

Table 5. Statistical results of the LNA area obtained in five runs of the single-objective optimization.

<table>
<thead>
<tr>
<th>POF used</th>
<th>Mean (µm²)</th>
<th>Standard Deviation (µm²)</th>
<th>Best (µm²)</th>
<th>Worst (µm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIDe-O</td>
<td>3.32 x10⁵</td>
<td>1678</td>
<td>3.29 x10⁵</td>
<td>3.69 x10⁵</td>
</tr>
<tr>
<td>EM</td>
<td>3.35 x10⁴</td>
<td>2790</td>
<td>3.33 x10⁴</td>
<td>3.97 x10⁴</td>
</tr>
</tbody>
</table>

Fig. 16. Inductors selected from the inductors’ POF generated with SIDe-O and used in the LNA circuit design.

In Table 4, it is possible to observe the obtained LNA designs with both methods and conclude that they are very similar. The capacitor values in Table 4, show the equivalent capacitance value together with the side length, that has been used as design variable. The selected inductors from the POF (in the case of the optimization performed with SIDe-O) can be seen in Fig. 16, where it is possible to perceive how the algorithm selected inductors with reduced area values in order to minimize the total area of the LNA.

Since optimization algorithms are stochastic operations, five independent runs were performed for the LNA optimization using both methods. The statistical analysis can be observed in Table 5, where it can be seen that the mean values achieved by both methods are similar.

The optimization of the LNA takes around 1.03 hours CPU time for both cases, however, the design time of the entire process (generation of inductors’ POF plus LNA optimization) can be significantly reduced by generating the inductors’ POF in the toolbox instead of performing EM simulations.

4. Conclusions

This paper presents a novel CAD toolbox for the design and optimization of integrated inductors based on surrogate modeling. The toolbox allows the simulation of a single inductor, single-objective optimizations and is the first toolbox to allow multi-objective optimization of inductors. The toolbox has proved to be very accurate and efficient in several circuit design experiments (design of VCOs and LNAs) in the context of top-down and bottom-up design methodologies. SIDe-O has demonstrated to be an accurate and efficient toolbox that can assist the RF designer in many different design examples.

Furthermore, the presented toolbox also allows the RF designer to create his/her own inductor surrogate models, which makes this toolbox usable for every technology node and inductor topology.

REFERENCES

inductors,” in IEEE Transactions on Computer-Aided Design of
Circuits and Systems, DOI: 10.1109/TCAD.2016.2564362

coefficient of two-port devices,” in Progress In Electromagnetics


1887116/momentum-s(op)/planar-em-simulator?cc=ES&lc=eng
[accessed, September 2016]


methods for meta-model validation with recommendations for
evolutionary computation”, Evolutionary Computation, vol. 20, No.
2, pp. 249-275, 2012.

performance trade-offs of integrated inductors,” in IEEE Transactions
on Computer-Aided Design of Integrated Circuits and Systems,
vol.33, no.8, pp. 1269-1273, 2014.

using differential evolution,” in IEEE congress on Evolutionary

multiobjective genetic algorithm: NSGA-II," in IEEE Transactions on

a-ngsa-ii-program-in-matlab-v1-4 [accessed, September 2016]

itSimulation/HSPICE-Pages/default.aspx [accessed, September 2016]

[20] Cadence SpectreRF circuit simulator. https://www.cadence.com/content/cadence-
www/global/en_US/home/tools/custom-ic-analog-rf-design/circuit-
simulation/spectre-rf-option.html [accessed, September 2016]

comparative case study and the Strength Pareto Approach,” IEEE
Transactions on Evolutionary Computation, vol. 3, pp. 257-271,
1999.

F. Passos was born in Lisbon in 1989. He received the B.S. and M.S. degrees from the New
University of Lisbon in 2011 and 2013 respectively. Fabio is currently pursuing his PhD
degree from University of Seville. He is conducting his work at the Institute of
Microelectronics of Seville (IMSE-CNMC). He received an outstanding paper award in
MIXDES 2013 and was the SMACD 2016 EDA competition winner. His
research interests include the modeling of passive RF devices, computer-
aided design and design methodologies for analog and RF circuits.

E. Roca received the Ph.D. degree in Physics from the University of Barcelona, Spain, in 1995. From
November 1990 to April 1995, she worked at IMEC, Leuven, Belgium, in the field of infrared detection
aiming to obtain large arrays of CMOS compatible silicide Schottky diodes. Since 1995, she has been
with the Institute of Microelectronics of Seville, (IMEC-CNMC-CSIC), Spain, where she holds the
position of Tenured Scientist. Her research interests lie in the field of modeling and design
methodologies for analog, mixed-signal and RF integrated circuits. She has been
involved in several research projects with different institutions: Commission of the EU, ESA, ONR-NICOP, etc. She has also co-authored
more than 90 papers in international journals, books, and conference proceedings.

R. Castro-López received the “Licenciado en Física Electrónica” degree (M.S. degree on Electronic
Physics) and the “Doctor en Ciencias Físicas” (Ph.D. degree) from the University of Seville, Spain, in 1998
and 2005, respectively. Since 1998, he has been working at the Institute of Microelectronics of Seville
(CSIC-IMSE-CNMC) of the Spanish Microelectronics Center, where he now holds the position of Tenured
Scientist. His research interests lie in the field of integrated circuits, especially design and computer-aided design for analog and mixed-signal
circuits. He has participated in several national and international R&D
projects and co-authored more than 50 international scientific publications,
including journals, conference papers, book chapters and the book Reuse-
based Methodologies and tools in the Design of Analog and Mixed-Signal

F.V. Fernández got the Physics-Electronics degree from the University of Seville, Spain, in 1988 and
his Ph. D. degree in 1992. In 1993, he worked as a postdoctoral research fellow at Katholieke
Universiteit Leuven (Belgium). From 1995 to 2009, he was an Associate Professor at the Dept. of
Electronics and Electromagnetism of University of Seville, where he was promoted to full professor in
2009. He is also a researcher at IMSE-CNMC (University of Seville and CSIC). Dr. Fernández
was the Editor-in-Chief of Integration, the VLSI Journal (Elsevier) from 2005 to 2015. His research interests lie in the design and design
methodologies of analog, mixed-signal and radiofrequency circuits. Dr. Fernández has authored or edited five books and has co-authored more than
150 papers in international journals and conferences. He has served as
General Chair of three international conferences and regularly serves at the
Program Committee of several international conferences. He has also participated as researcher or main researcher in numerous national and
international R&D projects.