Implementation of Geometry Dependent Planar Inductor and Transformer Models in Cadence PSpice

Vladislav Petrov Durev and Elissaveta Dimitrova Gadjeva

Abstract - **Parameterized *PSpice* macromodels are developed in the paper for accurate simulation of spiral inductors and planar transformers. The macrodefinitions describing the models are presented. A verification of the developed macromodels is performed by comparison of the simulation results for the *S*-parameters using *PSpice* and measurement data given in [1, 6]. The developed macromodels can be used successfully in the RF circuit design process.**

Keywords – **Planar Inductors and Transformers, Computer Macromodels, *PSpice* Simulation, *S*-parameters.**

I. Introduction

With the rapid developments of wireless communication market, high-performance and low product cost on-chip radio frequency (RF) devices are needed. Widespread in developing of RF circuits are the high frequency blocks like planar inductors and transformers. Therefore, many researches had been conducted on various topics to improve or to control the properties of spiral inductors and transformers. For this reason, the analysis and optimization of these elements have been of great importance. Considerable research work has been done over the past several years [1-8]. The majority of the methods are based on numerical techniques, empirical formulas, and physical models. The geometry-dependent models use the simple π-structure and expressions for the series resistance which are functions of operating frequency. Many researches are concentrated on the development of accurate expressions describing the spiral inductors and transformers and on geometry optimization for improving the Q factor [7, 8]. Scalable inductor models are developed, which can be easily integrated into a CAD framework. The computer-aided design and optimization of RF integrated circuits requires the development of accurate, effective and simple computer models of the basic RF building blocks.

In the present paper, parameterized computer macromodels are developed for accurate simulation of spiral inductors and transformers applying the standard circuit simulator *Cadence PSpice* [9]. A verification of the developed macromodels is performed by comparison between the simulation results obtained using *PSpice* and measurement data given in [1,6]. The developed macromodels are easy for implementation in the standard *PSpice*-like circuit simulators and can be used successfully in the RF circuit design process.

V. Durev is with the Department of Electronics and Electronics Technologies, Faculty of Electronic Engineering and Technologies, Technical University of Sofia, 8 Kliment Ohridski blvd., 1000 Sofia, Bulgaria, e-mail: v_p_durev@yahoo.com

E. Gadjeva is with the Department of Electronics and Electronics Technologies, Faculty of Electronic Engineering and Technologies, Technical University of Sofia, 8 Kliment Ohridski blvd., 1000 Sofia, Bulgaria, e-mail: egadjeva@tu-sofia.bg

II. Computer PSpice Model of Spiral Inductor

The equivalent circuit with a π-structure of the spiral inductor [1] is used. The *PSpice* implementation of the model is shown in Fig. 1. The model parameters of the spiral inductor are R_s, C_s, R_{si}, C_{si}, C_{ox}, L_s.

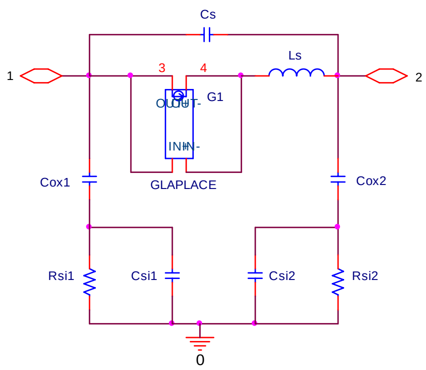

Fig. 1. *PSpice* computer macromodel of the spiral inductor

The inductor parameters are defined in Table I. The series resistance R_s is represented by the VCCS of GLAPLACE type. It is frequency dependent describing the influence of the skin effect. The circuit elements are defined by the following equations [1]:

$$R_s = \frac{l}{\sigma w \delta \left(1 - e^{-t/\delta}\right)}\ ;\ C_s = \frac{\varepsilon_{ox} n w^2}{t_{oxM1M2}}\ .;\ \sigma_{Al,Cu} = 1/\rho_{Al,Cu} \qquad (1)$$

$$\delta = \sqrt{\frac{2}{\omega\sigma\mu}}\ ;\ R_{si} = \frac{2}{G_{sub} l w}\ ;\ C_{ox} = \frac{\varepsilon_{ox} l w}{2 t_{ox}}\ ; C_{si} = \frac{C_{sub} l w}{2}$$

The inner diameter can be found using the expression:

$$D_{in} = D_{out} - 2\left[n\left(sp + w\right) - sp\right] \qquad (2)$$

The average diameter D_{avg} can be found using the values for the inner diameter D_{in} and the outer diameter D_{out}:

$$D_{avg} = \left(D_{in} + D_{out}\right)/2 \qquad (3)$$

The length of the spiral can be found using the expression:

$$l = 4 D_{avg} n\ . \qquad (4)$$

The inductance L_s of the planar spiral inductor is described using three analytical expressions in dependence on the geometry parameters [5].

TABLE 1. INPUT PARAMETERS FOR THE INDUCTOR SIMULATION

Inductor parameter	Value
Outer diameter D_{out}	300×10^{-6} m
Number of turns n	7
Width of spiral trace w	13×10^{-6} m
Metal thickness t	1×10^{-6} m
Line spacing sp	7×10^{-6} m
Thickness of the oxide insulator between the spiral and underpass t_{oxM1M2}	1.3×10^{-6} m
Thickness of the oxide layer between the spiral and substrate t_{ox}	4.5×10^{-6} m
Metal resistivity at DC for Al ρ_{Al}	3×10^{-8} Ω.m
Metal resistivity at DC for Cu ρ_{Cu}	2×10^{-8} Ω.m
Substrate conductance G_{sub}	4×10^{4} S/m2
Substrate capacitance C_{sub}	1.6×10^{-6} F/m2
Permittivity of the oxide ε_{ox}	3.451×10^{-11} F/m
Magnetic permeability of the free space μ	1.256×10^{-6} H/m

The first approximation is realized with the modification of the Wheeler formula with K coefficients:

$$L_s = K_1\mu_0\frac{n^2 d_{avg}}{1+K_2\rho}, \qquad (5)$$

where K_1 and K_2 depend on the inductor layout. In the case of square inductors K_1 =2.34 and K_2 = 2.75. The second approximation is done using current sheet approximation:

$$L_s = \frac{\mu n^2 d_{avg} c_1}{2}(\ln(c_2/\rho)+c_3\rho+c_4\rho^2) \qquad (6)$$

The coefficients c_1 to c_4 depend on the shape of the inductor. In the case of square inductors c_1=1.27, c_2= 2.07, c_3= 0.18 and c_4= 0.13. The third approximation is given with the α – coefficients expression:

$$L_s = \beta\, d_{out}^{\alpha_1} w^{\alpha_2} d_{avg}^{\alpha_3} n^{\alpha_4} sp^{\alpha_5} \,. \qquad (7)$$

In the case of square spiral inductor β=1.62×10^{-3}, α_1=–1.21,α_2=–0.147, α_3 = 2.4, α_4 = 1.78 and α_5= – 0.03 [5].

The series resistance R_s is modeled according to the input language of the *PSpice* simulator using the element voltage controlled current source (VCCS) GLAPLACE from the *ABM.lib* library. It allows to include the Laplace variable s in the expression for the controlling parameter.

As the frequency is not directly accessible for the user in *PSpice*, the following expression is applied to calculate ω from s variable:

$$\omega = \sqrt{-s^2} \qquad (8)$$

The macroses which are describing the parametrized model of the spiral inductor in *PSpice* are in the form:

```
Ls = {((@mju*@n*@n*@Davg*@C1)/2)*(log(@C2/@ro)
  +@C3*@ro+@C4*@ro*@ro)}
Ls = {(@K1*@mju*@n*@n*@Davg)/(1+@K2*@ro)}
Ls = {@beta*pwr(@Dout*1e6,@al1)*pwr(@w*1e6,@al2)*
  pwr(@Davg*1e6,@al3)*pwr(@n,@al4)*pwr(@sp*1e6,@al5)*1e-9}
Cs = {@n*@w*@w*@Eox/@toxM1M2}
Rs = {(@sigma*@w*sqrt(2/(sqrt(-s*s)*@mju*@sigma))*
  (1-exp(-@t/(sqrt(2/(sqrt(-s*s)*@mju*@sigma))))))}
Cox = {0.5*@L*@w*@Eox/@tox}
Rsi = {2/(@L*@w*@Gsub)}
Csi = {0.5*@L*@w*@Csub}
```

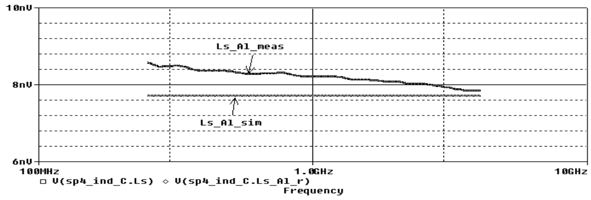

Fig. 2. The series inductance L_s vs frequency for *Al* spirals

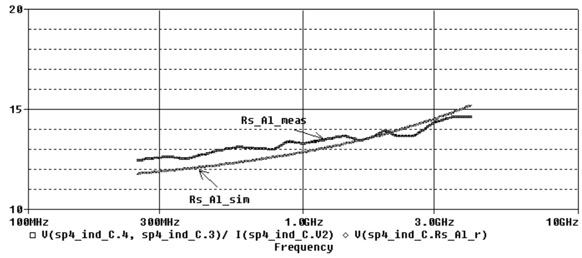

Fig. 3. The series resistance R_s vs frequency for *Al* spirals

The series inductance L_s vs frequency for *Al* spirals, the simulation results from the *PSpice* model compared to measured data from [1], are shown in Fig. 2. The series resistance R_s vs frequency for *Al* spirals, the simulation results from the *PSpice* model compared to measured data from [1], are presented in Fig. 3. .In order to calculate the Q-factor, an AC source V_1 is connected between the inductor ports 1 and 2 (Fig. 1). Q can be shown in *Probe* using the macroses:

Zin = v(1,2)/ I(V1)

Q = Img(Zin)/R(Zin)

The dependence for the Q-factor vs frequency for *Al* and *Cu* spirals using current sheet approximation for L_s [5], simulation results from the *PSpice* model compared to measured data from [1], are shown in Fig. 4 and Fig. 5 correspondingly.

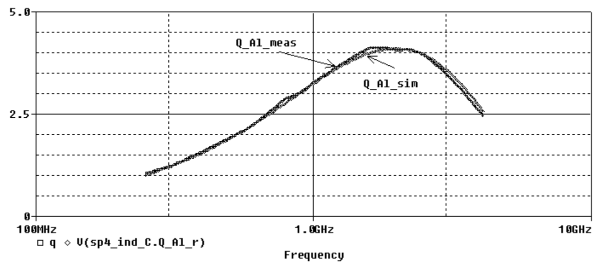

Fig. 4. Q-factor vs frequency for *Al* spirals using current sheet approximation for L_s [5]

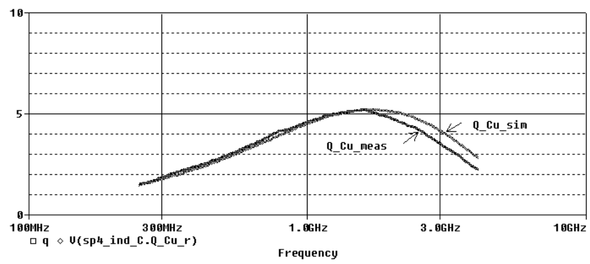

Fig. 5. Q-factor vs frequency for *Cu* spirals using current sheet approximation for L_s [5]

The dependence for the Q-factor vs frequency for *Al* spirals, simulation results from *PSpice* model compared to measured data from [1], are presented in Fig. 6 using modification of the Wheeler formula with K coefficients for L_s, as well as in Fig. 7 using the α – coefficients expression for L_s [5].

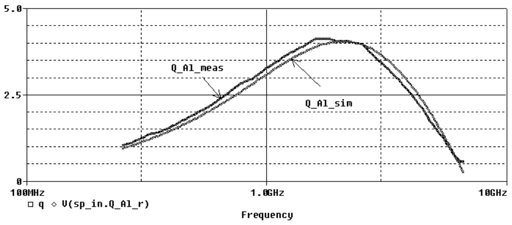

Fig. 6. *Q*-factor vs frequency for *Al* spirals using modification of the Wheeler formula with *K* coefficients for L_s [5]

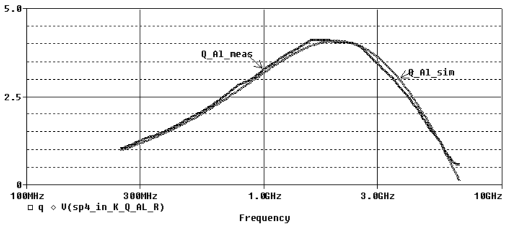

Fig. 7. *Q*-factor vs frequency for *Al* spirals using the α – coefficients expression for L_s [5]

The simulation results obtained by the *PSpice* macromodel are in excellent agreement with the results publishted in [1].

III. Computer PSpice Model of Planar Transformer

The modification of the Wheeler formula with *K* coefficients [5] is used for the calculation of the primary and secondary inductances L_p, and L_s of the spirals of the transformer. The expression for the series frequency-dependent resistance of the spirals is used to calculate the primary and the secondary series resistances of the transformer. The mutual inductance of the transformer is given using the formula:

$$M = k\sqrt{L_p L_s} \ . \quad (9)$$

D_s is the center-to-center spiral distance and is depending on the horizontal and the vertical spiral shifts X_s and Y_s. It can be calculated using the expression [6]:

$$D_s = \sqrt{X_s^2 + Y_s^2} \ . \quad (10)$$

The coupling coefficient *k* can be found using the expression:

$$k = 0.9 - D_s / D_{avg} \ , \quad (11)$$

$$C_{oxp} = \frac{1}{2}\frac{\varepsilon_{ox}}{t_{oxp}} lw \frac{A - A_{ov}}{A} \ ; \ C_{oxs} = \frac{1}{2}\frac{\varepsilon_{ox}}{t_{oxs}} lw \ , \quad (12)$$

$$C_{ov} = \frac{1}{2} lw \frac{\varepsilon_{ox}}{t_{oxp,oxs}} \frac{A_{OV}}{A} \ ; C_{oxm} = C_{oxp} + C_{oxs} \ , \quad (13)$$

$$A = D_{out}^2 \ ; \ A_{ov} = (D_{out} - X_s)(D_{out} - Y_s) \ . \quad (14)$$

Here *A* is the area of the spiral and A_{ov} is the overlapped area between the top (primary) and the bottom (secondary) spirals.

The *PSpice* computer macromodel of the spiral transformer is shown in Fig. 8. The input parameters for the transformer simulation are presented in Table II. The shapes of the top and bottom spirals are the same.

The two-port *S*-parameters can be shown in *Probe* using the macroses (Fig. 8):

S11 = 2*V(1) @1 - 1
S22 = 2*V(4) @2 - 1
S12 = 2*V(1)@2

TABLE 2. INPUT PARAMETERS FOR THE TRANSFORMER SIMULATION

Transformer parameter	Value
Outer diameter D_{out}	180×10^{-6} m
Number of turns *n*	11.75
Width of spiral trace *w*	3.2×10^{-6} m
Line spacing *sp*	2.084×10^{-6} m
Metal thickness primary spiral t_p	2.1×10^{-6} m
Metal thickness secondary spiral t_s	0.6×10^{-6} m
Thickness of the oxide layer between the primary spiral and the substrate tox_p	3.5×10^{-6} m
Thickness of the oxide layer between the secondary spiral and the substrate tox_s	2.2×10^{-6}m
Metal resistivity of the primary spiral at DC for Al ρ_{Alp}	3×10^{-8} Ω.m
Metal resistivity of the secondary spiral at DC for Al ρ_{Als}	5×10^{-8}Ω.m
Oxide thickness from top level metal (primary spiral) to bottom level metal (secondary spiral) t_{oxps}	0.8×10^{-6} m
Horizontal spiral shift for overlapped/diagonally shifted spirals X_s	0 / 50e^{-6} m
Vertical spiral shift for overlapped/diagonally shifted spirals Y_s	0 / 50e^{-6} m
Permittivity of the oxide ε_{ox}	3.451×10^{-11} F/m
Magnetic permeability of the free space μ	1.256×10^{-6} H/m

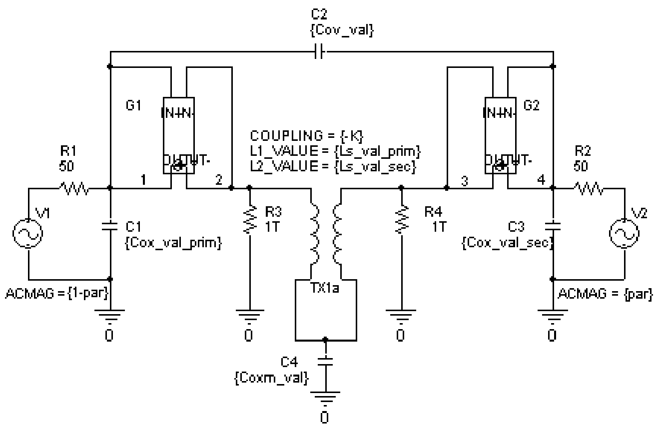

Fig. 8. Stacked transformer physical model [6]

The macroses describing the parameterized model of the planar transformer in *PSpice* are shown in Fig. 9. The real and imaginary parts of the *S*-parameters for transformer with overlapped spirals (X_s = 0, Y_s = 0) are simulated using the *PSpice* model using modification of the Wheeler formula with *K* coefficients for L_s. The simulation results compared to the measured data from [6] are shown in Fig. 10, Fig. 11 and Fig. 12 correspondingly. Similarly, the real and imaginary parts of the *S*-parameters for transformer with diagonally shifted spirals (X_s = 50μm, Y_s = 50μm), are obtained. The simulation results obtained by the *PSpice* macromodel are in excellent agreement with the results publishted in [6].

t_prim=2.1u t_sec=0.6u Din_prim={Dout_prim - 2.0*(n_prim* (sp_prim + w_prim) - sp_prim)} Din_sec={Dout_sec - 2.0*(n_sec* (sp_sec + w_sec) - sp_sec)} Dout_prim=180u Dout_sec=180u Davg_prim={0.5*(Din_prim + Dout_prim)} Davg_sec={0.5*(Din_sec + Dout_sec)} L_prim={4*Davg_prim*n_prim} L_sec={4*Davg_sec*n_sec} A={Dout_prim^2} Aov={(Dout_prim - Xs)*(Dout_prim - Ys)} sp_prim=2.084u sp_sec=2.084u n_prim=11.75 n_sec=11.75	w_prim=3.2u w_sec=3.2u tox_prim=3.5u tox_sec=2.2u tox_prim_sec=0.8u K={0.9 - Ds/Davg_prim} Ds={sqrt(Xs^2 + Ys^2)} Ls_val_prim={(K1*mju*((n_prim^2)*Davg_prim))/(1.0 + K2*((Dout_prim - Din_prim)/(Dout_prim +Din_prim)))} Ls_val_sec={(K1*mju*((n_sec^2)*Davg_sec))/(1.0 + K2*((Dout_sec - Din_sec)/(Dout_sec + Din_sec)))} Cox_val_prim={((0.5*L_prim*w_prim*Eox)/tox_prim)* ((A - Aov)/A)} Cox_val_sec={(0.5*L_sec*w_sec*Eox)/tox_sec} Cov_val={0.5*(Eox/tox_prim_sec)*L_prim*w_prim* (Aov/A)} Coxm_val={Cox_val_prim + Cox_val_sec}	Xs=0 Ys=0 mju=1.256e-6 beta=1.62e-3 c1=1.27 c2=2.07 c3=0.18 c4=0.13 al1=-1.21 al2=-0.147 al3=2.4 al4=1.78 al5=-0.03 K1=2.34 K2=2.75 Eox=3.451e-11 sigma_sec=20e6 sigma_prim= 33.33e6

Fig. 9. Macroses describing the parametrized model of the planar transformer

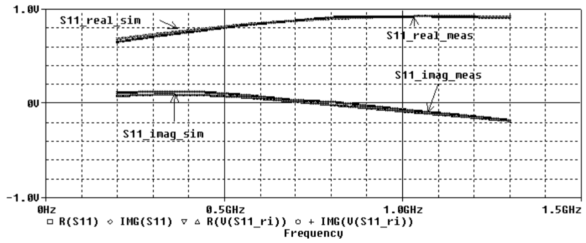

Fig. 10. Real and imaginary parts of S_{11} for transformer with overlapped spirals

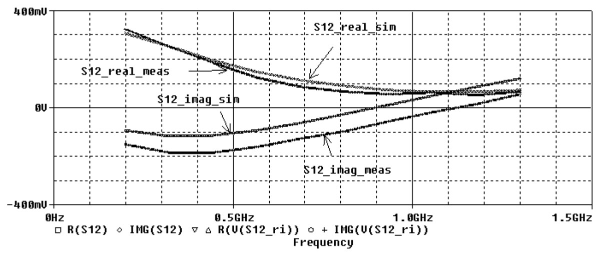

Fig. 11. Real and imaginary parts of S_{12} for transformer with overlapped spirals

IV. CONCLUSION

Parameterized computer macromodels have been developed for accurate simulation of spiral inductors and planar transformers using the standard circuit simulator *Cadence PSpice*. The macrodefinitions describing the models are presented. A verification of the developed macromodels is performed by comparison between the simulation results obtained using *PSpice* and the measurement data given in [1,6]. The developed macromodels can be used directly in the RF circuit design process.

V. ACKNOWLEDGEMENT

The investigations are supported by the project 091ni041-03/2009 with the R&D sector of the Technical University of Sofia.

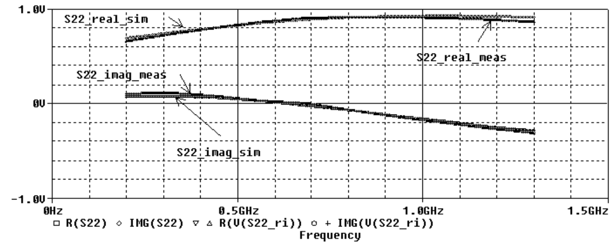

Fig. 12. Real and imaginary parts of S_{22} for transformer with overlapped spirals

REFERENCES

[1] C. P. Yue, C. Ryu, J. Lau, T. H. Lee, S. S. Wong. *A Physical model for planar spiral inductors on silicon*, Proc. IEEE Int. Electron Devices Meeting Tech. Dig. San Francisco, CA, Dec. 1996, pp. 155-158.

[2] C. P. Yue, S. S. Wong. *Design Strategy of On-Chip Inductors for Highly Integrated RF Systems*, DAC 99, New Orleans, Louisiana, USA, 1999, ACM 1-58113-092-9.

[3] N. Talwalkar, C. P. Yue, S. S. Wong. *Compact Modeling of High Frequency Phenomena for On-Chip Spiral Inductors*, Nanotech 2003, Vol. 2, www.nsti.org, ISBN 0-9728422-1-7.

[4] C. C. Yue, S. S. Wong. *On-Chip Spiral Inductors with Patterned Ground Shields for Si-Based RF IC's*, IEEE Journal of Solid-State Circuits, Vol. 33, No. 5, May 1998, pp. 743 – 752.

[5] S. Mohan, M. Hershenson, S. Boyd, T. Lee. *Simple Accurate Expressions for Planar Spiral Inductances*, IEEE Journal of Solid-State Circuits, v. 34, No. 10, Oct. 1999, pp.1419-1424.

[6] S. Mohan, C. Yue, M. Hershenson, S. Wong, T. H. Lee. *Modeling and Characterization of On-Chip Transformers*, Center for Integrated Systems, Stanford University, July 1998.

[7] M. Hershenson, S. S. Mohan, S. P. Boyd, T. H. Lee. *Optimization of inductor circuits via geometric programming,* Dept. of Electr. Eng., Stanford Univ., CA, USA, Proceedings 36th Design Automation Conference, 1999.

[8] E. Gadjeva, V. Durev, M. Hristov, D. Pukneva. *Optimization of Geometric Parameters of Spiral Inductors using Genetic Algorithms*, Proceedings of the International Conference Mixed Design of Integrated Circuits and System, MIXDES 2006, 22-24 June 2006, pp. 518-521.

[9] PSpice User's Guide, Cadence PCB Systems Division, USA, 2000.