Sensitivity and Tolerance Analysis in Analog Circuits using Symbolic Methods

Lucian MANDACHE(1), Mihai IORDACHE(2), Lucia DUMITRIU(2)
(1)*University of Craiova, Department of Electrical Engineering, Decebal Blv. 107, Craiova, Romania*
(2)*"Politehnica" University of Bucharest, Department of Electrical Engineering, Splaiul Independenței 313, Bucharest, Romania*
lmandache@elth.ucv.ro, iordache@elth.pub.ro, dlucia@elth.pub.ro

***Abstract* — The paper is focused on a new and practical approach to perform sensitivity and tolerance analysis of analog lumped circuits. Any linear circuit can contain passive elements, magnetically coupled inductors, excess elements, and any type of independent and controlled sources. Special strategies based on symbolic methods are used in order to reduce the computational effort and to minimize the numerical errors in the automatic design of these circuits. As part of this process, a new, modern, reliable and easy-to-use software tool for sensitivity and tolerance analysis has been developed, as a useful and valuable support for research and design engineers.**

***Index Terms* — sensitivity, tolerance analysis, analog circuit, symbolic methods, software tools.**

I. INTRODUCTION

The optimal design of electric and electronic systems must guarantee all required operating and reliability parameters with minimal manufacturing costs. After choosing the appropriate topology and performing optimization studies, this goal is strongly closed to the market price of the components. All circuit components are manufactured around a target value of their main parameter (the rated value) with an allowable tolerance. Smaller tolerance requires higher technological accuracy, which involves the increasing of the component costs and vice versa.

Because the design engineer must choose as many cheap components as possible, keeping the circuit performance, he must decide which components are critical and how much is the required value of the tolerance. Such a decision is possible only through a rigorous sensitivity analysis followed by tolerance analyses. The problem related to sensitivities and tolerances became necessary in connection to the development of electronic circuits and their serial production [1].

So, only for the critical components one uses high quality and expensive products, the noncritical components being cheaper. In this manner, the cost minimization is achieved and the unwanted behavior of the circuit will be avoided [2].

The sensitivity and tolerance accurate analysis is generally a difficult task from the point of view of the computational effort in connection to the circuit topology and operating mode, numerical errors and difficult interpretation of the results.

Although any electric circuit must be subject of sensitivity and tolerance analysis, the area is restricted here around the linear lumped circuits as well as the passive and active analog filters.

Let $H(s, p_1, ..., p_m)$ be a network function of interest, with the complex frequency $s = j\omega$ and the vector of the independent parameters $\boldsymbol{p} = [p_1, p_2, ..., p_k, ..., p_m]^{\mathrm{t}}$ of the circuit components. The parameters can be resistances, inductances, capacitances, transfer parameters of the controlled sources, or some physical quantities that affect the element values (such as temperature). The sensitivity of a network function with respect to one parameter shows the influence of small deviations of this parameter on the network function, $\partial H / \partial p_k$ if all other parameters remain unchanged [3,4]. Higher values of the sensitivities designate the critical components.

Many methods to compute the circuit sensitivities were developed. A brute-force method is to vary the parameter of interest slightly, $\Delta p_k \to 0$, calculate the change in *H*, then the ratio $\Delta H / \Delta p_k$. The accuracy of such a method can be poor because of the round-off numerical errors given by the differences between two nearly equal numbers [3]. Thus, the computational effort is high because of the repeated analyses and multiple sensitivity evaluations at each frequency. Other methods have been promoted by many authors, e.g. the incremental-network approach, the adjoint-network approach, as well as symbolic-network-function approach [3-8]. It is not our goal to comment the advantages and disadvantages of these approaches, but it is generally recognized that the most powerful methods are based on symbolic algorithms [8,9].

The most advanced topics are referred to high-order sensitivities, pole-zero sensitivities, transient sensitivities, summed sensitivity invariants [3,10], but these approaches are less useful to achieve a common design of electronic systems.

Turning back to the first-order sensitivity analysis, the symbolic methods are not yet sufficiently exploited, this being the main reason of our research.

As was mentioned above, the sensitivity analysis is followed by tolerance analyses. Because almost all parameters of the circuit elements deviate simultaneously from their nominal values, a tolerance analysis is necessary in order to find the deviation range of the network function of interest or circuit response.

From de point of view of the design of electronic systems, the tolerance problem consists in finding the possible tolerances of the circuit parameters which guarantee an acceptable distribution of the circuit performance.

Many techniques of tolerance analysis were developed during the last decades. A good technique must be able to find the worst case given by the specified tolerances of the circuit elements [8,11-14].

A tolerance analysis is commonly based on stochastic models because of the random distribution of the parameters within their tolerance domain. The well known approach is Monte Carlo analysis, that requires repeated circuit simulations in which the parameter value samples are chosen with a normal (Gaussian) or uniform distribution [8,12]. A higher number of samples gives more accurate results, but involves increasing of the computational effort. Nevertheless, it has been adopted by many commercial analysis programs, like SPICE.

Other techniques deal with mathematics interval methods [12], root-sum-square or extreme-value analysis [13,14], but the efficiency of each method depends on the circuit complexity or on the operation mode.

We developed new approaches of Monte Carlo and extreme-value analysis, exploiting symbolic and partial symbolic computation methods in order to gain efficiency.

The section II of the paper describes the principles of the developed algorithms, the section III shows their implementation in a new CAD tool, and the section IV presents an example to show the capabilities of the new software, in comparison with a common SPICE simulation.

II. ANALYSIS ALGORITHMS

Let us consider a circuit of *n* nodes and *l* branches (where l_p are passive RLCM branches, l_E - independent voltage sources, l_J - independent current sources, l_{Ec} - controlled voltage sources and l_{Jc} - controlled current sources).

In order to exploit as much as possible the symbolic calculus, we start with the generation of the network function of interest (or circuit response) in the Laplace domain and in symbolic form. The network function can be any input or transfer impedance, admittance, voltage or current gain generally defined by:

$$H(s) = \frac{X_{out}(s)}{X_{in}(s)} \tag{1}$$

where the input signal is the quantity associated to an independent source and the output is any branch current, node voltage or branch voltage. According to our goal, the initial conditions are assumed to be zero and only one independent source is nonzero. Since the network function (1) does not depend on the circuit operation mode, one can consider the input as a $\delta(t)$ impulse, with the simplest Laplace transform $\mathcal{L}\{\delta(t)\} = 1$, so that $X_{in}(s) = 1$.

A mathematical model of the circuit is built using the modified nodal approach in the form detailed in [15,16]:

$$\boldsymbol{M}(s) \cdot \boldsymbol{X}(s) = \boldsymbol{N}(s) \tag{2}$$

where:

a) the vector of independent variables:

$$\boldsymbol{X}(s) = \left[\boldsymbol{V}^{\mathrm{t}}(s) \;\vdots\; \boldsymbol{I}_E^{\mathrm{t}}(s) \;\vdots\; \boldsymbol{I}_{Ec}^{\mathrm{t}}(s)\right]^{\mathrm{t}} \tag{3}$$

contains the node voltages and the zero-impedance branch currents of independent and controlled voltage sources; we remark that the controlling branches of current-controlled sources are modeled by independent zero-voltage sources, in order to maintain the generality of the algorithm [9,16,17];

b) the matrix

$$\boldsymbol{M}(s) = \left[\begin{array}{c:c:c} \boldsymbol{A}_p \boldsymbol{Y}(s) \boldsymbol{A}_p^{\mathrm{t}} + \boldsymbol{A}_{Jc} \boldsymbol{G}_c \boldsymbol{A}_J^{\mathrm{t}} & \boldsymbol{A}_E + \boldsymbol{A}_{Jc} \boldsymbol{B}_c & \boldsymbol{A}_{Ec} \\ \hdashline \boldsymbol{A}_E^{\mathrm{t}} & \boldsymbol{0}_{l_E \times l_E} & \boldsymbol{0}_{l_E \times l_{Ec}} \\ \hdashline \boldsymbol{A}_{Ec}^{\mathrm{t}} + \boldsymbol{A}_c \boldsymbol{A}_J^{\mathrm{t}} & \boldsymbol{R}_c & \boldsymbol{0}_{l_{Ec} \times l_{Ec}} \end{array}\right] \tag{4}$$

contains the branch admittance matrix $\boldsymbol{Y}(s)$, the transfer conductance matrix of the voltage-controlled current sources $\boldsymbol{G}_c$, the matrix of the current gains of the current-controlled current sources $\boldsymbol{B}_c$, the matrix of the voltage gains of the voltage-controlled voltage sources $\boldsymbol{A}_c$, transfer resistance matrix of the current-controlled voltage sources $\boldsymbol{R}_c$, and some partitions of the nodes-branches incidence matrix: $\boldsymbol{A}_p$, $\boldsymbol{A}_E$, $\boldsymbol{A}_J$, $\boldsymbol{A}_{Ec}$, $\boldsymbol{A}_{Jc}$; in the branch admittance matrix the mutual admittances, if exist, are placed on non-diagonal positions.

c) the matrix

$$\boldsymbol{N}(s) = -\begin{bmatrix} \boldsymbol{A}_J \boldsymbol{J}(s) \\ \boldsymbol{E}(s) \\ \boldsymbol{0}_{l_{Ec} \times 1} \end{bmatrix} \tag{5}$$

contains the independent source parameters.

Solving the algebraic system (2), the vector of the circuit response $\boldsymbol{X}(s)$ is obtained. One of its components is usually the quantity of interest. If not (as in the case of a passive branch current), it can be computed easily in terms of the above solution [17]. The quantity of interest is extracted and expressed as a rational function in *s*, representing the searched network function:

$$X_{out}(s) = \frac{P(s)}{Q(s)} = H(s) \tag{6}$$

Regarding the sensitivity analysis, a normalized value of the sensitivity is more convenient because it allows comparing the sensitivities with respect to many circuit parameters [3,8]:

$$S_{p_k}^H(s) = \frac{\partial H(s)}{\partial p_k} \cdot \frac{p_k}{H(s)} \tag{7}$$

Or

$$S_{p_k}^H(s) = \frac{\partial \ln H(s)}{\partial \ln p_k} \tag{8}$$

For sine wave input signals, the complex transfer function can be expressed as:

$$H(j\omega) = |H(j\omega)| \cdot e^{j \cdot \arg|H(j\omega)|} \tag{9}$$

from where

$$\ln H(j\omega) = \ln|H(j\omega)| + j \cdot \arg|H(j\omega)| = A(\omega) + j\varphi(\omega) \tag{10}$$

In (10) the magnitude function and the phase were used as:

$$A(\omega) = \ln|H(j\omega)|, \quad \varphi(\omega) = \arg|H(j\omega)| \tag{11}$$

By replacing the expression (10) in (8), the normalized sensitivity in the frequency domain becomes:

$$S_{p_k}^H(j\omega) = \frac{\partial \ln H(j\omega)}{\partial \ln p_k} = p_k \cdot \frac{\partial A(\omega)}{\partial p_k} + j\, p_k \cdot \frac{\partial \varphi(\omega)}{\partial p_k} \tag{12}$$

where the normalized (or relative) sensitivities of the magnitude and the phase appear:

$$S_{p_k}^{A}(\omega) = \mathrm{Re}\left(S_{p_k}^{H}(j\omega)\right) = p_k \cdot \frac{\partial A(\omega)}{\partial p_k} \tag{13}$$

$$S_{p_k}^{\varphi}(\omega) = \mathrm{Im}\left(S_{p_k}^{H}(j\omega)\right) = p_k \cdot \frac{\partial \varphi(\omega)}{\partial p_k} \tag{14}$$

These last definitions are the most useful in practice. Since the network function was found, the expressions (13)-(14) allow the sensitivity computation at any value of circuit parameters and any frequency.

In our approach two strategies for tolerance analysis were considered: a Monte Carlo approach with normal distribution of the samples, respectively an extreme-value strategy.

Since the Monte Carlo analysis is statistic-based method, many hundreds, or rather thousands of samples are required for satisfactory results. Assuming N be the number of samples, each sample being defined by a randomly established combination of the circuit parameters, N AC analyses must be performed. The obtained frequency responses define the expected area of the actual response. Theoretically, the worst-case can not be found in this manner, except for N extremely large $N \to \infty$.

If the symbolic form of the network function is known, and if n intermediary values of the frequency are considered, the tolerance analysis requires $n \cdot N$ successive evaluations of the network function and the same amount of values must be written in the computer memory.

The extreme-value strategy deals only with the minimum and maximum values of each circuit parameter. Therefore, if m parameters are considered, 2^m samples and 2^m AC analyses are necessary. In this manner, the worst possible upper and lower limits of the circuit response are obtained. The extreme-value analysis requires $n \cdot 2^m$ evaluations of the network function, but at each frequency only the maximum and minimum values have top be kept in the computer memory ($2n$ values). Hence less computer memory than for the statistic-based methods is required. Better results comparing to the statistical methods are obtained, but the computation effort can be unacceptable high for large-scale circuits.

III. CAD TOOL

Using the high performance computing environment MATLAB, the algorithms described in section II have been recently implemented in a comprehensive analysis program (named PATCA) conceived as a useful tool for computer aided design, as well as for research purposes [8]. It combines several capabilities, accomplishing the following performance criteria: usability, reliability, precision (error minimization), constructive flexibility, hardware and software resources requirement, compatibility with other analysis programs. An interactive graphical user interface facilitates handling the program; it contains push buttons, popup menus and editable text boxes that allow performing any command action and setting analysis parameters (fig. 1).

The input data is a SPICE-compatible netlist (.cir file), which can be created either by text editing or rather through the circuit diagram built using the Schematic editor of SPICE. This last solution allows performing preliminary AC analyses using SPICE, as well as witness (but of poorer quality) tolerance analyses.

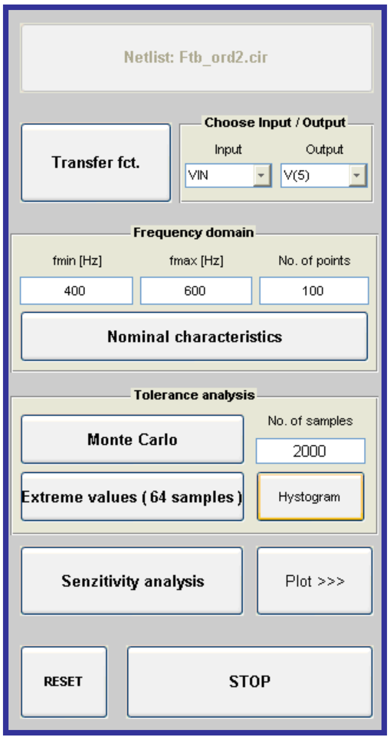

Fig. 1. Graphical user interface of PATCA.

Next to importing the netlist, the user can choose the desired input and output quantities using popup menus. Then, by pushing a button, he launches the computation of the network function that is computed in symbolic form and evaluated for the nominal values of the parameters. Its poles and zeros are computed and represented in the complex plan, as a qualitative image of the circuit behavior.

The next step requires setting the frequency domain and the number of intermediary frequency points (using the corresponding text boxes of the GUI), if they differ from those given in the netlist. Then the nominal frequency characteristics (magnitude and phase respectively) are computed and plotted.

The tolerance analysis is available either by a Monte Carlo approach with normal distribution of the samples (the number of samples is established by the user) or by an extreme-value strategy. The results are plotted near the nominal characteristics, showing their possible deviation range.

The sensitivity analysis results are plotted both for magnitudes and phases in normalized manner.

IV. EXAMPLE

Let us study the second-order band-pass filter in Sallen-Key topology, shown in Figure 2. Its nominal cutoff frequency is 500 Hz. As it is known, such a structure is very sensitive at the parameter deviations. The circuit diagram was built using SPICE and the corresponding netlist file was generated automatically:

```
*SPICE_NET
.AC LIN 100 400HZ 600HZ
*ALIAS  V(5)=UOUT
.PRINT AC  V(5)  VP(5)
R2 1 0 80.6 TOL=1%
```

```
VIN 2 0 AC 1
C1 1 3 0.1E-6 TOL=2%
C2 5 1 0.1E-6 TOL=2%
R3 3 5 127000 TOL=1%
E1 5 0 0 3 10000 TOL=20%
I1 0 3
R1 2 1 6340 TOL=10%
.END
```

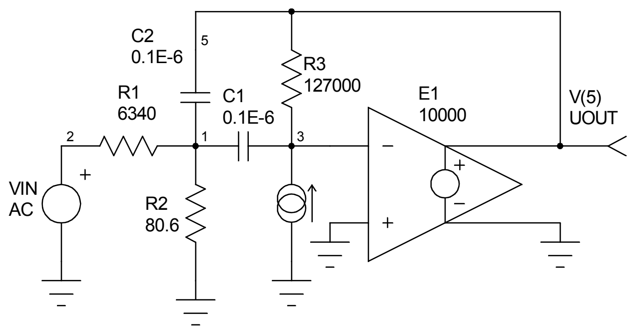

Fig. 2. Sallen-Key filter.

For purposes of systematically formulation of the circuit mathematical model based on modified nodal equations, the controlling branch of the voltage-controlled voltage source was modeled by a zero-independent current source.

In order to compute the network function

$$H(s) = \frac{V_5(s)}{V_2(s)}$$

the modified nodal equations in the Laplace domain are built and shown in the MATLAB main window:

(1/R2+s*C1+s*C2+1/R1)*V1-1/R1*V2-s*C1*V3-s*C2*V5 = 0

-1/R1*V1+1/R1*V2+I2 = 0

-s*C1*V1+(s*C1+1/R3)*V3-1/R3*V5 = 0

-s*C2*V1-1/R3*V3+(s*C2+1/R3)*V5+I6 = 0

V2 = 1

A6_7*V3+V5 = 0

The network function was extracted from the main window as well:

H(s) = V5(s)/VIN(s) = Numerator / Denominator

Numerator = -A6_7*R2*s*C1*R3

Denominator = (R3*C1*R2*R1*C2*A6_7+C1*R2*R1*C2*R3)*s^2+
+(A6_7*C2*R2*R1+A6_7*C1*R2*R1+R1*C1*R3+C2*R2*R1+
+C1*R3*R2+C1*R2*R1)*s+A6_7*R1+R2+R1+A6_7*R2

It is evaluated for the nominal values of the parameters and their poles and zeros are computed and plotted (Fig. 3):

```
                                              s
V5(s) / VIN(s) = -0.00015941*  ----------------------------------------------
                               1.011e-007 s^2 + 1.719e-005 s + 1
```

Zeros = 0

Poles = -8.5022e+001 +3.1442e+003i,
-8.5022e+001 -3.1442e+003i

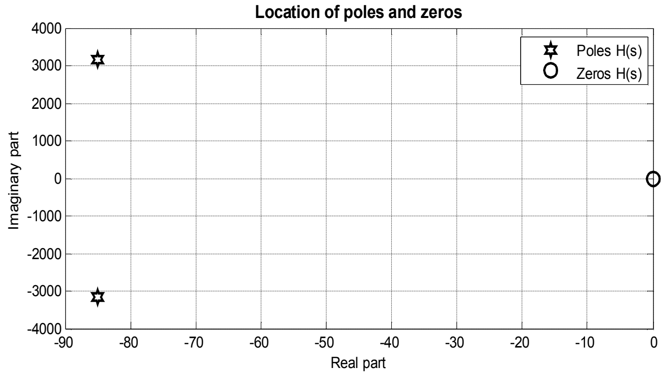

Fig. 3. Poles and zeros of the network function.

The tolerance analysis was firstly performed by the extreme-value method, where $2^6 = 64$ repeated analyses were necessary. The results are shown in Fig. 4.

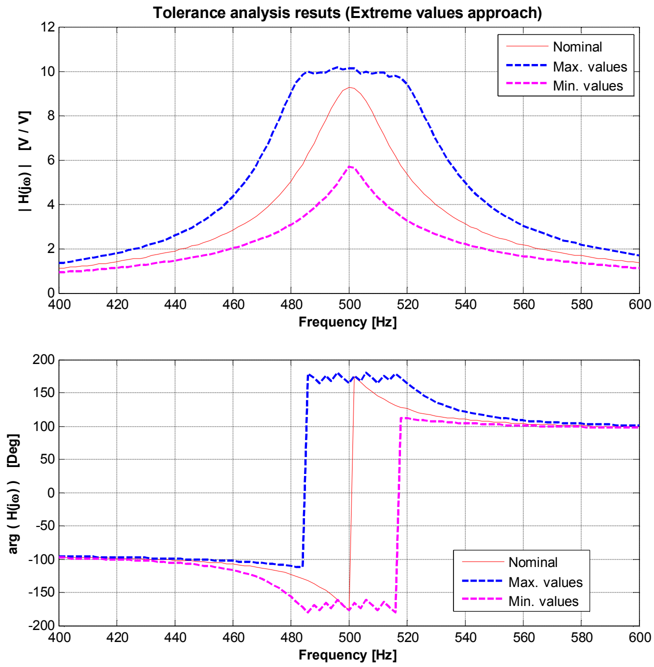

Fig. 4. Results of the tolerance analysis by the extreme-value analysis (EVA).

The Monte Carlo analysis was performed for 2000 samples. Only the results of magnitude are shown here, as well as the corresponding histogram of the maximum value distribution (Fig. 5).

It is interesting to compare these two tolerance analyses: although the last method requires a computation effort of more than 30 times greater, the obtained deviation range is narrower. Also, the worst case has not been covered, the classical Monte Carlo method being less reliable. Although, we can remark that it offers a useful image of the statistical distribution of the cutoff frequencies. The first (EVA) method requires 5 seconds as computation time, while the last – about 22 times longer.

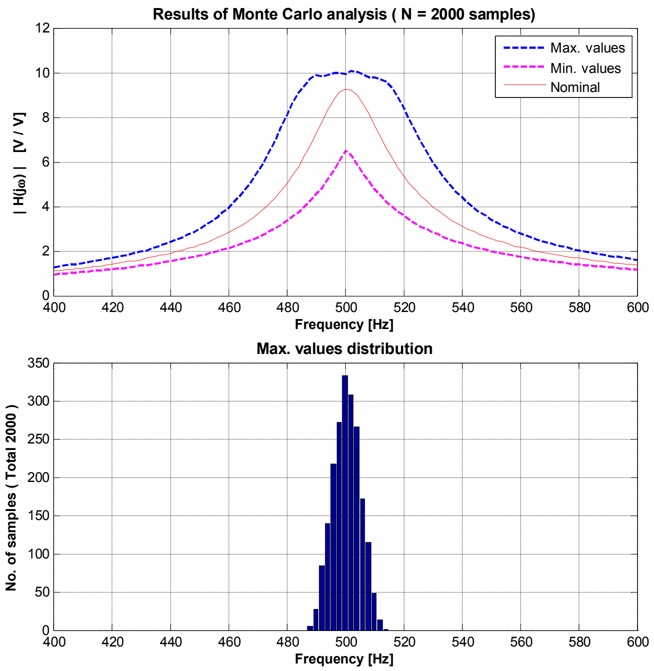

Fig. 5. Results of the tolerance analysis by the classical Monte Carlo analysis (MCA) (2000 samples).

For comparison, we performed a SPICE Monte Carlo analysis of the same circuit. The computing time required was more than 3 minutes for only 100 samples, and the plotting was inadequate (Fig. 6). Only the magnitude was plotted and the obtained deviation range was extremely narrow, proving a weaker performance.

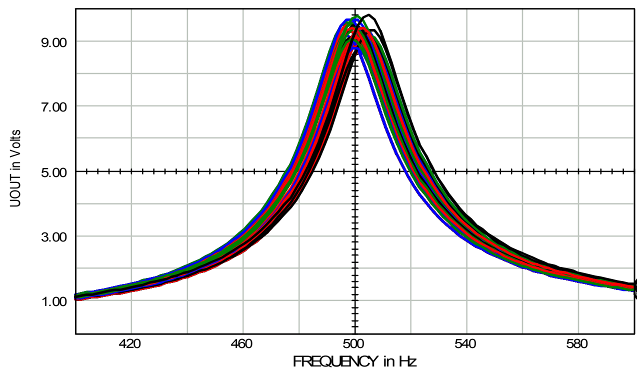

Fig. 6. Results of the tolerance analysis by the classical Monte Carlo analysis performed with SPICE (100 samples).

The sensitivity analysis gave the results shown in Fig. 7. We present the symbolic form of the network function sensitivity only with respect to the parameter R1, as it was computed by the program:

Sensitivity of the network function with respect to R1:

```
-1/(A6_7*R1+R2+A6_7*R2+R1*s*C1*R3+
+R3*s^2*C1*R2*R1*C2*A6_7+A6_7*s*C1*R2*R1+
+A6_7*s*C2*R2*R1+R1+s^2*C1*R2*R1*C2*R3+s*C1*R3*R2+
+s*C1*R2*R1+s*C2*R2*R1)*(A6_7+s*C1*R3+
+R3*s^2*C1*R2*C2*A6_7+A6_7*s*C1*R2+A6_7*s*C2*R2+1+
+s^2*C1*R2*C2*R3+s*C1*R2+s*C2*R2)*R1
```

As one can see in Fig. 7, the circuit response is less responsive at the deviation of R1 in vicinity of the cutoff frequency, as compared to R2, R3, C1 and C2 (around 12 times in terms of magnitude and 100 times in terms of phase). A similar comment is suitable for the voltage gain of the controlled source. These are reasons to choose the resistor R1 from a lower class of tolerance (see the netlist content above). The chosen circuit elements correspond to a possible deviation of the cutoff frequency of 3% (in the worst case), as one can see in Fig. 4.

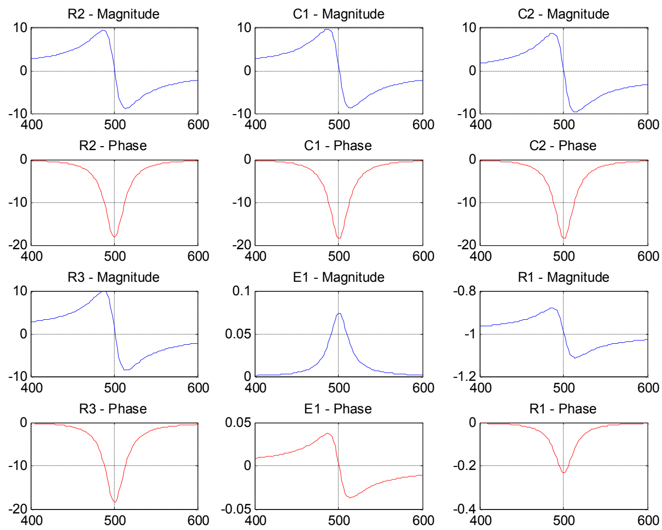

Fig. 7. Normalized sensitivities of the magnitude and the phase with respect to deviation parameters.

V. CONCLUSION

Efficient algorithms for sensitivity and tolerance analysis were developed, based on symbolic and partial symbolic computation methods. They are focused on the linear circuits, like passive and active analog filters, which are very sensitive at the parameter deviations.

A comprehensive sensitivity and tolerance analysis program for linear analog circuits has been implemented, as a necessary and useful tool for computer aided design and research purposes. Any circuit topology can be treated, including magnetically coupled inductors, controlled sources and excess elements.

A complete example is given, proving the program resources and performances.

ACKNOWLEDGMENTS

This work was supported by the Romanian Ministry of Education under Grant PCE 539/2008.

REFERENCES

[1] H.W. Bode, *Network Analysis and Feedback Amplifier Design*, Van Nostrand Reinhold Company, New York, 1945.

[2] Kun-Lin Hsieh, *The study of cost-tolerance model by incorporating process capability index into product lifecycle cost*, The International Journal of Advanced Manufacturing Technology, Springer London, ISSN 0268-3768, Volume 28, Numbers 5-6 / March, 2006, pag. 638-642.

[3] L. O. Chua, P. M. Lin, *Computer-Aided Analysis of Electronic Circuits – Algorithms and Computational Techniques*, Preutice-Hall, In., Englewood Cliffs, New Jersey, 1975.

[4] P.M. Lin, *Sensitivity analysis of large linear networks using symbolic programs*, Proceedings of The IEEE International Symposium on Circuits and Systems ISCAS '92, 10-13 May 1992, vol.3, pp.1145-1148.

[5] L. Vallese, *Incremental versus adjoint models for network sensitivity analysis*, IEEE Transactions on Circuits and Systems, vol.21, no.1, pp. 46- 49, Jan 1974.

[6] Herng-Jer Lee, Chia-Chi Chu, Wu-Shiung Feng, *Interconnect modeling and sensitivity analysis using adjoint networks reduction technique*, Proceedings of the 2003 International Symposium on Circuits and Systems, ISCAS '03, vol. 4, pp. IV-648 - IV-651, 25-28 May 2003.

[7] F. Yuan, A. Opal, *Sensitivity analysis of periodically switched linear circuits using an adjoint network technique*, Proceedings of the 1999 IEEE International Symposium on Circuits and Systems, ISCAS '99, vol. 5, pp. 331-334.

[8] M. Iordache, L. Mandache, D. Niculae, *Analiza asistată de calculator a senzitivităţilor şi toleranţelor circuitelor analogice*, Ed. Matrix Rom, Bucureşti, 2010.

[9] M. Iordache, Lucia Dumitriu, *Simularea asistată de calculator a circuitelor analogice*, Ed. Politehnica press, Bucureşti, 2002.

[10] F. Vallette, G. Vasilescu, G. Alquié, *A Fully Recursive Approach to the Computation of Higher Order Sensitivities of Linear Active Circuits*, IEEE Transactions on circuits and systems - I: Fundamental theory and applications, Vol. 46, No. 8, August 1999, pp. 907-919.

[11] Lubomir Kolev, *Worst-Case Tolerance Analysis of Linear DC and AC Electric Circuits*, IEEE Transactions on Circuits and Systems - I: Fundamental theory and applications, Vol. 49, No. 12, December 2002, pp. 1693-1701.

[12] K. Singhal, J. F. Pinel, *Statistical Design Centering and Tolerancing Using Parametric Sampling*, IEEE Trans. on Circuits and Systems, Vol. CAS-28, No. 7, July 1981, pag. 692-702.

[13] Wei Tian, Xie-Ting Ling, Ruey-Wen Liu, *Novel Methods for Circuit Worst-Case Tolerance Analysis*, IEEE Trans. on Circuits and Systems- I: Fundamental Theory and Applications, Vol. 43, No. 4, April 1996, pag. 272-278.

[14] N. Evmorfopoulos, G. Stamoulis, J. Avaritsiotis, *A Monte Carlo Approach for Maximum Power Estimation Based on Extreme Value Theory*, IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems, Vol. 21, No. 4, April 2002, pp. 415-432.

[15] M. Iordache, L. Mandache, *Analiza asistată de calculator a circuitelor analogice neliniare*, Ed. Politehnica Press, Bucureşti, 2004.

[16] L. Mandache, D. Topan, *Simularea circuitelor electrice. Algoritmi şi programe de calcul*, Ed. Universitaria, Craiova, 2009.

[17] M. Iordache, Lucia Dumitriu, *Teoria modernă a circuitelor electrice - Vol. II - Fundamentare teoretică, Aplicaţii, Algoritmi şi Programe de calcul*, - Editura All Educational S.A., Bucureşti 2000.

[18] Y. Elcherif, P.M. Lin, *Transient analysis and sensitivity computation in piecewise-linear circuits*, IEEE Transactions on Circuits and Systems, vol.38, no.12, Dec 1991, pp.1525-1533.

[19] H. Kettani, B. R. Barmish, *A New Monte Carlo Circuit Simulation Paradigm with Specific Results for Resistive Networks*, IEEE Transactions on Circuits and Systems - I: Regular papers, Vol. 53, No. 6, June 2006, pp. 1289-1299.