A New Spice-Oriented Frequency-Domain Optimization Technique

Masayoshi ODA †, *Yoshihiro YAMAGAMI* †, *Yoshifumi NISHIO* †, *Junji KAWATA* ‡ *and Akio USHIDA* ‡
†Dept. of Electrical and Electronic Engineering, Tokushima University, 770-8506, Japan
E-mail:{masayoshi,yamagami,nishio}@ee.tokushima-u.ac.jp
‡Dept. of Mechanical and Electrical Electronic Engineering, Tokushima Bunri University, 769-2193, Japan
E-mail:{kawata,ushida}@fe.bunri-u.ac.jp

***Abstract*— There are many kinds of optimization techniques for designing high-performance RF circuits. In this paper, we propose a new frequency-domain Spice-oriented optimization algorithm using the steepest descent method. We have developed a simulator executing the frequency-domain analysis based on the harmonic balance (HB) method, where all the nonlinear devices such bipolar transistors and MOSFETs are replaced by the equivalent HB modules. The objective functions in the optimization are estimated by the DC analysis of the modified HB circuits.**

On the other hand, our steepest descent algorithm is realized by equivalent circuit model. Thus, the optimum solution is stably found by the transient analysis of Spice. We show the formulation of HB circuits in section II, the Spice-oriented optimization algorithm in section III and the interesting examples in section IV.

I. Introduction

There have been proposed many kinds of optimization algorithms for designing analog integrated circuits [1-5]. Especially, the frequency-domain optimization method is very important for designing the high performance RF circuits and the communication systems. Our algorithm is combined with frequency-domain harmonic balance (HB) method for solving nonlinear circuit and the optimization technique based on the steepest descent method.

The Volterra series methods [6-8] are widely used for the analysis of nonlinear circuits in the frequency domain, which can be also easily applied to design the circuits [3,5-6], because they can get the solutions in the analytical forms. Although the algorithms are based on the elegant bilinear theory, it is not so easy to derive the higher order Volterra kernels. Furthermore, the applications are difficult for large scale systems containing many nonlinear elements [6-7]. Remark that the methods can be only applied to the weakly nonlinear circuits such that the nonlinear devices are modeled by power series. The convergences to the strong nonlinearities are not guaranteed.

The HB method is also widely used for solving nonlinear circuits [10-13] in the frequency domain. It has such a property that it can be stably applied even to the strong nonlinear circuits having bipolar transistors and/or MOSFETs. On the other hand, they need many troublesome tasks for getting the circuit equations and the applications of HB method.

Therefore, we have developed a Spice-oriented algorithm based on the HB method using Analog Behavior Models (ABMs), where all the nonlinear devices are replaced by corresponding HB models, and linear inductors and capacitors are transformed into the corresponding HB elements coupled with the controlled sources [11]. Once these device modules such as bipolar transistors and MOSFETs are stored in our computer library, a given circuit is easily transformed into the HB circuit. Thus, the frequency-domain analysis such as frequency response curves can be easily obtained by solving the circuit with DC analysis of Spice. We show our HB method and the device modules of nonlinear devices in section II. These solutions from HB circuit are used in the following frequency-domain optimization techniques.

Now, let us discuss our optimization algorithm in the frequency-domain. First, we define the objective function as follow,

$$\Phi(\boldsymbol{x},\boldsymbol{y}), \quad \boldsymbol{x} \in \boldsymbol{R}^n, \quad \boldsymbol{y} \in \boldsymbol{R}^m, \tag{1}$$

where
$\boldsymbol{x}$:circuit variables such as voltages and currents,
$\boldsymbol{y}$:parameters (ex., L, R, C, voltage and current sources).

We want to minimize or maximize the function (1) by adjusting $\boldsymbol{y}$, where $\boldsymbol{x}$ are obtained from the HB circuit [1] . Although there are many optimization techniques [1-5], we use a well-known steepest descent algorithm using Spice. Although the descent direction is generally decided by the sensitivity analysis [2], we find it with the numerical differentiation techniques because our modified HB circuit has very complicated structures. Then, our Spice-oriented optimization algorithm is realized by the equivalent circuit, and the optimum point can be found by the equilibrium point in the transient analysis. From our many simulation results, we could find out the optimum point stably when it has unique point.

We show our optimization algorithm and the equivalent circuit model in section III, and interesting examples in section IV, respectively.

[1]Note that the problem of maximization is reduced to the minimization when we change the sign of Φ in (1). Hence, we will only discuss here the problem of minimization.

II. Harmonic balance circuits of nonlinear devices

Analog integrated circuits are usually composed of many kinds of nonlinear devices such as diodes, bipolar transistors and MOSFETs, whose Spice models are described by the several special functions such as exponential, square-root, piecewise continuous functions and so on [14]. The Fourier expansions in these devices cannot be described by analytical forms. On the other hand, these devices in our HB circuits are replaced by the equivalent circuit models using ABMs of Spice which execute the Fourier expansions. To understand our ideas, we consider a two-terminal element described by

$$i = \hat{f}(v). \tag{2}$$

Assume the input and output waveforms as follows;

$$\left.\begin{aligned} v(t) &= V_0 + \sum_{k=1}^{M}\left(V_{2k-1}\cos k\omega t + V_{2k}\sin k\omega t\right) \\ i(t) &= I_0 + \sum_{k=1}^{M}\left(I_{2k-1}\cos k\omega t + I_{2k}\sin k\omega t\right) \end{aligned}\right\}, \tag{3}$$

where M denotes the higher harmonic component to be taken account in the analysis. The output Fourier coefficients are described as follows;

$$\left.\begin{aligned} I_0 &= f_0\left(V_0, V_1, \cdots, V_{2M}\right) \\ I_1 &= f_1\left(V_0, V_1, \cdots, V_{2M}\right) \\ &\cdots\cdots\cdots \\ I_{2M} &= f_{2M}\left(V_0, V_1, \cdots, V_{2M}\right) \end{aligned}\right\}, \tag{4}$$

where the Fourier coefficients can be numerically calculated by the following formulas;

$$\left.\begin{aligned} I_0 &= \frac{1}{2\pi}\int_0^{2\pi} f(v)dt \\ I_{2k-1} &= \frac{1}{2\pi}\int_0^{\pi} f(v)\cos k\omega t dt \\ I_{2k} &= \frac{1}{\pi}\int_0^{2\pi} f(v)\sin k\omega t dt \end{aligned}\right\}, \tag{5}$$

where $k = 1, 2, \cdots, M$. The formulas can be applied to any kind of function even to the piecewise continuous functions [14]. The integrations can be numerically calculated using the trapezoidal integration formula, and that they are realized with ABMs of Spice, schematically [13]. Since bipolar transistors and MOSFETs also have the parasitic capacitors and/or inductors, we need to take account of these elements in the device modules.

A. Parasitic capacitor

Now, assume that a nonlinear capacitor is the voltage-controlled given by

$$q_C = \hat{q_C}\left(v_C\right). \tag{6}$$

We assume the capacitor voltage as follow;

$$v_C = V_{0,C} + \sum_{k=1}^{M}\left(V_{2k-1,C}\cos k\omega t + V_{2k,C}\sin k\omega t\right). \tag{7}$$

Then, we have

$$\left.\begin{aligned} q_C &= Q_{0,C} + \sum_{k=1}^{M}\left(Q_{2k-1,C}\cos k\omega t + Q_{2k,C}\sin k\omega t\right) \\ \Rightarrow i_C &= \sum_{k=1}^{M} k\omega\left(-Q_{2k-1,C}\sin k\omega t + Q_{2k,C}\cos k\omega t\right) \end{aligned}\right\}. \tag{8}$$

Thus, the nonlinear device models consisted of the nonlinear resistors and capacitors are realized by the equivalent circuits with the voltage-controlled current sources given by (4) and (8).

B. Parasitic inductor

In the same way, we assume current-controlled inductor as follows;

$$\phi_L = \hat{\phi}_L\left(i_L\right). \tag{9}$$

We assume the inductor current as follows;

$$i_L = I_{0,L} + \sum_{k=1}^{M}\left(I_{2k-1,L}\cos k\omega t + I_{2k,L}\sin k\omega t\right). \tag{10}$$

Then, we have

$$\left.\begin{aligned} \phi_L &= \Phi_{0,L} + \sum_{k=1}^{M}\left(\Phi_{2k-1,L}\cos k\omega t + \Phi_{2k,L}\sin k\omega t\right) \\ \Rightarrow v_L &= \sum_{k=1}^{M} k\omega\left(-\Phi_{2k-1,L}\sin k\omega t + \Phi_{2k,L}\cos k\omega t\right) \end{aligned}\right\}. \tag{11}$$

In this case, the nonlinear device model consisted of the resisters and inductors are realized by the equivalent circuits with the current-controlled voltage sources given by (4) and (11).

Thus, 3-terminal bipolar transistors modeled by Ebers-Moll model and/or Gummel-Poon model [14] are schematically realized as shown in Fig.1. The corresponding device model to MOSFETs is shown in Fig.2.

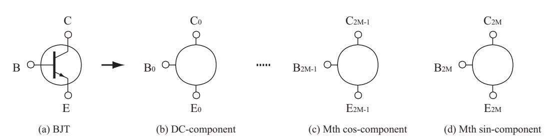

Fig. 1. HB module of the bipolar transistor.

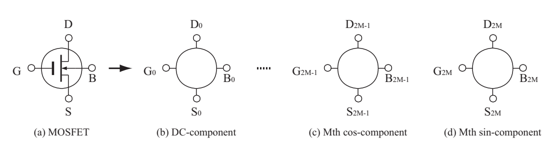

Fig. 2. HB module of the MOSFET.

Note that the HB circuit consists of (2M+1)-subcircuits corresponding to the DC, the fundamental and up to the Mth higher harmonic components.

Once these device modules are installed in our computer library, we can easily formulate the HB circuits, and carry out the frequency-domain analysis.

III. Optimization algorithm

Now, we consider our frequency-domain optimization. Generally, the objective function (1) may be composed of the several requirements such as the maximum output, low distortion, phase margin, and so on. The ratios of requirements p_k should depend on the subject, in advance. Thus, the object function is described as follow;

$$\Phi(\boldsymbol{x}, \boldsymbol{y}) = \sum_{k=1}^{K} p_k \phi_k(\boldsymbol{x}, \boldsymbol{y}), \tag{12}$$

where the signs of p_k should be chosen properly,

$$|p_k| < 1, \tag{13}$$

and K is a number of requirements. In this case, $\boldsymbol{x}$ is the circuit variables such as node voltages and/or currents which are estimated by the solutions from the HB circuit in section II, whose circuit equation is given as follows;

$$\boldsymbol{F}(\boldsymbol{x}, \boldsymbol{y}) = \boldsymbol{0},\ \boldsymbol{F}(\cdot, \cdot) : \boldsymbol{R}^n \mapsto \boldsymbol{R}^n. \tag{14}$$

Then, the optimum point satisfies the following relations;

$$\frac{\partial \Phi}{\partial y_k} = 0,\ (k = 1, 2, \cdots, m). \tag{15}$$

When we use the steepest descent algorithm, the descent direction is given by $-\frac{\partial \Phi}{\partial y_k} < 0,\ (k = 1, 2, \cdots, m)$ in the parameter space $\boldsymbol{y} \in \boldsymbol{R}^m$. Although we can apply the sensitivity analysis to find the direction of steepest descent, our HB circuit containing bipolar transistors and MOSFETs are too complicated for the sensitivity analysis. Therefore, we apply the numerical differentiations to the calculation of the descent directions. Namely, we have for a variable s,

$$\left.\begin{aligned} \frac{\partial y_1}{\partial s} &= -\frac{\Phi_1(\boldsymbol{x}, y_1 + \Delta y_1, y_2, \cdots, y_m) - \Phi_0(\boldsymbol{x}, \boldsymbol{y})}{\Delta y_1} \\ \frac{\partial y_2}{\partial s} &= -\frac{\Phi_2(\boldsymbol{x}, y_1, y_2 + \Delta y_2, \cdots, y_m) - \Phi_0(\boldsymbol{x}, \boldsymbol{y})}{\Delta y_2} \\ &\cdots\cdots\cdots \\ \frac{\partial y_m}{\partial s} &= -\frac{\Phi_m(\boldsymbol{x}, y_1, y_2, \cdots, y_m + \Delta y_m) - \Phi_0(\boldsymbol{x}, \boldsymbol{y})}{\Delta y_m} \end{aligned}\right\}. \tag{16}$$

The flowchart of our optimization algorithm is shown in Fig.3.

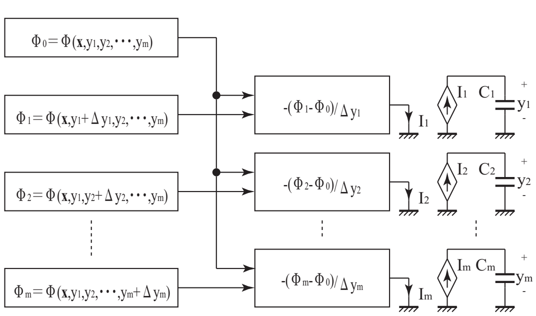

Fig. 3. Flowchart of our steepest descent method.

The objective function $\Phi_0(\boldsymbol{x}, \boldsymbol{y})$ is estimated by the circuit variables $\boldsymbol{x}$ and the parameter values $\boldsymbol{y}$. The parameters $\boldsymbol{y}$ are obtained as the voltages at the capacitors in our simulator which satisfy

$$\left.\begin{aligned} C_1 \frac{dy_1}{dt} &= -\frac{\Phi_1 - \Phi_0}{\Delta y_1} \\ C_2 \frac{dy_2}{dt} &= -\frac{\Phi_2 - \Phi_0}{\Delta y_2} \\ &\cdots\cdots\cdots \\ C_m \frac{dy_m}{dt} &= -\frac{\Phi_m - \Phi_0}{\Delta y_m} \end{aligned}\right\}. \tag{17}$$

Note that the capacitor values play to improve the convergence ratios in our optimization.

IV. Illustrative examples

To understand the ideas of our optimization algorithm, let us consider a simple problem to find out the optimum capacitance and inductance giving the maximum power consumption at R_L in Fig.4.

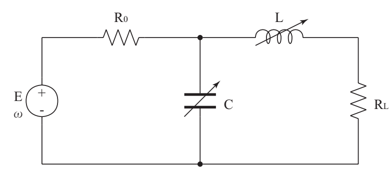

Fig. 4. A simple optimization problem.

They are theoretically given as follows;

$$L = \frac{1}{\omega}\sqrt{R_L\,(R_0 - R_L)},\quad C = \frac{1}{\omega R_0}\sqrt{\frac{R_0 - R_L}{R_L}} \tag{18}$$

Now, let us find the values with our optimization approach shown in Fig.5.

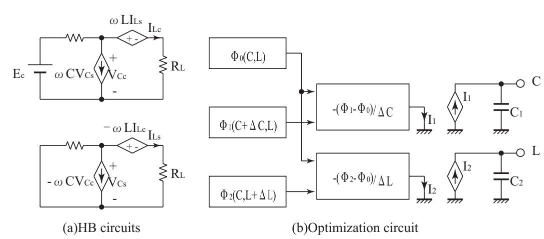

Fig. 5. Optimization circuit,
$\omega = 1\text{rad/s}$, $E_c = 1\text{V}$, $R_L = 1\Omega$, $R_0 = 3\Omega$.

Fig.5(a) shows the Cosine and Sine subcircuits [13] in our HB circuit. Since the circuit is driven by $E_C \cos \omega t$, the Cosine circuit has a DC source E_C. C and L are replaced by the controlled sources, respectively. The objective function is defined by

$$\Phi\,(C, L) = R_L\left(I_{Lc}^2 + I_{Ls}^2\right). \tag{19}$$

Then, the optimization circuit shown in Fig.5(b) consists of 3 subcircuits $\Phi\,(C, L)$, $\Phi\,(C + \Delta C, L)$, $\Phi\,(C, L + \Delta L)$, where C and L are given by the voltages of capacitors C_1 and C_2, respectively The result with the transient analysis of Spice is shown in Fig.6. The values C and L are exactly equal to those values from (18).

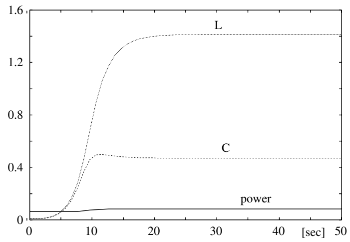

Fig. 6. Spice simulation results, $\Delta C = 0.001$, $\Delta L = 0.001$.

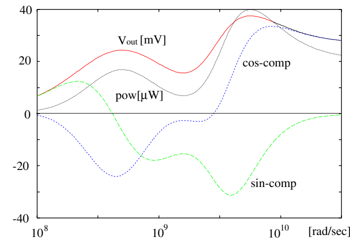

Fig. 8. Frequency response curve, $I_{CC} = 1\text{mA}$, $R_L = 36.22\Omega$, $v_{in} = 0.1 \cos \omega t$.

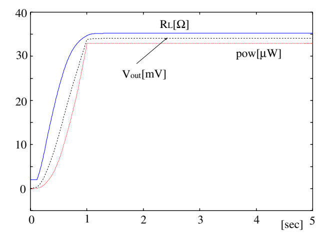

Fig. 9. The result of our steepest descent method.

Next, we consider a high frequency RF power amplifier shown in Fig.7 which is used in mobile-phones [5], where L_gs are the parasitic inductances. We want to optimize the circuit parameters in such manner that the output power consumption at R_L becomes maximum by adjusting R_L and I_{CC}. We first modeled the transistors with Gummel-Poon model, and obtained the HB modules, where we consider up to the third higher harmonic components.

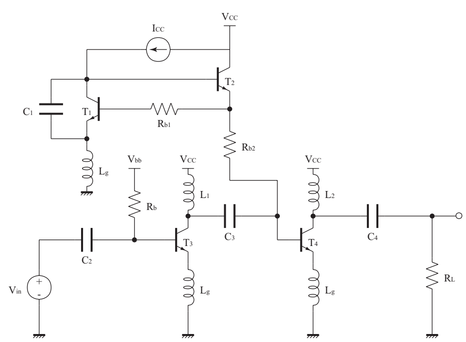

Fig. 7. RF power amplifier, $L_1 = L_2 = 0.1\text{mH}$, $L_G = 10\text{nH}$, $C_1 = C_2 = C_3 = C_4 = 1\text{nF}$, $V_{CC} = 3\text{V}$, $V_{bb} = 3\text{V}$, $R_{b1} = R_{b2} = 1\Omega$, $R_b = 1\Omega$.

We have calculated the frequency response curve in the region $\omega = 10^8 - 10^{11}$. The waveform is nearly sinusoidal, and the amplitudes are exactly equal to the result from transient analysis. In our optimization algorithm for getting the maximum output power, we have changed the current sources and the load resistor in the region $I_{CC} = 1mA - 7mA$, and $R_L = 0.1 - 70\Omega$, respectively. Thus, we can stably find out the optimum point as shown in Fig.9.

V. Conclusions and Remarks

In this paper, we have proposed a new Spice-oriented frequency-domain optimization technique, whose objective function can be obtained from the HB circuit. We first transform the nonlinear devices such as bipolar transistors and MOSFETs into the corresponding HB modules. Our optimization technique depends on the steepest descent algorithm which is realized by the equivalent circuit. The optimum point is given as the equilibrium point the transient response, which can be stably solved by the transient analysis of Spice.

For future problem, we need to extend our algorithm to the problems with the multiple optimum points.

References

[1] R. K. Brayton and R. Spence, *Sensitivity and Optimization*, Elsevier Scientific Pub. co., 1980.

[2] G. D. Vendelin, A. M. Pavio and U. L. Rohde, *Microwave Circuit Design using Linear and Nonlinear Techniques*, John Wiley and Sons, Inc., 1990.

[3] J. Wood and D. E. Root, *Fundamentals of Nonlinear Behavior Modeling for RF and Microwave Design*, Artech House, Inc., 2005.

[4] D. Leenaerts, J. van der Tang and C. Vaucher, *Circuit Design for RF Transceivers*, Kluwer Academic Publishers, 2001.

[5] H. Floberg, *Symbolic Analysis in Analog Integrated Circuit Design*, Kluwer Academic Publishers, 1997.

[6] P. Wambacq and W. Sansen, *Distortion Analysis of Analog Integrated Circuits*, Kluwer Academic Publishers, 1998.

[7] D. D. Weiner and J. E. Spina, *Sinusoidal Analysis and Modeling of Weakly Nonlinear Circuits with Application to Nonlinear Interference Effects*, Van Nostrand Reinhold, 1980.

[8] L. O. Chua and C-Y. Ng, "Frequency-domain analysis of nonlinear systems: formulation of transfer functions", *IEE Trans. Electronic Circuits and Systems*, vol.3, no.6, pp.257-269, 1979.

[9] K. S. Kundert, J. K. White and A. Sangiovanni-Vincentelli, *Steady-State Methods for Simulating Analog and microwave circuits*, Kluwer Academic Publishers, 1990.

[10] A. Ushida, T. Adachi and L. O. Chua, "Steady-state analysis of nonlinear circuits based on hybrid methods", *IEEE Trans. on Circuits and System-I: Fundamental Theory and Applications*, vol.1, pp.649-661, 1992.

[11] A. Ushida, Y. Yamagami and Y. Nishio, "Frequency responses of nonlinear networks using curve tracing algorithm", *ISCAS 2002*, vol.1, pp.641-644, 2002.

[12] R. J. Gilmor and M. B. Steer, "Nonlinear circuit analysis using the method of harmonic balance- A review of the art. Part.1. Introductory concepts", *Int. Jour. of Microwave and Millimeter-Wave Computer-Aided Eng.*, vol.1, no.1, pp.22-37, 1991.

[13] R. J. Gilmor and M. B. Steer, "Nonlinear circuit analysis using the method of harmonic balance- A review of the art. Part.2. Introductory concepts", *Int. Jour. of Microwave and Millimeter-Wave Computer-Aided Eng.*, vol.1, no.2, pp.159-180, 1991.

[14] R. L. Geiger, P. E. Allen and N. R. Strader, *VLSI: Design Techniques for Analog and Digital Circuits*, McGraw-Hill, 1990.