NOLTA 2006
11-14 september
Bologna - ITALY

On Optimization Algorithm for Attaining The Maximum DC Gain of CMOS Amplifiers

M. Oda[†], *Y. Yamagami*[†], *Y. Nishio*[†], *J. Kawata*[‡], *A. Ushida*[‡]

† Dept. of Electrical and Electronic Engineering, Tokushima University, Tokushima 770-8506, Japan
‡ Dept. of Mechanical and Elecronic Engineering, Tokushima Bunri University, Kagawa 769-2193, Japan

Abstract—
CMOS operational amplifiers are most important building blocks of analog circuits. In this paper, we propose a Spice-oriented design algorithm of CMOS amplifies for attaining the maximum DC gain. Our optimization algorithm is based on well-known steepest descent method, where the gradient direction is decided by the analysis of sensitivity circuits for the designing parameters such as MOS sizes and bias voltages. In order to develop the user-friendly designing tool, we have firstly tried to replace MOSFETs by the corresponding sensitivity modules. After then, our steepest descent algorithm is realized by equivalent nonlinear RC circuits with ABMs (analog behavior models), so that the solution can be found as an equilibrium point with transient analysis of Spice. We found from some numerical experiments that the optimum points can be stably obtained by the transient analysis.

1. Introduction

CMOS operational amplifiers are widely used as the building blocks in analog circuits [1-4]. For designing the amplifiers, there are several design criteria such as the maximum gain, the minimum power consumption and so on. For example of attaining the maximum gain of an amplifier, the gain largely depends on the bias voltages and MOS sizes, so that they are chosen as the design parameters. Traditionally, the optimum parameters were found by trial and error process with Spice. It was really time-consuming when the design parameters are increased [6]. On the other hand, the steepest descent method is widely used for this kind of problems [5]. However, it is very efficient only when the objective functions are given in the analytical form.

Hence, in this paper, we propose a practical steepest descent algorithm based on Spice combining the sensitivity analysis. Firstly, we need to define the *objective function* as follow:

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

where
$\boldsymbol{x}$: circuit variables such as voltages and currents.
$\boldsymbol{p}$: optimization parameters such as bias voltages, resistor's values and the dimensions of MOSFETs W[width in μm] and L [length μm] and so on [6,7]
Now, let us discuss the design algorithm of amplifiers for attaining the maximum DC gain based on Spice-oriented steepest descent method [5,7]. Namely, the gradient direction is decided by the solutions of sensitivity circuits [8], and the steepest descent algorithm can be realized by the equivalent RC circuits combining with nonlinear controlled current sources. In order to develop user-friendly simulators, we tried to transforms MOSFETs into the corresponding *sensitivity modules.* The optimization algorithm to attain the maximum gain is given by

$$\frac{dp_i}{ds} = -\frac{dS_v(\boldsymbol{p})}{dp_i}, \quad i = 1, 2, \ldots, k, \tag{2}$$

where $S(\boldsymbol{p})$ is the sensitivity gain and p_i are optimization parameters. Unfortunately, it is impossible to evaluate $\frac{dS(\boldsymbol{p})}{dp_i}$ from the sensitivity analysis, so that we need to introduce the *numerical differentiation* as follows:

$$\begin{aligned} \frac{dp_i}{ds} &= -\frac{S_v(\mathbf{p} + \Delta\mathbf{p}) - S_v(\mathbf{p})}{\Delta p_i}, \quad i = 1, 2, \ldots, k, \\ \Delta\boldsymbol{p} &= (0, 0, \ldots, \Delta p_i, \ldots, 0)^T \end{aligned} \tag{3}$$

with sufficiently small $\Delta\boldsymbol{p}$. Then, replacing a variable "s" by time "t", our descent algorithm can be realized by the equivalent nonlinear RC circuits with the solutions of the sensitivity circuit and ABMs of Spice[9]. In this way, we can find out the optimum parameters at the equilibrium point from the transient analysis.

We show the sensitivity circuit using the tableau approach, and the above optimization algorithm in section 2. The sensitivity modules of MOSFETs are shown in section 3. They are very useful to develop user-friendly simulators. We show interesting examples of inverter amplifier to attain the maximum gain and CMOS amplifier in section 4.

2. Sensitivity analysis and Spice-oriented optimization algorithm

Sensitivity analysis

The steepest descent method is the most basic optimization approach, where the gradient direction is decided by solutions of the sensitivity circuit. Let us derive the sensitivity circuit via tableau approach [8]. The *tableau equation* is given by:

$$\begin{bmatrix} \boldsymbol{K}_i & \boldsymbol{K}_v & \boldsymbol{0} \\ \boldsymbol{0} & \boldsymbol{1} & -\boldsymbol{A}^T \\ \boldsymbol{A} & \boldsymbol{0} & \boldsymbol{0} \end{bmatrix} \begin{bmatrix} \boldsymbol{i} \\ \boldsymbol{v} \\ \boldsymbol{v}_n \end{bmatrix} - \begin{bmatrix} \boldsymbol{g}(\boldsymbol{v}, \boldsymbol{i}) \\ \boldsymbol{E} \\ \boldsymbol{AJ} \end{bmatrix} = \begin{bmatrix} \boldsymbol{0} \\ \boldsymbol{0} \\ \boldsymbol{0} \end{bmatrix} \tag{4}$$

The first row is coming from the Ohm's low, second and third are Kirchhoff's voltage and current lows, respectively. $\boldsymbol{A}$ is the incidence matrix and $\boldsymbol{K}_i$, $\boldsymbol{K}_v$ are $(b \times b)$ square matrix composed of "1" and "0" for b elements. $\boldsymbol{E}$ and $\boldsymbol{J}$

show the voltage and current sources. Now, we define the *sensitivity* as follow:
Sensitivity: Let us define the sensitivity by

$$S_{k,i} \equiv \lim_{\Delta p_i \to 0} \frac{\Delta x_k}{\Delta p_i} \tag{5}$$

Now, let us calculate the variational equation from (3). For parameter variations of the nonlinear resistors, we have

$$\begin{aligned} &\boldsymbol{g}(\boldsymbol{v}_0 + \Delta \boldsymbol{v}, \boldsymbol{i}_0 + \Delta \boldsymbol{i}) + \left.\tfrac{\partial \boldsymbol{g}(\boldsymbol{v},\boldsymbol{i})}{\partial \boldsymbol{p}}\right|_{\boldsymbol{v}_0,\boldsymbol{i}_0} \Delta \boldsymbol{p} \\ &\simeq \boldsymbol{g}(\boldsymbol{v}_0, \boldsymbol{i}_0) + \left.\tfrac{\partial \boldsymbol{g}(\boldsymbol{v},\boldsymbol{i})}{\partial \boldsymbol{v}}\right|_{\boldsymbol{v}_0,\boldsymbol{i}_0} \Delta \mathbf{v} \\ &+ \left.\tfrac{\partial \boldsymbol{g}(\boldsymbol{v},\boldsymbol{i})}{\partial \boldsymbol{i}}\right|_{\boldsymbol{v}_0,\boldsymbol{i}_0} \Delta \boldsymbol{i} + \left.\tfrac{\partial \boldsymbol{g}(\boldsymbol{v},\boldsymbol{i})}{\partial \boldsymbol{p}}\right|_{\boldsymbol{v}_0,\boldsymbol{i}_0} \Delta \boldsymbol{p} \end{aligned} \tag{6}$$

Thus, we have the following circuit equation for calculating the sensitivities:

$$\begin{aligned} &\begin{bmatrix} \boldsymbol{K}_i & \boldsymbol{K}_v & \mathbf{0} \\ \mathbf{0} & \mathbf{1} & -\boldsymbol{A}^T \\ \boldsymbol{A} & \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \boldsymbol{S}_{i,p_i} \\ \boldsymbol{S}_{v,p_i} \\ \boldsymbol{S}_{v_n,p_i} \end{bmatrix} - \begin{bmatrix} \left.\frac{\partial \boldsymbol{g}(\boldsymbol{v},\boldsymbol{i})}{\partial \boldsymbol{v}}\right|_{\boldsymbol{v}_0,\boldsymbol{i}_0} \boldsymbol{S}_{v,p_i} \\ \mathbf{0} \\ \mathbf{0} \end{bmatrix} \\ &- \begin{bmatrix} \left.\frac{\partial \mathbf{g}(\mathbf{v},\mathbf{i})}{\partial \mathbf{i}}\right|_{\mathbf{v}_0,\mathbf{i}_0} \mathbf{S}_{i,p_i} \\ \delta(i) \\ \boldsymbol{A}\boldsymbol{\delta}(i) \end{bmatrix} = \begin{bmatrix} \left.\frac{\partial \boldsymbol{g}(\boldsymbol{v},\boldsymbol{i})}{\partial \boldsymbol{p}}\right|_{\boldsymbol{v}_0,\boldsymbol{i}_0} \delta(i) \\ \mathbf{0} \\ \mathbf{0} \end{bmatrix}, \end{aligned} \tag{7}$$

where $\boldsymbol{\delta}(i)$ means a delta function satisfying

$$\begin{matrix} & 1 & 2 & \dots & i & \dots & k & \\ \boldsymbol{\delta}(i) = [& 0, & 0, & \dots, & 1 & \dots, & 0 &]^T. \end{matrix}$$

Thus, from (6), we can derive the sensitivity circuit for ith p_i parameter. Note that the circuit configuration is equal to the original one except for the nonlinear elements being replaced by the linear incremental resistors at the operating points $\boldsymbol{V}_0, \boldsymbol{I}_0$. Now, we summarize the algorithm for deriving the sensitivity circuit.

1. When a voltage source is chosen as an optimization parameter, it is set to "1[V]". Other sources are removed as the short-circuits.
2. When a current source is chosen as an optimization parameter, it is set to "1[A]". Other sources are removed as the open-circuits.
3. When a resistor chosen as an optimization parameter, it is replaced by the linear incremental resistor with the controlled source $\left.\frac{\partial \boldsymbol{g}(\boldsymbol{v},\boldsymbol{i})}{\partial \boldsymbol{p}}\right|_{\boldsymbol{v}_0,\boldsymbol{i}_0} \boldsymbol{\delta}(i)$. Other resistors are only replaced by the incremental resistors. These transformations are shown in Fig.1.

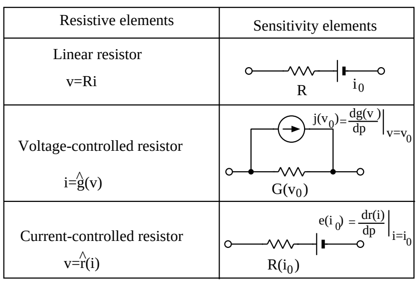
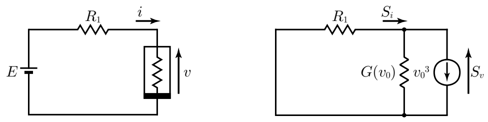


Fig.1. Sensitivity elements for resistors.

Example: Now, consider a simple example for deriving the sensitivity circuit. We assume that c_3 in nonlinear resistor

$$i = c_1 v + c_3 v^3$$

is chosen as an optimization parameter. Then, the sensitivity circuit is given by Fig.2(b).



(a) nonlinear resistive circuit. (b) Sensitivity circuit.
Fig.2. Sensitivity circuit.

Variational method: We perturb $c_3 \Rightarrow c_3 + \Delta c_3$, then, we have the following circuit equation and the variational equation:

$$c_1(v_0 + \Delta v) + (c_3 + \Delta c_3)(v_0 + \Delta v)^3 + \frac{(v_0 + \Delta v) - E}{R_1} = 0$$

$$\Rightarrow c_1 \Delta v + \Delta c_3 v_0^3 + 3 c_3 v_0^2 \Delta v + \frac{\Delta v}{R_1} = 0$$

Therefore, the sensitivity of the resistor voltage is given by

$$S_v = \lim_{\Delta c_3 \to 0} \frac{\Delta v}{\Delta c_3} = -\frac{v_0^3}{G(v_0) + \frac{1}{R_1}}, \quad \text{for} \quad G(v_0) = c_1 + 3c_3 v_0^2$$

Sensitivity analysis: From the sensitivity circuit Fig.2(b) with $\partial i / \partial c_3|_{v_0} = v_0^3$, we have the voltage drop at the resistor $G(v_0)$ is given

$$S_v = -\frac{v_0^3}{G(v_0) + \frac{1}{R_1}}$$

which is exactly equal to the solution from the variational method.

Spice-oriented optimization technique

Now, we consider optimization problems of DC circuits, and define the *objective function* as follows:

$$\Phi(\boldsymbol{v}, \boldsymbol{i}, \boldsymbol{p}) = \hat{\phi}(v_1, v_2, \dots, v_n, i_1, i_2, \dots, i_n, p_1, p_2, \dots, p_k) \tag{8}$$

where $\boldsymbol{v}$, $\boldsymbol{i}$ are circuit variables, and $\boldsymbol{p}$ is the optimization parameters. Applying the steepest descent method to (7), we have

$$\left.\begin{aligned} \tfrac{\partial p_1}{\partial s} &= -\hat{\phi}(S_{v_1,p_1}, \dots, S_{v_n,p_1}, S_{i_1,p_1}, \dots, S_{i_n,p_1}, 1, 0, \dots, 0) \\ \tfrac{\partial p_2}{\partial s} &= -\hat{\phi}(S_{v_1,p_2}, \dots, S_{v_n,p_2}, S_{i_1,p_2}, \dots, S_{i_n,p_2}, 0, 1, \dots, 0) \\ &\cdots\cdots\cdots\cdots\cdots \\ \tfrac{\partial p_k}{\partial s} &= -\hat{\phi}(S_{v_1,p_k}, \dots, S_{v_n,p_k}, S_{i_1,p_k}, \dots, S_{i_n,p_k}, 0, 0, \dots, 1) \end{aligned}\right\} \tag{9}$$

where $\{S_{v_i,p_j}, S_{i_i,p_j}, i = 1, 2, \dots, n, j = 1, 2, \dots, k\}$ are solutions from the sensitivity analysis. Note that we need k times sensitivity analysis to execute the steepest descent method. Changing variable "s" with time "t", the equation (8) is realized by a coupled RC circuits using controlled-current sources as shown in Fig.3, where the value capacitance Cs are related to the convergence speed, and R_0s are sufficient large dummy resistances. On the other hand,

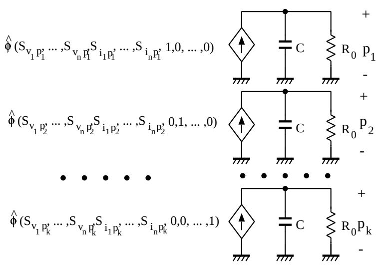


Fig.3. Circuit configuration of the steepest decent method.

the optimization attaining the maximum gain is realized by the use of numerical differentiations as given in (2). The relation (2) is also realized by the circuit shown in Fig.4 with small Δp_i. where $\boldsymbol{S}(\boldsymbol{p} + \delta(\Delta p_i))$ and $\boldsymbol{S}(\boldsymbol{p})$ are so-

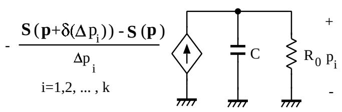


Fig.4. Circuit configuration of attaining the maximum gain.

lutions from the sensitivity analysis. Thus, the optimum point can be found by the transient analysis of Spice.

3. Sensitivity modules of MOSFETs

Now, let us derive the sensitivity modules of MOSFETs in DC. Once these modules are stored in our computer as the packages, our optimization algorithms can be easily carried out with Spice. We apply piecewise continuous Shichman-Hodges model [1-4] for nMOS as follows:

1. **Linear region** $(v_{GS} > v_T,\ 0 < v_{DS} < v_{GS} - v_T)$

$$i_D = \frac{k_n W}{L}\left[(v_{GS} - v_T) - \frac{v_{DS}}{2}\right] v_{DS}(1 + \lambda v_{DS}) \quad (10)$$

2. **Saturation region**$(v_{GS} > v_T,\ v_{DS} \geq v_{GS} - v_T)$

$$i_D = \frac{k_n W}{2L}(v_{GS} - v_T)^2(1 + \lambda v_{DS}) \quad (11)$$

where the threshold voltage is given by

$$v_t = v_{T0} + \gamma\left(\sqrt{\phi - v_{BS}} - \sqrt{\phi}\right) \quad (12)$$

Thus, we have the following sensitivity elements at the corresponding operating points:

1. **Linear region**

$$G_{GS} \equiv \frac{\partial i_D}{\partial v_{GS}} = \frac{k_n W}{L} v_{DS0}(1 + \lambda v_{DS0}) \quad (13)$$

$$G_{DS} \equiv \frac{\partial i_D}{\partial v_{DS}} = \frac{k_n W}{L} \times \left[-\frac{3}{2}\lambda v_{DS0}^2 - v_{DS0} + (1 + 2\lambda)(v_{GS0} - v_{T0})\right] \quad (14)$$

$$G_{BS} \equiv \frac{\partial i_D}{\partial v_{BS}} = \frac{\gamma k_n W}{2L} \frac{v_{DS0}(1 + \lambda v_{DS0})}{\sqrt{\phi - v_{BS0}}} \quad (15)$$

2. **Saturation region**

$$G_{GS} \equiv \frac{\partial i_D}{\partial v_{GS}} = \frac{k_n W}{L}(v_{GS0} - v_{T0})(1 + \lambda v_{DS0}) \quad (16)$$

$$G_{DS} \equiv \frac{\partial i_D}{\partial v_{DS}} = \frac{k_n W}{2L}(v_{GS0} - v_{T0})^2 \lambda \quad (17)$$

$$G_{BS} \equiv \frac{\partial i_D}{\partial v_{BS}} = \frac{\gamma k_n W}{2L} \frac{(v_{GS0} - v_{T0})(1 + \lambda v_{DS0})}{\sqrt{\phi - v_{BS0}}} \quad (18)$$

For the optimization parameter (W,L), we have the following current sensitivities:

1. **Linear region**

$$S_{in} = \frac{k_n}{L}\left[(v_{GS0} - v_{T0}) - \frac{v_{DS0}}{2}\right] v_{DS0}(1+\lambda v_{DS0}), \text{ for } L, \quad (19)$$

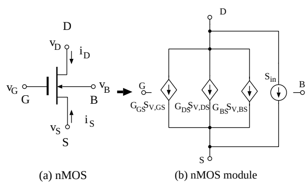
$$S_{in} = -\frac{k_n W}{L^2}\left[(v_{GS0} - v_{T0}) - \frac{v_{DS0}}{2}\right] v_{DS0}(1+\lambda v_{DS0}), \text{ for } W, \quad (20)$$

2. **Saturation region**

$$S_{in} = \frac{k_n}{2L}(v_{GS0} - v_{T0})^2(1 + \lambda v_{DS0}), \text{ for } L, \quad (21)$$

$$S_{in} = -\frac{k_n W}{2L^2}(v_{GS0} - v_{T0})^2(1 + \lambda v_{DS0}), \text{ for } L, \quad (22)$$

where "0" means voltage at the operating point. These modules are realized with ABMs of Spice, whose nMOS module is shown by Fig.5 (b). We can also realize pMOS

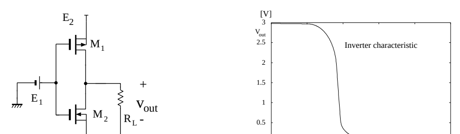


Fog.5. (a) nMOS, (b) nMOS module.

sensitivity modules in the same way.

4. Illustrative examples.

CMOS inverter amplifier.
Most of CMOS operational amplifiers contain some kinds of inverters Fig.6(a) as the building blocks, whose transfer characteristic is given by Fig.6(b), where MOS parameters are given by Table 1.



(a) CMOS inverter. (b) Input-output characteristic.
Fig.6. CMOS inverter.

Table.1 MOS parameter[9] (MKS system).

Mark	pMOS	nMOS
L	1.2×10^{-6}	1.2×10^{-6}
W	7.8×10^{-6}	7.8×10^{-6}
L_D	900.1×10^{-12}	900.1×10^{-12}
pV_{T0}, nV_{T0}	-0.8311	0.6081
k_p, K_n	19.34×10^{-6}	74.21×10^{-6}
λ	0.1	0.2
$p\gamma, n\gamma$	0.3046	0.6166
C_{j0}	259.97×10^{-6}	280.65×10^{-6}

Other parameters
$$A_S = A_D = 20 \times 10^{-12}, \quad t_{ox} = 30.4 \times 10^{-9}, \, \phi = 0.7$$
$$C_{ox} = 1.135 \times 10^{-3}$$

The gain is given by the following relation

$$G_{out} = \frac{dV_{out}}{dE_1}. \tag{23}$$

If the input voltage is set at series to E_1. It seems to have the maximum around $E_1 = 0.37[V]$. Now, let us calculate the maximum gain with the steepest descent method shown is section 2.2. Our Spice-oriented optimization algorithm is realized the equivalent circuit shown by Fig.7, where the above 3 circuits are inverters, and the below 3 are corresponding sensitivity circuits, where they are set to $E_1 = 1[V]$ in the sensitivity analysis because of the gain ofE_1. The optimum point is obtained by the following

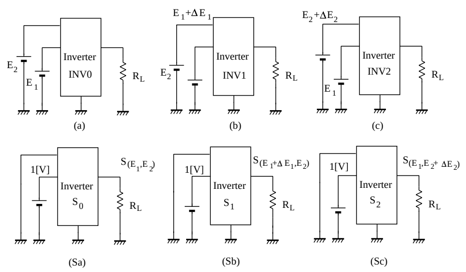


Fig.7. Equivalent circuit for getting the maximum gain.

steepest descent method shown Fig.4.

$$\left.\begin{array}{l} C\frac{dE_1}{dt} = -\frac{S_v(E_1+\Delta E_1,E_2)-S_v(E_1,E_2)}{\Delta E_1} \\ C\frac{dE_2}{dt} = -\frac{S_v(E_1,E_2+\Delta E_2,)-S_v(E_1,E_2)}{\Delta E_2} \end{array}\right\} \tag{24}$$

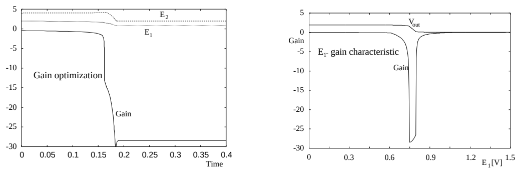


Fig.8 (a) Gain optimization (b) Gain sensitivity.

$E_1 = 0.74696[V]$, $E_2 = 1.9467[V]$, $G_{out,max} = -28.444$, $C = 0.01$, $R_L = 1000[k\Omega]$.

Thus, the optimum point corresponds to the equilibrium point, which is found by the transient analysis of Spice. To investigate the result, we calculate the gain characteristic for the variable E_1 with $E_2 = 1.946$ as shown in Fig.8(b). The result is exactly equal to the result from Fig.8(a).

AB amplifier

Now, we consider AB amplifier shown in Fig.9(a) [1]. It is a class of differential amplifier and consisted of 14 MOSFETs. At first, we optimize the gain by changing the sizes (nL, nW, pL, pW) between $1\mu m < L, W < 10\mu m$, where we classify the transistors into 3 groups according to pMOS mirror circuits (1,2,3,4,5,6), nMOS differential inputs and mirror circuit (1,3,4,5,6,8) and nMOS current sources (2,7). We also consider the bias voltage V_B so that we optimized seven parameters and have the result shown by Fig.9 (c).

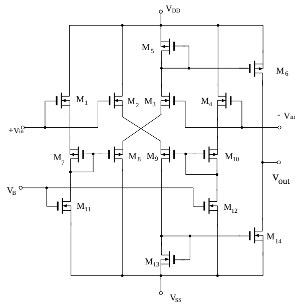


Fig.9(a) CMOS AB amplifier.

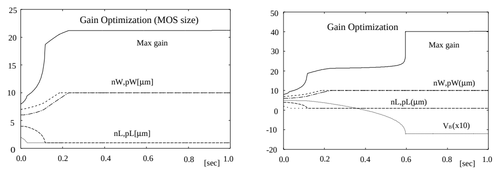


Fig.9(b) Optimization for sizes, Max Gain =21.21, (c) Optimization for sizes and bias V_B.

$pL = nL = 1\mu m, \quad pW = nW = 10\mu m, \quad V_B = -1.206[V], \quad Max\ Gain = 40.12$

5. Conclusions and remarks

In this paper, we propose a Spice-oriented design algorithm of CMOS amplifies for attaining the maximum DC gain. The optimization algorithm is based on the steepest descent method such that the gradient direction is decided by the analysis of the sensitivity circuit. Since our optimization method is realized by the nonlinear RC circuits with ABMs, the solution is found by the transient analysis of Spice. From our numerical experiments, we can get the solution, stably. For future work, we will apply our method to bipolar transistor circuits and the others.

References

[1] G.Roubik, *Introduction to CMOS OP-AMPs and Comparators,* Wiley-Interscience Pub., 1999.

[2] F.Maloberti, *Analog Design for CMOS VLSI Systems,* KLuwer Academic Pub., 2001.

[3] S.M.Kang, and Y.Leblebici, *CMOSW Digital Integrated Circuits: Analysis and Design,* McGraw-Hill Co.Inc., 1996.

[4] A.S.Sedra and K.C.Smith, *Microelectronic Circuits,* Oxford Univ. Press, 2004.

[5] J.A.Snyman *Practical Mathematical Optimization: An Introduction to Basic Optimization Theory and Classical and New Gradient-Based Algorithm,* Springer, 2005.

[6] K.Ono, K.Saeki,Y.Sekine,"A design of an operational amplifier with high DC gain," *The Papers of Tech. meeting on Electronic Circuits, IEE Japan,* ECT-04-99, pp.1-5, 2004.

[7] M.Oda, Y.Yamagami, Y.Nishio, J.Kawata and A.Ushida, "A Spice-oriented frequency-domain optimization," *ISCAS'06*, to appear, 2006.

[8] L.O.Chua and P-M Lin, *Computer-Aided Analysis of Electronic Circuits: Algorithms and Computational Techniques,* Prentice-Hall, 1975.

[9] MicroSim: *PSpice A/D circuit Analysis User's Guides,* MicroSim Co. 1995.