IEICE TRANS. FUNDAMENTALS, VOL.E90–A, NO.2 FEBRUARY 2007

LETTER *Special Section on Analog Circuit Techniques and Related Topics*

Spice-Oriented Frequency-Domain Analysis of Nonlinear Electronic Circuits

Junji KAWATA[†a)], **Yousuke TANIGUCHI**[††b)], **Masayoshi ODA**[††c)], **Yoshihiro YAMAGAMI**[††d)], **Yoshifumi NISHIO**[††e)], ***Members***, ***and*** **Akio USHIDA**[†f)], ***Fellow***

SUMMARY Distortion analysis of nonlinear circuits is very important for designing analog integrated circuits and communication systems. In this letter, we propose an efficient frequency-domain approach for calculating frequency response curves, which is based on HB (*harmonic balance*) method combining with ABMs (Analog Behavior Models) of Spice. Firstly, nonlinear devices such as bipolar transistors and MOSFETs are transformed into the *HB device modules* executing the Fourier transformations. Using these modules, the *determining equation of the HB method* is formed by the *equivalent sine-cosine circuit* in the schematic form or netlist. It consists of the coupled resistive circuits, so that it can be efficiently solved by the DC analysis of Spice. In our algorithm, we need not to derive any troublesome circuit equations, and any kinds of the transformations.

key words: *frequency-domain analysis, Volterra series method, spice-oriented harmonic balance method, HB device modules*

1. Introduction

Frequency-domain analysis of nonlinear electronic circuits is very important for designing integrated circuits and communication systems. The Volterra series methods are widely used for the analysis [1]–[3], especially in Europe, because the methods give the solutions in analytical forms. Although their algorithms are theoretically elegant, it is not so easy to derive the higher order Volterra kernels for the large scale systems containing many nonlinear elements. They can be only applied to the weakly nonlinear circuits, and the solutions may be erroneous for strong nonlinear circuits, because the methods are based on the bilinear theorem. Furthermore, the nonlinear characteristics should be approximated by the polynomial functions, where Taylor expansions are used to the approximations at the vicinities of DC operating points [3]. However, these tasks are not easy for the complicated devices such as Gummel-Poon model of bipolar transistors, higher frequency models of MOSFETs and the piecewise linear models [4], [5].

Manuscript received September 8, 2006.
Final manuscript received October 10, 2006.

[†]The authors are with the Department of Mechanical and Electronic Engineering, Faculty of Engineering, Tokushima Bunri University, Sanuki-shi, 769-2193 Japan.
[††]The authors are with the Department of Electrical and Electronic Engineering, Faculty of Engineering, Tokushima University, Tokushima-shi, 770-8506 Japan.
a) E-mail: kawata@fe.bunri-u.ac.jp
b) E-mail: taniguchi@ee.tokushima-u.ac.jp
c) E-mail: masayoshi@ee.tokushima-u.ac.jp
d) E-mail: yamagami@ee.tokushima-u.ac.jp
e) E-mail: nishio@ee.tokushima-u.ac.jp
f) E-mail: ushida@fe.bunri-u.ac.jp

DOI: 10.1093/ietfec/e90–a.2.406

On the other hand, many algorithms in the time-domain have been proposed for calculating accurate steady-state waveforms [6], [7]. Unfortunately, they are rather time-consuming for the cases of calculating frequency response curves.

HB (harmonic balance) method is well-known, which gives good results even for choosing a few essential harmonic components [8], [9]. Thus, the HB method, in contrast to Volterra method, can be stably applied to strong nonlinear systems. Two types of HB simulators are proposed for the steady-state analysis of electronic circuits. First one is a nodal type in the meaning that HB method is applied to all the nodal equations [6], [10]–[12]. The second one is based on circuit partitioning techniques [7], [13]–[15]. They are using DFT or FFT at the nodal equations and the partitioning ports. The HB determining equations are solved by numerical method such as Newton-Raphson and/or relaxation methods. Therefore, the solutions are only found in the numerical forms.

We propose here Spice-oriented HB method for obtaining the frequency response curves of nonlinear circuits. Firstly, we have developed *Fourier transformation circuit* executing the Fourier expansions to nonlinear devices. Using the circuit, all kinds of the nonlinear devices such as bipolar transistors and MOSFETs are transformed into the corresponding *HB modules*. Combining with them, a given circuit is transformed into the equivalent HB circuit in the schematic form or net-list [16], [17], which corresponds to the *determining equation of HB method* and can be solved by DC analysis of Spice simulator or solution curve tracing method [18], [19]. Thus, our HB method can continuously calculate the characteristic curves like as Volterra series method.

2. Fourier Transformation Circuit Model

Analog integrated circuits consist of many kinds of nonlinear devices such as diodes, bipolar transistors and MOSFETs, whose Spice models are usually described by the several special functions such as exponential, square-root, piecewise continuous functions and so on [4], [5]. For these device models, the Fourier coefficients of the output waveforms cannot be described by the analytical forms. Therefore, we will introduce Spice-oriented Fourier expansion techniques to these device models. Once we have obtained the HB modules, we can easily carry out the frequency-

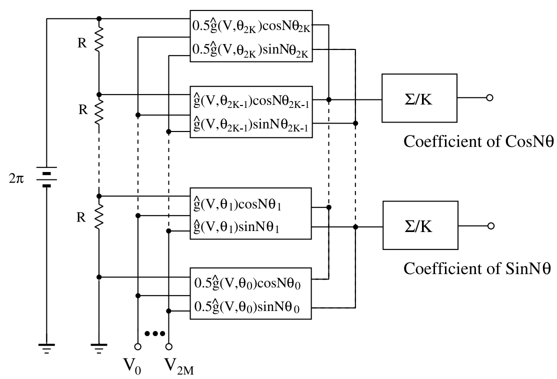

Fig. 1 Fourier transformation circuit model.

domain analysis.

To understand our *Fourier transformation circuit* shown in Fig. 1, we consider one-dimensional Fourier expansion of a nonlinear function given by

$$i = \hat{g}(v). \tag{1}$$

Suppose the input and output waveforms as follows:

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

where M denotes the highest harmonic component to be taken into account in our analysis. The coefficients can be calculated by the following formulae:

$$\left.\begin{aligned} I_0 &= \frac{1}{2\pi}\int_0^{2\pi}\hat{g}(v)\, d(\omega t) \\ I_{2k-1} &= \frac{1}{\pi}\int_0^{2\pi}\hat{g}(v)\cos k\omega t\, d(\omega t), \\ I_{2k} &= \frac{1}{\pi}\int_0^{2\pi}\hat{g}(v)\sin k\omega t\, d(\omega t) \\ k &= 1, 2, \ldots, M \end{aligned}\right\}. \tag{3}$$

The coefficients cannot be given in the analytical forms for the special functions $\hat{g}(v)$. Therefore, we apply the trapezoidal formula to the integration (3) as follows:

$$\begin{aligned} &\int_a^b \hat{g}(v)\, d(\omega t) = \frac{h}{2}(\hat{g}_0 + \hat{g}_{2K}) + h(\hat{g}_1 + \hat{g}_2 + \cdots + \hat{g}_{2K-1}), \\ &\quad \hat{g}_i = \hat{g}(v(t_i)), \quad \text{for } \omega t_i = 0, \pi/K, \cdots, (2K-1)\pi/K, 2\pi \end{aligned} \tag{4}$$

where the step size of the integration is $h = (b-a)/2K (= \pi/K)$ for $2K$ divisions. Then, the truncation error is given by $\hat{g}^{(2)}h^3K/6$, where $^{(2)}$ denotes the second derivative. We assume the input as follows:

$$v(\theta) = V_0 + \sum_{k=1}^{M}(V_{2k-1}\cos k\theta + V_{2k}\sin k\theta), \quad \theta = \omega t \tag{5}$$

then the *Fourier transformation circuit model* for calculating the Nth harmonic component is shown in Fig. 1, where

$$\left.\begin{aligned} I_{2N-1} &= \frac{1}{\pi}\int_0^{2\pi}\hat{g}(v)\cos N\theta\, d\theta = \frac{1}{2K}(\hat{g}_0 + \hat{g}_{2K}) \\ &\quad + \frac{1}{K}(\hat{g}_1\cos N\theta_1 + \hat{g}_2\cos N\theta_2 + \cdots + \hat{g}_{2K-1}\cos N\theta_{2K-1}) \\ I_{2N} &= \frac{1}{\pi}\int_0^{2\pi}\hat{g}(v)\sin N\theta\, d\theta = \frac{1}{K}(\hat{g}_1\sin N\theta_1 + \hat{g}_2\sin N\theta_2 \\ &\quad + \cdots + \hat{g}_{2K-1}\sin N\theta_{2K-1}) \end{aligned}\right\}. \tag{6}$$

Each block in Fig. 1 is realized by the ABMs of Spice [20], and the outputs of this circuit provide the terms I_{2N-1} and I_{2N} in (6), where the interval $[0, 2\pi]$ of the integration is divided by $2K$ equal divisions. The value of $\theta_k = 2\pi k/2K$ is given by the node voltage at the kth resistor in the resistive circuit.

To investigate the accuracy of our Fourier transformation circuit, we calculate the following Fourier expansion:

$$e^{x\cos\theta} = I_0(x) + I_1(x)\cos\theta + I_2(x)\cos 2\theta + \cdots, \tag{7}$$

whose Fourier coefficients $I_N(x), N = 0, 1, 2\ldots$ are given by the *modified Bessel functions* as follows:

$$I_N(x) = \frac{1}{2\pi}\int_{-\pi}^{\pi}e^{x\cos\theta}\cos N\theta\, d\theta, \quad N = 0, 1, 2, \ldots. \tag{8}$$

According to the simulation result for the case of $N = 1$, $x = 10$ and $h = 2\pi/20$, $I_1(10) = 2761$, which is exactly equal to the table of Bessel function [21]. Thus, Fourier transformation circuit model can get the sufficiently accurate solution with $2K = 20$ divisions of the interval $[0, 2\pi]$.

3. HB Modules of Nonlinear Devices

We will derive *HB modules* of the nonlinear devices such as bipolar transistors and MOSFETs using Fourier transformation circuit shown in Fig. 1.

3.1 HB Module of Bipolar Transistor

Let us obtain HB module to Gummel-Poon model of bipolar transistor. We assume the input waveforms of collector, base and emitter (resp. $v_C(t)$, $v_B(t)$ and $v_E(t)$) as follows:

$$\begin{aligned} v_i(t) &= V_{i,0} + \sum_{k=1}^{M}(V_{i,2k-1}\cos k\omega t + V_{i,2k}\sin k\omega t), \\ & i = C, B, E \end{aligned} \tag{9.1}$$

We transform them into the voltage differences of the base-collector and base-emitter

$$v_{BC}(t) = v_B(t) - v_C(t), \quad v_{BE}(t) = v_B(t) - v_E(t). \tag{9.2}$$

Firstly, using the Fourier transformation circuit of Fig. 1, the Fourier coefficients of nonlinear resistive currents $\{i'_B(t),\ i'_C(t)\}$ and charges of parasitic capacitors

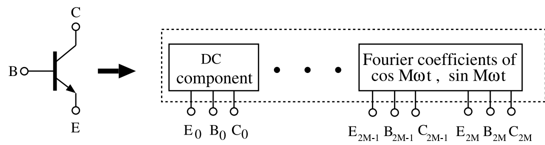

Fig. 2 HB module of bipolar transistor.

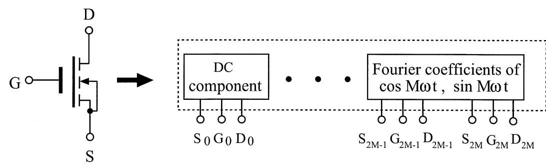

Fig. 3 HB module of MOSFET.

$\{q_{BC}(t),\ q_{BE}(t)\}$ are calculated. Combining these currents and charges, the output currents are estimated from the following relations:

$$\left.\begin{aligned} i_B(t) &= i'_B(t) + \frac{dq_{BE}(t)}{dt} + \frac{dq_{BC}(t)}{dt} \\ i_C(t) &= i'_C(t) - \frac{dq_{BC}(t)}{dt} \end{aligned}\right\}, \tag{9.3}$$

They are also described by the Fourier expansions

$$i_i(t) = I_{i,0} + \sum_{k=1}^{M}(I_{i,2k-1}\cos k\omega t + I_{i,2k}\sin k\omega t), \quad i = C, B, \tag{9.4}$$

where the coefficients are given as follows:

$$\begin{aligned} &I_{B,0} = I'_{B,0}, \quad I_{B,2k-1} = I'_{B,2k-1} + k\omega(Q_{BE,2k} + Q_{BC,2k}), \\ &I_{B,2k} = I'_{B,2k} - k\omega(Q_{BE,2k-1} + Q_{BC,2k-1}) \end{aligned} \tag{9.5}$$

$$\begin{aligned} &I_{C,0} = I'_{C,0}, \quad I_{C,2k-1} = I'_{C,2k-1} - k\omega Q_{BC,2k}, \\ &I_{C,2k} = I'_{C,2k} + k\omega Q_{BC,2k-1} \end{aligned} \tag{9.6}$$

$$k = 1, 2, \ldots, M$$

Thus, we have the HB module for bipolar transistor as shown in Fig. 2. The blocks {"$\cos k\omega t$, $\sin k\omega t$", $k = 1\ldots, M$} correspond to the Fourier coefficients (9.5) and (9.6), which are constructed by ABMs of Spice and the Fourier transformation circuit of Fig. 1. Note that they are coupled resistively and do not contain any dynamical element.

3.2 HB Module of MOSFET

There are many types of MOS models depending on the levels of accuracy [4], [5]. Their HB modules are also built by the same ways as the bipolar transistor. The three terminal HB module is shown in Fig. 3.

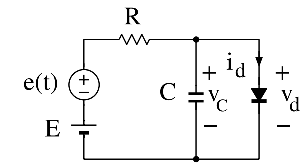

Fig. 4 Diode circuit. ($R = 1$ [kΩ], $C = 1$ [μF], $E = 1$ [V])

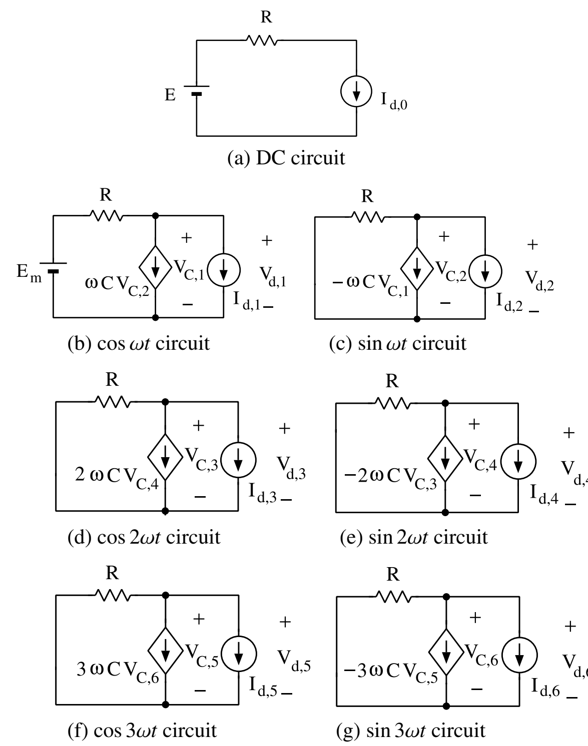

Fig. 5 Equivalent HB circuit of Fig. 4.

4. Illustrative Examples

4.1 Example of HB Method and Comparison with Volterra Series

To understand our HB method, consider a simple diode circuit as shown in Fig. 4. We assume the input voltage and the diode current as follows:

$$e(t) = E_m \cos \omega t, \quad i_d(t) = 10^{-8}\exp(40v_d) \tag{10}$$

Then we set the capacitor and diode voltages as:

$$\left.\begin{aligned} v_C(t) &= V_{C,0} + \sum_{k=1}^{3}(V_{C,2k-1}\cos k\omega t + V_{C,2k}\sin k\omega t) \\ v_d(t) &= V_{d,0} + \sum_{k=1}^{3}(V_{d,2k-1}\cos k\omega t + V_{d,2k}\sin k\omega t) \end{aligned}\right\}, \tag{11}$$

and the Fourier expansion of diode as:

$$i_d(t) = I_{d,0} + \sum_{k=1}^{3}(I_{d,2k-1}\cos k\omega t + I_{d,2k}\sin k\omega t). \tag{12}$$

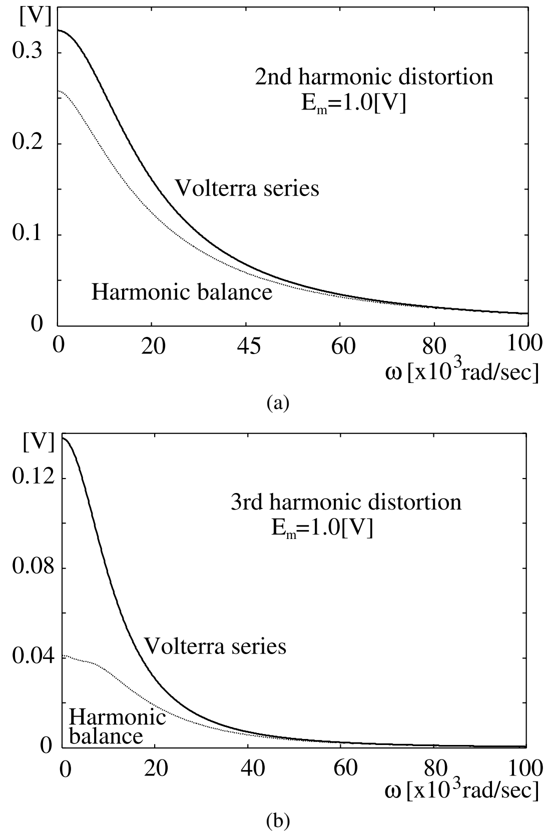

Fig. 6 (a) 2nd and (b) 3rd order harmonic distortion. (E_m = 1.0 [V])

Table 1 Comparison of accuracy. ($\omega = 20 \times 10^3$)

	Volterra series		Harmonic balance		Transient analysis	
E_m	HD2	HD3	HD2	HD3	HD2	HD3
0.1	0.01608	0.00031	0.01602	0.00031	0.01276	0.00184
0.5	0.08044	0.00773	0.07526	0.00679	0.07113	0.00491
0.7	0.11262	0.01515	0.09898	0.01179	0.09803	0.01002
1.0	0.16088	0.03092	0.12436	0.01886	0.12978	0.01873
2.0	0.32176	0.12367	0.14042	0.03438	0.16978	0.03546

We have the equivalent HB circuit as shown in Fig. 5. The nonlinear controlled currents $\{ I_{d,k}, k = 0, 1, \ldots, 6 \}$ are the functions of $\{ V_{d,k}, k = 0, 1, \ldots, 6 \}$, which are obtained by the Fourier transformation circuit of Fig. 1. Note that the equivalent HB circuit of Fig. 5 corresponds to the *determining equation of the HB method*, and can be solved by DC analysis of Spice. We also solved the circuit of Fig. 4 by Volterra series method, and compared both results. The 2nd and 3rd order harmonic distortion curves are shown in Fig. 6.

We also compared them with the results obtained by transient analysis and FFT[†]. Table 1 shows the results for different input voltages. Note that there are slight differences between HB and Volterra methods for small input, but the results are completely different for large input. Although we neglected harmonics larger than 3rd order in our HB method, HB method is more accurate than the Volterra series method.

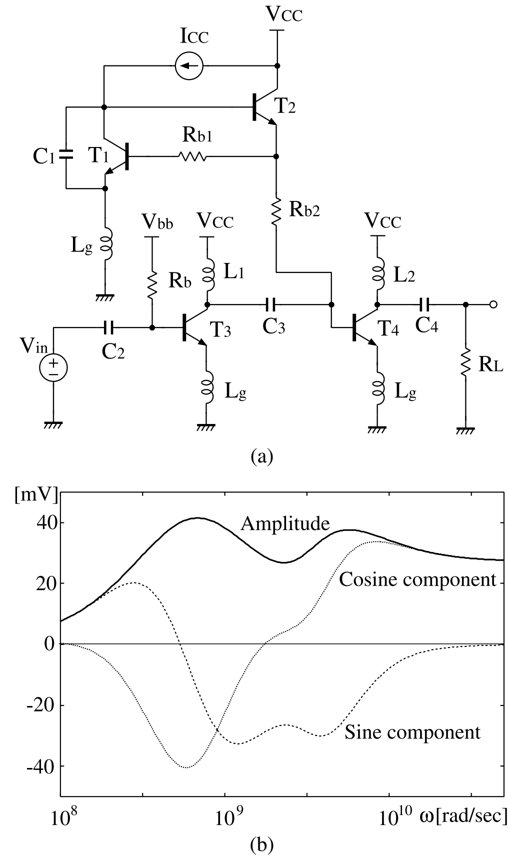

Fig. 7 (a) RF power amplifier, (b) Frequency response curve. $L_1 = L_2 =$ 0.1 [mH], L_g = 10 [nH], $C_1 = C_2 = C_3 = C_4$ = 1 [nF], V_{CC} = 3 [V], V_{bb} = 3 [V], $R_{b1} = R_{b2}$ = 1 [Ω], R_b = 1 [Ω], I_{CC} = 1 [mA], R_L = 36 [Ω], $v_{in} = 0.1 \cos \omega t$.

4.2 RF Power Amplifier

We consider a high frequency RF power amplifier shown in Fig. 7(a), which is used in a mobile-phone [23], where L_g's are parasitic inductances. Firstly, the transistors are modeled by the Gummel-Poon model, whose typical parameters are given in [5]. We considered up to 3rd harmonic components and implemented the equivalent HB circuit with transistor modules of Fig. 3.

We have obtained the frequency response curves for $\omega = 10^8$–10^{11} [rad/sec], as shown in Fig. 7(b), where only fundamental frequency component is shown, because the waveform is nearly sinusoidal. The waveform is sufficiently accurate in comparison to the result of the transient analysis by Spice. It takes 41 [sec] to get the frequency response curve with PSpice[††].

[†]The results may have truncation errors in transient analysis, but they can be assumed to be accurate.

[††]The simulation is carried out by a PC with the following spec; CPU: PentiumM 733 (1.10 GHz), Main memory: 256 MB, OS: Windows XP Professional SP2.

5. Conclusions and Remarks

In this letter, we have proposed Spice-oriented HB (harmonic balance) method for calculating frequency response curves. At first, the nonlinear devices such as bipolar transistors and MOSFETs are transformed into the HB modules. Using these device modules, we transformed the nonlinear circuit into the equivalent HB circuit which corresponds to the determining equations of the HB method. It consists of $(2M + 1)$-coupled sub-circuits when we consider M higher harmonic components, and it can be easily solved with the DC analysis of Spice.

References

[1] M. Schetzen, The Volterra and Wiener Theorems of Nonlinear Systems, John Wiley and Sons, 1978.
[2] J. Wood and D.E. Root, Fundamentals of Nonlinear Behavioral Modeling for RF and Microwave Design, Artech House, 2005.
[3] P. Wambacq and W. Sansen, Distortion Analysis of Analog Integrated Circuits, Kluwer Academic Pub., 1998.
[4] M. Miura, T. Shiino, and K. Mori, Circuit-Simulation Technique and MOSFET Modeling, Riaraizu Rikou Senta, 2003.
[5] A.S. Sedra and K.C. Smith, Microelectronic Circuit, Oxford Univ. Press, 2004.
[6] K.S. Kundert, J.K. White, and A. Sangiovanni-Vincentelli, Steady-State Methods for Simulating Analog and Microwave Circuits, Kluwer Academic Pub., 1990.
[7] A. Ushida, T. Adachi, and L.O. Chua, "Steady-state analysis of nonlinear circuits based on hybrid methods," IEEE Trans. Circuits Syst. I, Fundam. Theory Appl., vol.39, no.9, pp.649–661, 1992.
[8] C. Hayashi, Nonlinear Oscillations in Physical Systems, McGraw-Hill, 1964.
[9] Y. Ueda, The Road to Chaos-II, Aerial Press. Inc., 2001.
[10] R. Telichevesky, K.S. Kundart, and J.K. White, "Efficient steady-state analysis based on matrix-free Krylov-subspace methods," Proc. ACM/IEEE Design Automation Conference, pp.480–485, 1995.
[11] R.J. Gilmore and M.B. Steer, "Nonlinear circuit analysis using the method of harmonic balance: A review of the Art. Part I. Introductory concepts," Int. J. Microw. Millim. Wave Comput.-Aided Eng., vol.1, pp.22–37, 1991.
[12] R.J. Gilmore and M.B. Steer, "Nonlinear circuit analysis using the method of harmonic balance: A review of the Art. Part II. Advanced concepts," Int. J. Microw. Millim. Wave Comput.-Aided Eng., vol.1, pp.159–180, 1991.
[13] H.G. Brachtendorf, G. Welsh, R. Laur, and A. Bunse-Gerstner, "Numerical steady state analysis of electronic circuits driven by multi-tone signals," Electrical Engineering, vol.79, no.2, pp.103–112, Springer-Verlag, 1996.
[14] Y. Yamagami, Y. Nishio, A. Ushida, M. Takahashi, and K. Ogawa, "Analysis of communication circuits based on multidimensional Fourier transformation," IEEE Trans. Comput.-Aided Des. Integr. Circuits Syst., vol.18, no.8, pp.1165–1177, 1999.
[15] S.A. Mass, Nonlinear Microwave Circuits, Artech House, 1988.
[16] A. Ushida, Y. Yamagami, and Y. Nishio, "Frequency responses of nonlinear networks using curve tracing algorithm," ISCAS 2002, vol.I, pp.641–644, 2002.
[17] Y. Yamagami, H. Yabe, Y. Nishio, and A. Ushida, "Distortion analysis of nonlinear networks based on Spice-oriented harmonic balance method," ISCAS04, pp.633–636, 2004.
[18] A. Ushida, Y. Yamagami, Y. Nishio, I. Kinouchi, and Y. Inoue, "An efficient algorithm for finding multiple DC solutions on the Spice-oriented Newton homotopy method," IEEE Trans. Comput.-Aided Des. Integr. Circuits Syst., vol.21, pp.337–348, 2002.
[19] A. Ushida and M. Tanaka, Electronic Circuit Simulation, pp.166–186, Corona Co., 2002.
[20] MicroSim: PSpice A/D Circuit Analysis User's Guides, MicroSim Co., 1995.
[21] K.K. Clarke and D.T. Hess, Communication Circuits: Analysis and Design, Addison-Wesley Pub. Co., 1971.
[22] J.E. Meyer, "MOS models and circuit simulations," RCA Review, vol.32, no.1, pp.42–63, 1971.
[23] D. Leenaerts, J.V. Tang, and C. Vaucher, Circuit Design for RF Transverse, Kluwer Academic Pub., 2001.
[24] R. Gregorian, Introduction to CMOS OP-Amps and Comparators, John Wiley and Sons, 1999.