2005 RISP International Workshop on Nonlinear
Circuit and Signal Processing (NCSP'05)
Hawaii, USA, Mar. 4-6, 2005.

Frequency Response Curves of Nonlinear Electronic Circuits Using Fourier Transformer

Y.Yamagami†, *Y.Nishio*†, *J.Kawata*‡, *A.Ushida*‡
†Department of Electrical and Electronic Engineering,
Tokushima University, Tokushima, 770-8506 JAPAN
‡Department of Mechanical and Electronic Engineering
Tokushima Bunri University, Kagawa, 769-2193 JAPAN

Abstract

Distortion analysis of nonlinear circuits is very important for designing analog integrated circuits. Especially, the frequency response curves are useful to know the global properties of the circuits. Of course, AC-analysis of Spice is used for this purpose, but it can only get the response curves for the linear incremental circuits at the DC operating point. Thus, we need to develop a user-friendly simulator to calculate the response curves for the practical forced inputs. The characteristic curves may be largely different from those of the incremental linear circuits. We propose here an efficient approach based on the Spice-oriented harmonic balance method, where the Fourier transformations of the nonlinear devices such as bipolar transistors and MOSFETs are carried out with our *Fourier transfer circuit model*. Note that the circuit models are constructed by the analog behavior models (ABMs) of Spice. The Fourier circuits corresponding to the determining equations in our harmonic balance method are schematically described by coupled resistive DC, Cosine and Sine circuits. Therefore, the frequency response curves can be easily obtained with a use of DC analysis of Spice.

1. Introduction

The distortion analysis in the frequency domain is very important for designing the nonlinear analog integrated circuits. Especially, the frequency response curves are very useful to know the global behaviors of the circuits. The Volterra series method has been widely used [1-5], because the functions to solve the characteristic curves are obtained in the analytical forms. The idea is based on the bilinear theorem, which can be applied to the weakly nonlinear circuits. Although the algorithm is theoretically elegant, it is not so easy to derive the higher order Volterra kernels needed for the higher order distortion analysis [1]. Furthermore, the nonlinear characteristics of the devices in the method should be described by the polynomial forms which can be calculated by the Taylor expansions in the vicinity of each DC operating point of nonlinear devices [2]. This task is not so easy, especially, for the complicated device models such as the Gummel-Poon model of bipolar transistors and the higher level models of MOSFETs in the high frequency domain[6,7]. On the other hand, there have been proposed many algorithms for calculating the exact steady-state waveforms of nonlinear circuits [8-10]. However, they are inefficient for getting the characteristics such as frequency response curves in wide frequency region.

In this paper, we present a new efficient Spice-oriented harmonic balance method for calculating the frequency response curves at the fundamental and the higher harmonic components in the distortion analysis. Integrated circuits are usually composed of many nonlinear devices such as diodes, bipolar transistors and/or MOSFETs, whose models are described by the different forms depending on the demanded frequency domain. Namely, we need to take into consider the parasitic elements in the higher frequency which make the models very complicate. However, the higher oder models applicable in the wide frequency domain should be used for calculating the frequency response curves. Firstly, we have developed packaged modules of the nonlinear devices using the analog behavior models (ABMs) of Spice [12], which execute the Fourier transformations [11] of the devices. Thus, all the devices are replaced by the corresponding packaged modules. After then, the linear sub-circuit is transformed into the corresponding Cosine and Sine circuits consisted of the resistive elements and controlled sources. Now, we simply call the circuits *harmonic circuits* at corresponding order of the harmonic analysis.

Since we have developed the packaged modules of all kinds of nonlinear devices such as diodes, bipolar transistors and MOSFETs as user's library in my computer, it is easy to get the frequency response curves with DC analysis of Spice. Thus, we have developed the user-friendly simulator for calculating the frequency response curves of nonlinear electronic circuits.

We show the Fourier transfer circuit in section 2, and the packaged models of nonlinear devices in section 3. The interesting illustrative examples are shown in section 4.

2. Fourier transfer circuit model

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 special functions such as the exponential, square-root, piecewise continuous functions and so on [7]. For these devices, it was impossible to execute the Fourier expansions in the analytical forms. Therefore, we have tried to obtain the Fourier expansions in the circuit models with ABMs of Spice [11]. We first show a simulator carrying out the Fourier expansion of nonlinear one port resistive element driven by a periodic input. Simplicity, assume that the nonlinear resistor is a voltage-controlled element as follows:

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

Let us assume the input and the 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 take account in the analysis. Thus, the output Fourier coefficients are described by the function of the input $\mathbf{V}$ as follows:

$$\left.\begin{aligned} I_0 &= f_0(V_0, V_1, \cdots, V_{2M}) \\ I_1 &= f_1(V_0, V_1, \cdots, V_{2M}) \\ &\dots\dots\dots\dots \\ V_{2M} &= f_{2M}(V_0, V_1, \cdots, V_{2M}) \end{aligned}\right\} \tag{3}$$

Note that the Fourier coefficients I_k can be given by the explicit functions of $(V_0, V_1, \ldots, V_{2M})$ only if the nonlinear function $\hat{g}(v)$ is described by the polynomial function. Therefore, we will propose here a *Fourier transfer circuit model* which can be applied to any kind of circuit elements as shown in Fig.1. Each Fourier coefficient for $\hat{g}(v)$ can be obtained by the discrete Fourier transformation as follows:

$$\left.\begin{aligned} & I_0 = \frac{1}{2\pi}\int_0^{2\pi}\hat{g}(v)dt \\ & I_{2k_1} = \frac{1}{\pi}\int_0^{2\pi}\hat{g}(v)\cos k\omega t dt, I_{2k} = \frac{1}{\pi}\int_0^{2\pi}\hat{g}(v)\sin k\omega t dt \\ & k = 1, 2, \ldots, M \end{aligned}\right\} \tag{4}$$

Now, let us apply a trapezoidal integration formula to (4) as follows;

$$\int_a^b \hat{g}(v)dt = \frac{h}{2}(\hat{g}_0 + \hat{g}_n) + h(\hat{g}_1 + \hat{g}_2 + \cdots + \hat{g}_{n-1}) \tag{5}$$

where the step size of the integration is $h = (a-b)/n$. Then, the truncation error is given by $\hat{g}^{(2)}h^2/12n$, where $^{(2)}$ shows the second derivative. Replacing the integrations in (4) by (5), we can realize the equivalent circuit model for the relations (4) with ABMs of Spice. To understand the circuit model, we assume the input

$$v(\theta) = V_0 + \sum_{k=1}^{M}(V_{2k-1}\cos k\theta + V_{2k}\sin k\theta) \tag{6}$$

The Fourier transfer circuit model for calculating the Nth higher harmonic component is shown in Fig.1. Applying integral formula (5) to (4), we have

$$\left.\begin{aligned} & I_{2N-1} = \int_0^{2\pi}\hat{g}(v)\cos N\theta d\theta = \frac{\pi}{2K}(\hat{g}_0 + \hat{g}_{2K}) + \frac{\pi}{K}(\hat{g}_1\cos N\theta_1 \\ & +\hat{g}_2\cos N\theta_2 + \cdots + \hat{g}_{2K-1}\cos N\theta_{2K-1}) \\ & I_{2N} = \int_0^{2\pi}\hat{g}(v)\sin N\theta d\theta = \frac{\pi}{K}(\hat{g}_1\sin N\theta_1 + \hat{g}_2\sin N\theta_2 \\ & + \cdots + \hat{g}_{2K-1}\sin N\theta_{2K-1}) \end{aligned}\right\} \tag{7}$$

The blocks in Fig.1 are constructed by the ABMs of Spice which calculate the each term in the relation (7). In the calculations, the interval $[0, 2\pi]$ of the integration is divided by $2K$ equal divisions, so that the value of $\theta_k = 2\pi/2K$ is obtained as a node voltage at the kth resistor in the resistive circuit.

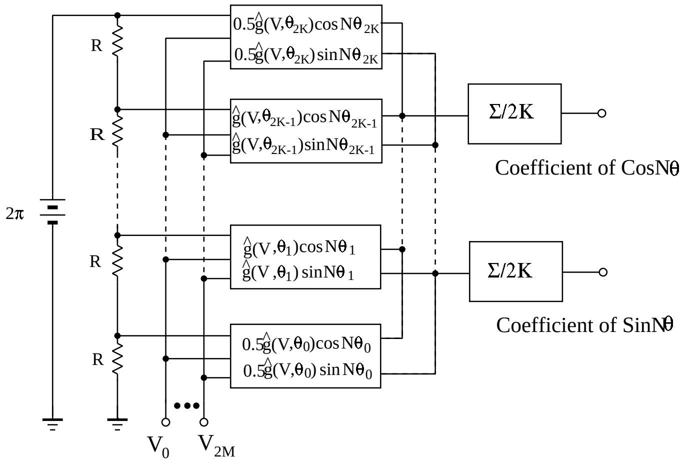

Fig.1 Fourier transfer circuit model.

To investigate the accuracy of our Fourier transfer circuit model, we first calculate the following function:

$$e^{x\cos\theta} = I_0(x) + I_1(x)\cos\theta + I_2(x)\cos 2\theta + \cdots \tag{8}$$

whose Fourier coefficients are given by modified Bessel functions [13] as follows:

$$I_N(x) = \frac{1}{2\pi}\int_{-\pi}^{\pi} e^{x\cos\theta}\cos N\theta d\theta, \quad N = 0, 1, 2, \ldots \tag{9}$$

The simulation result for $h = 2\pi/20$ with Fig.1 is $I_1(10) = 2761$ at $N = 1,\ x = 10$ which is exactly equal to the value from the table of Bessel function [13]. Thus, we found that the Fourier transfer circuit model can get the sufficiently exact solution even with $n = 10$ to 20 divisions of the interval 2π.

Next, we apply it to the Fourier transformation of MOSFET, whose first level in Spice model is described by a piecewise continuous function [6] as follows:

1. Linear region: $(V_{GS} > V_T,\ V_{GS} - V_T \geq V_{DS} > 0)$

$$I_D = \frac{KW}{L}\left[(V_{GS} - V_T) - \frac{V_{DS}}{2}\right]V_{DS}(1 + \lambda V_{DS}) \tag{10.1}$$

2. Saturation region: $(V_{GS} > V_T,\ V_{DS} > V_{GS} - V_T)$

$$I_D = \frac{KW}{L}(V_{GS} - V_T)^2(1 + \lambda V_{DS}) \tag{10.2}$$

The result of Fourier expansions for the input $V_{GS}\cos\omega t$ is shown in Fig.2.. Thus, the Fourier transfer circuit model Fig.1 will be applied to any kind of circuit elements contained in analog integrated circuits.

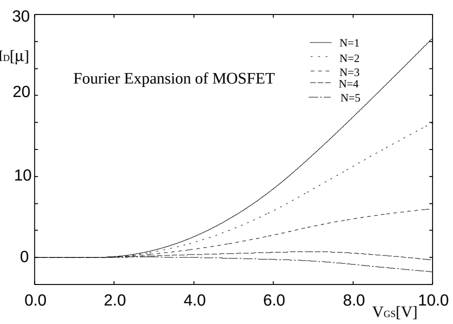

Fig.2 Fourier transformation for MOSFET.

3. Fourier transformation of nonlinear circuits

Now, we consider the distortion analysis of nonlinear electronic circuits. Analog ICs contain many kind of nonlinear devices such as diodes, bipolar transistors and MOSFETs. In this section, we show a technique to replace the devices by the corresponding modules using the Fourier transfer circuit model in section 2. These customizations are very useful to develop user-friendly simulators of our harmonic balance method. In this case, we need to take account of the parasitic capacitors in the modelings at higher frequency domain [6-7]. Assume that the currents and charges of the devices have voltage controlled nonlinear characteristics. The Spice models are given as follows:

(a) Diodes:
In the modelings of diodes at higher frequency domain, the currents i_D are composed of the DC current i_d and charge q_D given as follows:

$$i_d = I_S\left(\exp(\frac{v_d}{nV_t}) - 1\right), \quad V_t = \frac{kT}{q} \tag{11}$$

The charge characteristics are described by the piecewise continuous functions as follows:

$$q_D = \frac{C_{j0}A_n}{[1-(v_d/\phi_B)]^m} + \frac{\tau I_S A_n}{nV_t}\exp\left(\frac{v_d}{nV_t}\right), \quad \text{for} \quad (v_d < F_c\phi_B) \tag{12.1}$$

$$q_D = \frac{C_{j0}A_n}{[1-F_c]^{(1+m)}}\left(1 - F_c(1+m) + \frac{mv_d}{\phi_B}\right) + \frac{\tau I_S A_n}{nV_t}\exp\left(\frac{v_d}{nV_t}\right), \quad \text{for} \quad (v_d \geq F_c\phi_B) \tag{12.2}$$

where the examples of the default values [6] are given by $I_s = 10^{-14}[A]$, $n = 1$, $\tau = 10^{-10}[s]$, $\phi_B = 0.8[V]$, $m = 0.5$, $C_{j0} = 2. \times 10^{-12}[F]$, $F_c = 0.5$, $A_n = 1$. $k = 1.381 \times 10^{-23}[J/K]$, $q = 1.602 \times 10^{-19}[C]$, and T is the device temperature in degrees Kelvin. Thus, $V_t \simeq 0.026[V]$.

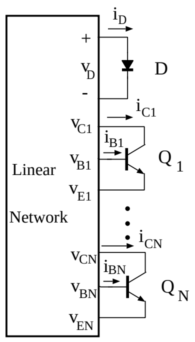

Fig.3 Nonlinear electronic circuit.

(b) Bipolar transistors:
The collector and base currents of bipolar transistors at higher frequency domain also consist of the DC currents and the charges of the base-collector, and the base-emitter. The DC currents are given by Gummel-Poon model as follows:

$$i_C = \frac{I_S}{Q_B}\left(\exp\left(\frac{v_{BE}}{V_t}\right) - 1\right) - I_S\left(\frac{1}{Q_B} + \frac{1}{B_R}\right)\left(\exp\left(\frac{v_{BC}}{V_t}\right) - 1\right) \tag{13.1}$$

$$i_B = \frac{I_S}{Q_F}\left(\exp\left(\frac{v_{BE}}{V_t}\right) - 1\right) + \frac{I_S}{Q_R}\left(\exp\left(\frac{v_{BC}}{V_t}\right) - 1\right) \tag{13.2}$$

where

$$\frac{1}{Q_B} \simeq 1 - \frac{v_{BC}}{V_{AF}} - \frac{v_{BE}}{V_{AF}}$$

The two capacitor charges are the same as (12). An examples of the default values other than diode are shown in section 4 [6].

(c) MOSFETs
Nowadays, the sizes of MOSFETs are becoming smaller and smaller, so that there are many types of the models [6,7]. The simplest Shichman-Hodges model (level 1) is given in (10). We also need to consider four parasitic capacitor charges q_{DS} drain-gate, q_{GS} source-gate, q_{BS} substrate-source and q_{DB} drain-substrate at the higher frequency domain. Their charge characteristics are given by (12) in the simplest model [6,7].

Now, let us consider Fourier circuit models of the above devices. In oder to develop the used-friendly simulators, we make the Fourier transfer circuit models for all of the devices in the forms of modules. At first, we consider the Fourier transfer model of a bipolar transistor given by (12) and (13). Assume they are driven from 3 terminal voltage as follows:

$$\left.\begin{aligned} v_C &= V_{C,0} + \sum_{k=1}^{M}(V_{C,2k-1}\cos k\theta + V_{C,2k}\sin k\theta) \\ v_B &= V_{B,0} + \sum_{k=1}^{M}(V_{B,2k-1}\cos k\theta + V_{B,2k}\sin k\theta) \\ v_E &= V_{E,0} + \sum_{k=1}^{M}(V_{E,2k-1}\cos k\theta + V_{E,2k}\sin k\theta) \\ &\theta = \omega t \end{aligned}\right\} \tag{14}$$

We calculate the currents $(i_C(\theta_i),\ i_B(\theta_i), |\ \theta_i = (2\pi/K)i, i = 0, 12, \ldots, K)$ and the charges $(q_{BC}(\theta_i),\ q_{BE}(\theta_i))$ by (13) and (12) with $\theta_i = i \times (2\pi/K)$. Thus, using Fourier transfer circuit shown in Fig.1, we can get the Fourier coefficients described by

$$\left.\begin{aligned} i_C &= I_{C,0} + \sum_{k=1}^{M}(I_{C,2k-1}\cos k\theta + V_{I,2k}\sin k\theta) \\ i_B &= I_{B,0} + \sum_{k=1}^{M}(I_{B,2k-1}\cos k\theta + I_{B,2k}\sin k\theta) \end{aligned}\right\} \tag{15.1}$$

$$\left.\begin{aligned} q_{BC} &= Q_{BC,0} + \sum_{k=1}^{M}(Q_{BC,2k-1}\cos k\theta + Q_{BC,2k}\sin k\theta) \\ q_{BE} &= Q_{BE,0} + \sum_{k=1}^{M}(Q_{BE,2k-1}\cos k\theta + Q_{BE,2k}\sin k\theta) \end{aligned}\right\} \tag{15.2}$$

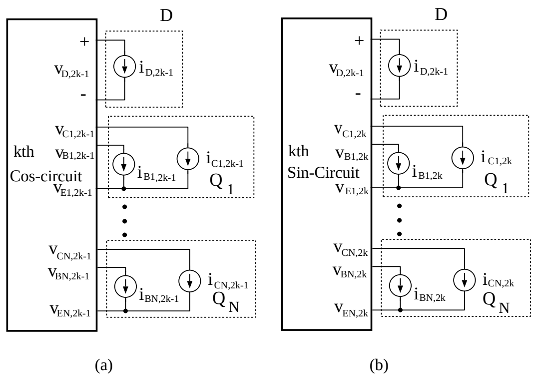

Fig.4 Fourier circuit of Fig. 3 at kth higher harmonic component.

Next, the charges (15.2) need to be transformed into the corresponding currents as follows:

$$\left.\begin{aligned} i_{BC} &= \sum_{k=1}^{M}(-kQ_{BC,2k-1}\sin k\theta + kQ_{BC,2k}\cos k\theta) \\ i_{BE} &= \sum_{k=1}^{M}(-kQ_{BE,2k-1}\sin k\theta + kQ_{BE,2k}\cos k\theta) \end{aligned}\right\} \tag{16}$$

Thus, the Fourier coefficients of the collector and base currents are given by

$$\left.\begin{aligned} i_C &= I_{C,0} + \sum_{k=1}^{M}(I_{C,2k-1} - kQ_{BC,2k})\cos k\theta \\ &\quad +(I_{C,2k} + kQ_{BC,2k-1})\sin k\theta] \\ i_B &= I_{B_0} + \sum_{k=1}^{M}[(I_{B,2k-1} + kQ_{BC,2k} + kQ_{BE,2k})\cos k\theta \\ &\quad +(I_{B,2k} - kQ_{BC,2k-1} - kQ_{BE,2k-1})\sin k\theta] \end{aligned}\right\} \tag{17}$$

Note that the transistor modules are only constructed by the AMBs of Spice, so that it does not contain any dynamic element. This technique is easily applied to the diodes and MOSFETs.

On the other hand, the linear sub-network in Fig.3 is transform to the Cosine and Sine circuits [11], whose the Fourier circuit for kth higher harmonic component is shown in Fig 4. Thus, the Fourier circuit corresponding to the determining equation of the harmonic balance method can be easily calculated by the use of DC analysis of Spice.

4. An illustrative example

Let us calculate the frequency response curves of a tuned-amplifier circuit [13] shown in Fig. 5, where we take account the DC, and the fundamental to the third harmonic components.

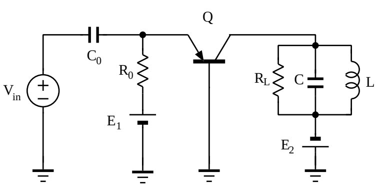

Fig.5 Tuned- amplifier.

$v_{in}(t) = 0.05\cos\omega t, \quad E_1 = E_2 = 10[V], \quad C_0 = 1[\mu F],$
$C = 50[pf], \quad L = 20[\mu H], \quad R_0 = 2[k\Omega], \quad R_L = 1.5[k\Omega]$

The transistor is modeled by a Gummel-Poon model, whose parameters are given by

$I_S = 10^{-14}[A]$	$\tau = 10^{-10}[sec]$	$V_t = 0.026[V]$	$\phi_B = 0.75$
$B_F = 470$	$C_{j0} = 0.1[pF]$	$V_{AF} = 150[V]$	$F_c = 0.74$
$B_R = 1$			

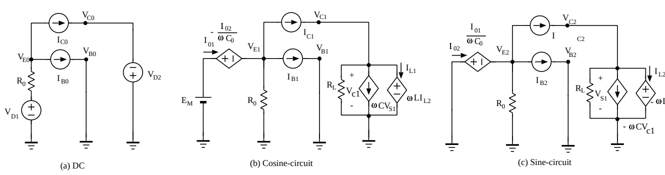

Fig.6 Fourier circuit of the tuned-amplitude.

The Fourier circuits are shown in Fig.6, where the angular frequency is changed in the following manner:

$$\omega = 10^4 \times 10^t$$

To get the DC solutions stably, we set two DC sources as follows:

$V_{D1} = V_{D2} = 10t$, for $0 < t < 1$, $V_{D1} = V_{D2} = 10[V]$, for $t > t$

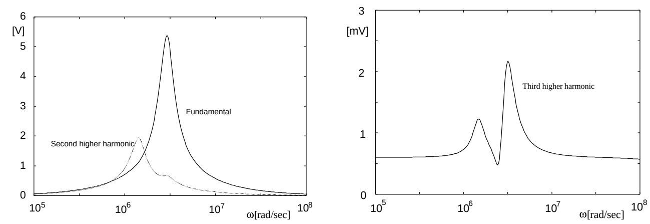

Fig.7 The frequency response curves.

5. Conclusions and remarks

The distortion analysis is very important for designing the high frequency analog integrated circuits. Especially, it is useful to investigate the properties of the frequency response curves in the wide domain.

In this paper, we have proposed an efficient technique for calculating the frequency response curves of nonlinear electronic circuits. At first, the nonlinear devices such as bipolar transistors and MOSFETs are transformed into the modules which execute the Fourier transformations. Using the modules, the nonlinear circuit is transformed into the Fourier transfer circuit which is corresponding to the determining equations of the harmonic balance method. Once, we have developed the modules for all kinds of nonlinear devices, it can be easily applied any kind of circuits. The Fourier circuits are solved with the DC analysis of Spice. Thus, our simulator is quite user-friendly.

As future works, we are going to develop frequency analyzers with two independent input frequencies where we adopt two dimensional Fourier transformation technique. It will be applied to analysis of mixers and modulators.

References

[1] M.Schetzen, *The Volterra and Wiener Theorems of Nonlinear Systems,* John Wiley and Sons, 1978.

[2] 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.

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

[4] B.J.Leon and D.J.Schaefer, "Volterra series and Picard iteration for nonlinear circuits and systems," *IEEE Trans. on Circuits and Systems,* vol.CAS-25, no.9, pp.789-793, 1078.

[5] 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.

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

[7] M.Miura, T.Shiino and K.Mori, *Circuit-Simulation Technique and MOSFET modeling*, Riaraizu Rikou Senta (Japanese), 2003

[8] T.J.Aprille and T.N.Trick, "Steady-state analysis of nonlinear circuits with periodic input," *Proc. IEEE,* vol.60, pp.108-114, 1972.

[9] K.S.Kundert, J.K.White and A.Sangiovanni-Vincentelli, *Steady-State Methods for Simulating Analog and Microwave Circuits,* Kluwer Academic Pub., 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 Systems-I: Fundamental Theory and Applications,* vol.39, no.9, pp.649-661, 1992.

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

[12] MicroSim: PSpice A/D Circuit Analysis User's Guides, MicroSim Co. 1995.

[13] K.K.Clarke and D.T.Hess, *Communication Circuits: Analysis and Design,* Addison-Wesley Pub. Co., 1971.