Distortion Analysis of Nonlinear Networks Based on SPICE-Oriented Harmonic Balance Method

Y. Yamagami†, *H. Yabe*†, *Y. Nishio*†, *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 in the frequency domain is very important for designing the analog integrated circuits. We propose here a new method based on the SPICE-oriented harmonic balance method, where the Fourier transformations to the bipolar transistors and/or MOSFETs are carried out using *new Fourier transfer circuit model*. The circuit is composed of the analog behavior models(ABM) of SPICE, which can be applied to any kind of nonlinear circuit elements described by the exponential and/or piecewise linear functions. Furthermore, the determining equation of our harmonic balance method is schematically described by coupled resistive DC, Cosine and Sine circuits, so that we can easily obtain the frequency response curves with the DC analysis of SPICE. Thus, our approach is quite user-friendly, because we need not derive the circuit equation and the determining equations in our harmonic balance method.

1. Introduction

The distortion analysis in the frequency domain is very important for designing the nonlinear analog integrated circuits. The Volterra series methods are widely used for the analysis of weakly nonlinear systems [1-3]. Although the algorithm is theoretically elegant, it is not so easy to derive the Volterra kernels in the higher order distortions analysis [4]. Furthermore, their nonlinear characteristic in the method must be described in the form of polynomial, so that they can be obtained by the Taylor expansion in the vicinity of the DC operating point of a nonlinear elements [2]. This task is also not so easy especially for the complicated circuit models such as the high frequency Gummel-Poon model of bipolar transistors and/or Shichman-Hodges model of MOSFETs [5]. On the other hand, there have been proposed many algorithms for calculating the exact steady-state waveforms of nonlinear circuits driven by the periodic and/or multi-tone signals [6-8]. Unfortunately, they are inefficient for calculating the nonlinear frequency response curves.

In this paper, we present a new harmonic balance method for calculating the frequency response curves in the distortion analysis of nonlinear integrated circuits. The determining equations can be schematically described by the equivalent DC, Cosine and Sine circuits, where *Fourier transfer circuit model* can be efficiently used for the Fourier expansion of the nonlinear characteristics. The circuit is composed of the resistive analog behavior models (ABM) of SPICE so that the frequency response curves can be obtained by the application of curve tracing algorithm [12]. Note that the Fourier expansion can be efficiently applied to any kind of circuit elements such as bipolar transistors, MOSFETs and so on. Remark that the method in the reference [9] can be only applied to the circuit whose elements are described by the polynomial functions. The Fourier circuit model is shown in section 2, and the equivalent DC, Cosine and Sine circuits are shown in section 3. Each circuit topology is the same as the original one. Of course, the technique can be easily extended to the higher harmonic distortion analyses. We show an interesting illustrative example in section 4. Since our method needs not to derive both the circuit equation and the determining equations, it is really user-friendly algorithm for calculating nonlinear response curves.

2. Fourier transfer circuit model

Analog integrated circuits are usually composed of many kinds of nonlinear elements such as diodes, bipolar transistors and MOSFETs, whose models are described by the special functions such as the exponential, square-root, piecewise continuous functions and so on [5]. Let us discuss how to calculate the frequency response curves. In this case, we first need the Fourier expansion to the nonlinear elements driven by the periodic input.

Let us assume the input and the output waveforms as follows;

$$\left.\begin{aligned} x(t) &= A_0 + \sum_{k=1}^{M}(A_{2k-1}\cos k\omega t + A_{2k}\sin k\omega t) \\ y(t) &= B_0 + \sum_{k=1}^{M}(B_{2k-1}\cos k\omega t + B_{2k}\sin k\omega t) \end{aligned}\right\} \quad (1)$$

where M denotes the highest harmonic component to take account in the analysis. Thus, the output Fourier coefficients need to be described by the functions of input amplitudes as follows;

$$\left.\begin{aligned} B_0 &= g_0(A_0, A_1, \cdots, A_{2M}) \\ B_1 &= g_1(A_0, A_1, \cdots, A_{2M}) \\ &\cdots\cdots\cdots\cdots \\ B_{2M} &= g_{2M}(A_0, A_1, \cdots, A_{2M}) \end{aligned}\right\} \quad (2)$$

Note that the Fourier coefficients B_k can be given by the explicit functions of $(A_0, A_1, \ldots, A_{2M})$, only when the nonlinear function is described by the polynomial function [11]. Therefore, we propose here a new technique using *Fourier transfer circuit model* as shown in Fig.1 and 2.

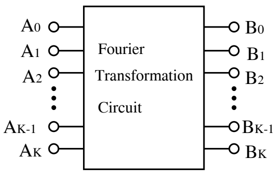

Fig.1 Block diagram realizing the relation (2).

Each Fourier coefficient for the function $f(x)$ can be numerically obtained in the following formula:

$$\left.\begin{array}{l} B_0 = \dfrac{1}{2\pi}\displaystyle\int_0^{2\pi} f(x)dt \\ B_{2k-1} = \dfrac{1}{\pi}\displaystyle\int_0^{2\pi} f(x)\cos k\omega t dt, B_{2k} = \dfrac{1}{\pi}\displaystyle\int_0^{2\pi} f(x)\sin k\omega t dt \\ k = 1, 2, \ldots, M \end{array}\right\} \quad (3)$$

Let us apply the trapezoidal integration formula to (3) as follows;

$$\int_a^b f(x)dx = \frac{h}{2}(f_0 + f_n) + h(f_1 + f_2 + \cdots + f_{n-1}) \quad (4)$$

where the step-size of the integration is $h = (a-b)/n$. Then, the truncation error is given by $f^{(2)}h^2/12n$. Using this formula, we will realize the equivalent circuit model satisfying (3) with ABMs of SPICE. To understand the operation, we simply assume that the input is $x = A\sin\theta$. The Fourier transfer circuit model shown by Fig.2 is composed by ABM blocks, whose operations such as multiplications, exponential, trigonometric functions and so on can be done by the functions written by the Fortran languages. On the other hand, the integration interval $[0, 2\pi]$ is divided by n sections using a number of n resistors, so that each input node voltage of the ABM block is given by $\theta_k = 2\pi k/n$ at kth node. Thus, the resultant current sources are given by $f(A, \theta_k)\cos N\theta_k, f(A, \theta_k)\sin N\theta_k, k = 1, 2, \ldots, n$. Summing them, the outputs correspond to the coefficient of $\cos N\theta$ and $\sin N\theta$ (13). In this case, if the input contains the higher harmonic components, we need to set the number of m terminals $(A_1, A_2, \ldots, A_m)$ instead of the single terminal A in Fig.2.

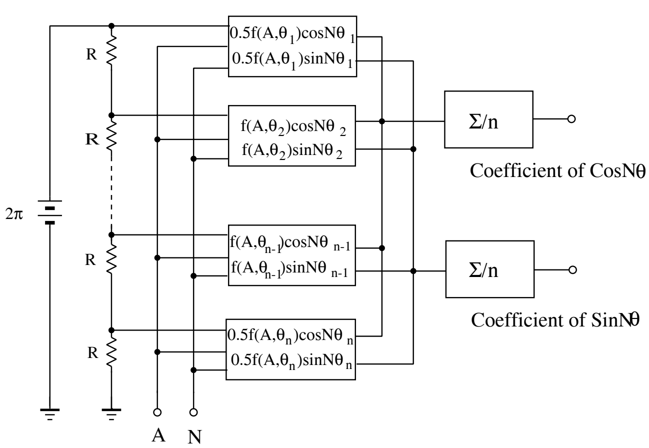

Fig.2 Fourier transfer circuit model.

The fact that the circuit model is a resistive circuit which is important for the calculation of the frequency response curves with curve tracing algorithm [12].

To investigate the numerical accuracy, we first calculate a modified Bessel function as follows;

$$I_N(x) = \frac{1}{2\pi}\int_{-\pi}^{\pi} e^{x\cos\theta}\cos N\theta d\theta \quad (5)$$

The simulation results with $h = 2\pi/20$ is shown in Fig.3. The value $I_1(10) = 2761$ at $N = 1$, $x = 10$ is exactly equal to that from the Table of Bessel function [10]. Note that this kind of Fourier expansion to the exponential function is very important to the analysis of the circuit containing diodes and bipolar transistors [5].

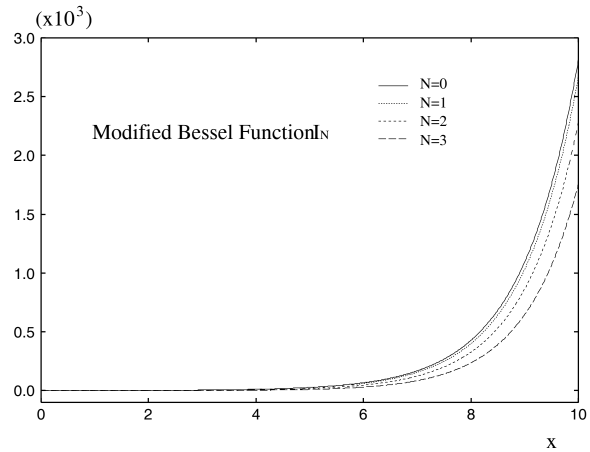

Fig.3 Fourier transformation for modified Bessel function.

Next, we apply it to the Fourier expansion of MOSFET, whose characteristic in SPICE model is described by a piecewise continuous function [5] as follows:

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

$$I_D = \frac{KW}{L}\left[(V_{GS} - V_T) - \frac{V_{DS}}{2}\right]V_{DS}(1 + \lambda V_{DS}) \quad (6.1)$$

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

$$I_D = \frac{KW}{L}(V_{GS} - V_T)^2(1 + \lambda V_{DS}) \quad (6.2)$$

The result of Fourier expansions for the input $V_{GS}\cos\omega t$ is shown in Fig.4.

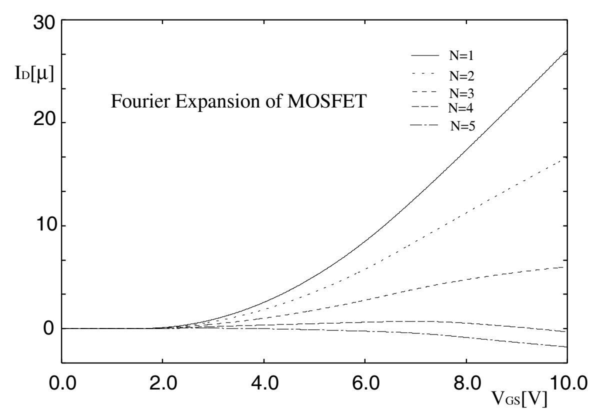

Fig.4 Fourier transformation for MOSFET.
$V_{DS} = 3[V],\ K = 3.87[\mu A],\ \lambda = 0.01605,$
$W = 2[\mu m],\ L = 2[\mu],\ V_T = 0.827[V]$

Thus, the Fourier transfer circuit model shown in Fig.2 can be efficiently applied to any kind of circuit elements contained in analog integrated circuits.

3. Frequency response curve of nonlinear circuits

Nowadays, the frequency response curves of the fundamental
frequency (H_1), the second harmonic distortion(HD_2) and the third harmonic distortion(HD_3) are usually calculated by the use of the Volterra series method [3], where the nonlinear characteristics must be described by the polynomial functions. On the other hand, our distortion analysis using the Fourier transfer circuit model can be efficiently applied to the circuits containing any kind of the elements described by exponential, piecewise linear functions and others.

To understand our harmonic balance method, we consider the following circuit equation;

$$\mathbf{f}(\dot{\mathbf{v}}, \mathbf{v}, \mathbf{w}, \omega t) = \mathbf{0}, \quad \mathbf{f}: R^{2n+m} \mapsto R^{n+m} \tag{7}$$

Although the steady-state waveforms may contain many higher harmonic components, for simplicity, we consider only the DC and fundamental frequency components as follows;

$$\left.\begin{array}{l} \mathbf{v}(t) = \mathbf{V}_0 + \mathbf{V}_1 \cos\omega t + \mathbf{V}_2 \sin\omega t \\ \mathbf{w}(t) = \mathbf{W}_0 + \mathbf{W}_1 \cos\omega t + \mathbf{W}_2 \sin\omega t \end{array}\right\} \tag{8}$$

Substituting (8) into (7) and applying the harmonic balance method, we have the following determining equations;

$$\left.\begin{array}{l} \mathbf{F}_0(\mathbf{V}_0, \mathbf{V}_1, \mathbf{V}_2, \mathbf{W}_0, \mathbf{W}_1, \mathbf{W}_2, \omega) = \mathbf{0} \cdots \mathrm{DC} \\ \mathbf{F}_c(\mathbf{V}_0, \mathbf{V}_1, \mathbf{V}_2, \mathbf{W}_0, \mathbf{W}_1, \mathbf{W}_2, \omega) = \mathbf{0} \cdots \cos\omega t \\ \mathbf{F}_s(\mathbf{V}_0, \mathbf{V}_1, \mathbf{V}_2, \mathbf{W}_0, \mathbf{W}_1, \mathbf{W}_2, \omega) = \mathbf{0} \cdots \sin\omega t \end{array}\right\} \tag{9}$$

In generally, the derivation of (9) from the circuit equation, and the analysis of the determining equations are troublesome. We propose a SPICE-oriented algorithm for getting the DC-circuit, Cosine-circuit and Sine-circuit corresponding to the above the relations (9).

a. **Inductive elements**: Assume that the nonlinear inductor is described by current-controlled characteristic as follows;

$$\phi_L = \hat{\phi}_L(i_L) \tag{10.1}$$

For the current $i_L = I_{L0} + I_{L1}\cos\omega t + I_{L2}\sin\omega t$, we have

$$v_L(t) \simeq -\omega\Phi_{L1}\sin\omega t + \Phi_{L2}\cos\omega t \tag{10.2}$$

where Φ_{L1} and Φ_{L2} are function of $\{I_{L0}, I_{L1}, I_{L2}\}$. They are calculated by the Fourier transfer circuit shown by Fig.2.

For the linear inductor L, the coefficient are simply given by the linear current-controlled voltage sources $\omega L I_{L2}$ in the Cosine-circuit and $-\omega L I_{L1}$ in the Sine-circuit, respectively [9].

b. **Capacitive elements**: High frequency bipolar transistor and MOFET models [5] usually contain nonlinear depletion and diffusion capacitors which are described by the voltage-controlled characteristics as follow;

$$q_C = \hat{q}_C(v_C) \tag{11.1}$$

For the voltage $v_C = V_{C0} + V_{C1}\cos\omega t + V_{C2}\sin\omega t$, we have

$$i_C(t) \simeq -\omega Q_{C1}\sin\omega t + \omega Q_{C2}\cos\omega t \tag{11.2}$$

where Q_{C1} and Q_{C2} are function of $\{V_{L0}, V_{L1}, V_{L2}\}$, and they are also estimated by the Fourier transfer circuit shown in Fig.2.

For the linear capacitor C, the Fourier coefficients are given by the linear voltage-controlled current sources $\omega C V_{V2}$ in the Cosine-circuit and $-\omega C V_{C1}$ in the Sine-circuit, respectively [9].

c. **Resistive elements**: Many of integrated circuit elements are described by the nonlinear voltage-controlled current sources as follows;

$$i_G = \hat{i}_G(v_G) \tag{12.1}$$

For the voltage $v_G = V_{G0} + V_{G1}\cos\omega t + V_{G2}\sin\omega t$, we have

$$i_G(t) \simeq I_{G0} + I_{G1}\cos\omega t + I_{G2}\sin\omega t \tag{12.2}$$

where I_{G0}, I_{G1} and I_{G2} are described by the function of $\{V_{G0}, V_{G1}, V_{G2}\}$. They are estimated by the use of the Fourier transfer circuit model shown by Fig.2. For linear resistors, we need not any transformation [9].

Note that the inductive elements in the DC-circuit are removed by the shorted-circuits, and the capacitive elements by the opened-circuits.

Thus, the circuit topologies corresponding to the DC, Cosine and Sine circuits and their higher harmonic components are equal to the original circuit, and they are coupled in each other with the controlled sources.

4. An illustrative example

Now, consider distortion analysis of a simple high frequency amplifier [3] shown by Fig.5(a). The Ebers-Moll of transistor [5] is shown by Fig.5(b), where the diode models are given by

$$i_{D1} = 10^{-14}\{\exp(40v_{be} - 1)\}[A],$$
$$i_{D2} = 10^{-14}\{\exp(40v_{bc} - 1)\}[A]$$

and

$$R_s = 10[k\Omega], \quad R_L = 10[k\Omega], \quad V_s = 800[mV]$$
$$\alpha_F = 0.99, \qquad \alpha_R = 0.3$$

$$C_1 = 10{+}10v_{bc}^3[pC], \quad C_2 = 10{+}10v_{be}^3[pC], \quad v_{in} = V_{m,in}\cos\omega t$$

For calculating the frequency response of the fundamental, the second order distortion (HD_2) and the third order distortion(HD_3), we assumed the waveforms as follows;

$$v_{be} = V_{be,0} + \sum_{k=1}^{3}(V_{be,2k-1}\cos k\omega t + V_{be,2k}\sin k\omega t)$$
$$v_{bc} = V_{bc,0} + \sum_{k=1}^{3}(V_{bc,2k-1}\cos k\omega t + V_{bc,2k}\sin k\omega t)$$

Applying the harmonic balance method, we obtained the DC, Cosine and Sine circuit as shown in Fig.5 (c),(d) and (e), respectively. The circuit models for calculating the higher order distortions are obtained in the same manner, where

I_{C0}, I_{E0} are the DC voltage-controlled current sources, and the voltage-controlled current sources ($I_{C,k}, I_{E,k}, k = 1, 2, 3$) are given by the function of the input frequency ω. These controlled sources can be calculated by the use of Fourier transfer circuit model shown in Fig.2.

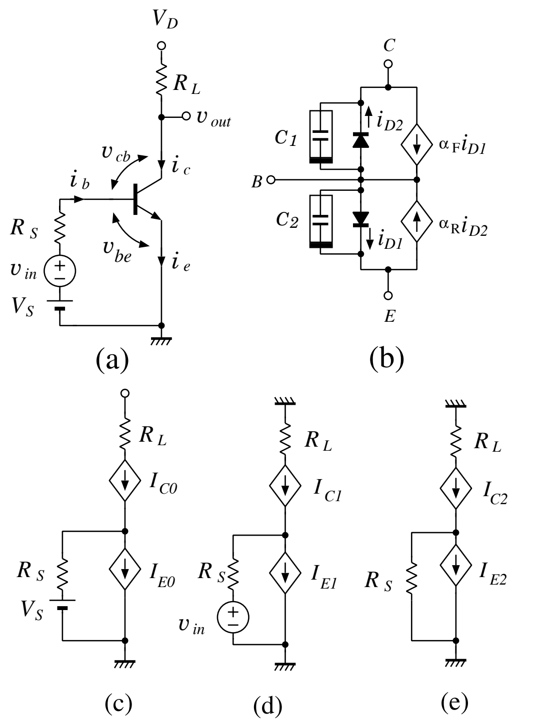

Fig.5 (a) Amplifier circuit, (b) High frequency Ebers-Moll model, (c) DC-circuit, (d) Cosine-circuit, (e) Sine-circuit

Continuously changing the input frequency with the DC analysis of SPICE, we can obtain the frequency response curves for the distortion analysis as shown in Fig.6, which is almost same as those of reference [3].

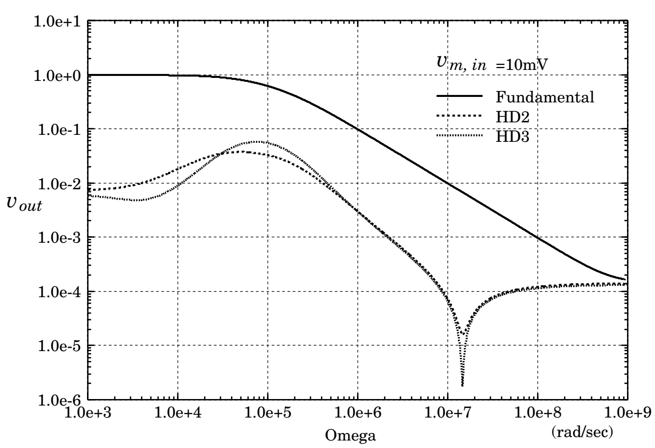

Fig.6 Distortion analysis.

5. Conclusions and remarks

The distortion analysis in frequency domain is very important for designing the high frequency analog integrated circuits. It is usually carried out by the Volterra series method, where the nonlinear characteristics must be described by the polynomial function in the vicinity of the DC operating point. Thus, the method can be only applied to weakly nonlinear circuits. Furthermore, the higher order kernels of the Volterra series are complicate, so that it is usually restricted to the analysis of relatively low order distortion analysis.

In this paper, we have proposed an efficient SPICE-oriented distortion analysis method based on the harmonic balance method, where we uses Fourier transfer circuit model. The model can be efficiently applied to any kind of nonlinear elements described by such as exponential and piecewise continuous functions. We also proposed the DC, Cosine and Sine circuits corresponding to the determining equation of the harmonic balance method, whose circuit topologies are equal to the original circuit. Thus, the algorithm of our distortion analysis is quite simple and user-friendly. We are going to apply the algorithm to the complicated circuits such as the 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] 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.

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

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

[7] K.S.Kundert, J.K.White and A.Sangiovanni-Vincentelli, *Steady-State Methods for Simulating Analog and Microwave Circuits,* Kluwer Academic Pub., 1990.

[8] 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.

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

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

[11] S.Wolfram, *Mathematica,* Addison-Wesley Pub. Co., 1988

[12] A.Ushida, Y.Yamagami, Y.Nishio, I.Kinouchi and Y.Inoue, "An efficient algorithm for finding multiple DC solutions based on the SPICE-oriented Newton homotopy method," *IEEE Trans. on CAD.* vol.21, no.3, pp.337-348, 2002.