The 15th IEEE International Workshop on Nonlinear Dynamics of Electronic Systems
NDES'07, Tokushima, Japan, July 23-26, 2007

Comparisons Between Spice-Oriented Harmonic Balance Method and Volterra Series Method

Junji Kawata†, Takaaki Kinouchi‡, Yoshihiro Yamagami‡, Yoshifumi Nishio‡, Akio Ushida†
†Department of Mechanical and Electronic Engineering,
Tokushima Bunri University, Kagawa, 769-2193 JAPAN
‡Department of Electrical and Electronic Engineering,
Tokushima University, Tokushima, 770-8506 JAPAN

Abstract

It is very important to analyze the mixer and modulator circuits driven by multiple inputs frequencies. Both Volterra series and HB (harmonic balance) methods are widely used for the frequency-domain analysis of nonlinear circuits. We compared the errors of the above two methods using illustrative examples. In our HB method, the Fourier expansion is executed by MATLAB in symbolic form, and it is transformed into the net-list with Spice. We found that it is sometime happened to exceed the capacities of Spice, when we choose higher order polynomial approximations of the nonlinear devises, because the Fourier components are rapidly increased. Hence, we propose here a *new perturbation like Fourier expansion method* to the approximations.

1. Introduction

Frequency-domain analysis of nonlinear electronic circuits driven by multiple inputs are very important for designing RF circuits and communication systems. Volterra series methods are widely used for the analysis [1-3], because they give the solutions in analytical forms. There have been published many papers about computer-aided HB(harmonic balance) methods [4-6] during 1990-2000 which are using DFT, FFT and/or interpolation methods for the Fourier expansion. They apply the Fourier expansions to all the nodal equations of the circuit, so that the resultant systems become usually very large. Hence, they propose to solve them with the applications of a method like the Krylov-subspace GMRES algorithm instead of Newton method which will reduce the computation time. It seems that their methods are still time-consuming for large scale circuits.

On the other hand, the classical HB methods [7-8] can be applied to relatively small scale circuits, which find many interesting nonlinear phenomena such as bifurcations and chaos phenomena. Now, we are going to apply it to electronic circuits containing transistors and diodes. These devices have strong nonlinear characteristics and they are modeled by special functions such as exponential in Spice. In order to apply the HB method, their characteristics must be approximated by the polynomial functions. In this way, the Fourier expansions of the nonlinear devices are stored as modules in the net-lists of Spice with MATLAB. In this case, we have sometime happened a serious problem such that when we apply high order polynomial approximations to the devices, the Fourier expansion contains too many terms and it exceeds the capacity of Spice. So, we propose here *a new perturbation like Fourier expansion method* using MATLAB which reduces the resultant terms and the size of Spice netlist. The ideas are shown in section 2.

Using the above Fourier expansion method, nonlinear devices are replaced by HB modules as follows: Firstly, bipolar transistors and MOSFET are approximated by the Taylor expansions in the polynomial forms. The Fourier expansions can be executed by MATLAB in the symbolic forms and they are stored as the modules of bipolar transistor in our Spice library. Thus, it is possible to formulate the determining equation of HB in the forms of net-list with Spice. It can be easily solved by the DC analysis of Spice and the frequency-domain solutions are easily obtained.

Using an example, we will compare the results of Volttera series method and our HB method in section 2. In section 3, we show an interesting example of a mixer circuit.

2. Fourier expansion of nonlinear devices

2.1 Taylor approximation of nonlinear devices In this sub-section, we study the errors of HB method when we applied the Fourier expansion to the Taylor approximation of an exponential function. Transistors are usually modeled by Ebers-Moll model consisted of two diodes, whose nonlinear characteristic (i_d, v_d) is described by exponential function as follows;

$$i_d = I_s \exp v_d/V_T, \quad V_T \simeq 0.025. \tag{1}$$

Firstly, let us calculate the Fourier coefficients DC, $\cos\omega t$, $\cos 2\omega t$, $\cos 3\omega t$ of (1) driven by

$$v_d = V_{d0} + V_d \cos\omega t, \quad \text{for} \quad V_{d0} = 0.5822, \tag{2}$$

We also compare them with the results when we applied it to the 3rd order Taylor approximation as follows:

$$i_d = k_0 + k_1 v_d + k_2 v_d^2 + k_3 v_d^3, \tag{3}$$

for $I_s = 10^{-12}$, $V_T = 0.026$ in (1), which are shown in Table 1.

Table 1.
Exact Fourier coefficients and those of 3rd order Taylor expansion.

	Exact solutions				3rd order Taylor series			
V_d	DC	Fund.	2nd	3rd	DC	Fund.	2nd	3rd
0.02	0.0151	0.0113	0.0022	0.0003	0.0151	0.0112	0.0021	0.0003
0.05	0.0295	0.0413	0.0179	0.0055	0.0259	0.0390	0.0130	0.0043
0.07	0.0538	0.0859	0.0469	0.0191	0.0383	0.0720	0.0255	0.0119
0.10	0.1460	0.2542	0.1624	0.0870	0.0647	0.1560	0.0520	0.0347

We can see that the errors for the small input V_d are small enough, but they are rapidly increased for the large input. Next, when the higher order Taylor approximations are used, the results for the input such as $V_d = 0.1$ are shown together with the exact ones in Table 2.

Table 2.Solutions of order of the Taylor expansions and the exact solutions, for $V_d = 0.1$

	DC	Fund.	2nd	3rd
Exact	0.1459	0.2538	0.1670	0.0870
3rd order	0.0646	0.1559	0.0519	0.0347
5th order	0.1162	0.2250	0.1216	0.0696
7th order	0.1397	0.2483	0.1559	0.0832

Observe that if we apply it to 7th order Taylor approximation, the errors is again reduced to small enough. Thus, we found that the higher order Taylor approximation of the exponential function is needed for larger input.

2.2 Perturbation like Fourier expansion algorithm In practical transistor circuits, the input signals are sufficient small compared to the DC bias voltages. Let us describe it as follows:

$$v_{in}(t) = \varepsilon(V_1 \cos\omega_1 t + V_2 \cos\omega_2 t). \tag{4}$$

Then, the diode voltages together with the DC component V_{d0} are described by

$$v_d(t) = V_{d0} + \sum_{k=1}^{K} \varepsilon^k \left[\sum_{m=1}^{2k} (V_{d,k,2m-1} \cos\nu_{km} t + V_{d,k,2m} \sin\nu_{km} t) \right] \tag{5.1}$$

$$\nu_{km} = p_{1,km}\omega_1 + p_{2,km}\omega_2, \text{ for integers } |p_{1,km}|, |p_{2,km}| \le k \tag{5.2}$$

We take an account the following terms for the input (4);

$$\begin{array}{lll} \varepsilon \text{ order } \nu_{1m} & \cdots & \omega_1,\ omega_2 \\ \varepsilon^2 \text{ order } \nu_{2m} & \cdots & 2\omega_1,\ 2\omega_2,\ \omega_1+\omega_2,\ |\omega_1-\omega_2| \\ \varepsilon^3 \text{ order } \nu_{3m} & \cdots & 3\omega_1,\ 3\omega_2,\ 2\omega_1+\omega_2,\ \omega_1+2\omega_2, \\ & \cdots & |2\omega_1-\omega_2|,\ |\omega_1-2\omega_2| \\ etc. & & \end{array} \tag{6}$$

In this case, two cross term $v_1(t) \times v_2(t)$ having the forms of (5.1) is given by

$$v_3(t) = v_1(t) \times v_2(t)$$
$$\simeq V_{30} + \sum_{k=1}^{K} \varepsilon^k \left[\sum_{m=1}^{2k} (V_{3,k,2m-1} \cos\nu_{km} t + V_{3,k,2m} \sin\nu_{km} t) \right]. \tag{7}$$

If we consider all the terms lower than Kth order about ε, they can be estimated by the combinations of the terms of (5.1) where the higher order terms than $\varepsilon^(K+1)$ are neglected.

Thus, we call the algorithm *perturbation like Fourier expansion* in the meaning that our method contain all the terms less than order Kth of ε. Thus, we can reduce the number of cross terms. Observe that if choose a larger K, we can expect much more exact Fourier expansion.

2.3 Spice-oriented HB method We propose here a Spice-oriented HB (harmonic balance) method combining with MATLAB, where the *determining equation* is given by the equivalent circuit and/or net-list of Spice [9]. Firstly, we describe the nonlinear characteristics in the polynomial forms. The schematic diagram of diode example is shown in Fig.1, where it is driven by the two inputs

$$v_1(t) = \varepsilon V_1 \cos\omega_1 t, \quad v_2(t) = \varepsilon V_2 \cos\omega_2 t. \tag{8}$$

Firstly, it is approximated by Mth order Taylor expansion as follow:

$$i_d = k_0(V_{d0}) + k_1(V_{d0})v_d + \cdots + k_M(V_{d0})v_d^M \tag{9}$$

with an unknown operating point V_{d0}. Now, we rewrite (9) into the following Horner's scheme

$$\left.\begin{array}{l} i_{d1}(v_d) = K_{M-1}(V_{d0}) + K_M(V_{d0})v_d \\ i_{d2}(v_d) = k_{M-2}(V_{d0}) + i_{d1}(v_d)v_d \\ \dots\dots\dots\dots\dots\dots\dots\dots \\ i_{dM}(v_d) = k_0(V_{d0}) + i_{dM-1}(v_d)v_d \end{array}\right\}, \tag{10}$$

so that we have $i_d = i_{dM}(v_d)$. Each cross product $i_{dk}(v_d) \times v_d$ in (10) is calculated by the method of (7) using MATLAB. Thus, we can execute the Fourier expansion and get the module of the diode as shown in Fig.1(b). Now, we formulate "Sine-Cosine circuit"[9,10], which corresponds to the determining equation of HB.

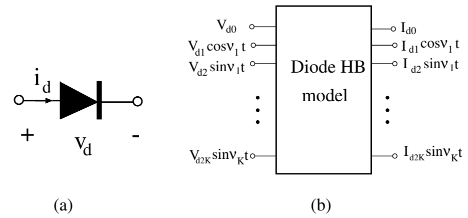

Fig.1 (a) a diode model, (b) HB module with MATLAB.

The circuit is resistive, and it can be easily solved with the DC analysis of Spice. Thus, we can get the frequency-domain solutions such as frequency response curves and so on.

It is convenient to develop the HB modules for bipolar transistors. In this case, if we model the transistor with Ebers-Moll model, the emitter, base and corrector currents are given as follows:

$$\left.\begin{array}{l} i_E = I_s \exp(\frac{v_{BE}}{V_T}) - \alpha_R I_s \exp(\frac{v_{BC}}{V_T}) \\ i_C = \alpha_F I_s \exp(\frac{v_{BE}}{V_T}) - I_s \exp(\frac{v_{BC}}{V_T}) \\ i_B = (1-\alpha_F) I_s \exp(\frac{v_{BE}}{V_T}) + (1-\alpha_R) I_s \exp(\frac{v_{BC}}{V_T}) \end{array}\right\} \tag{11}$$

for $I_S - 10^{-12}$, $V_T = 0.026$, $\alpha_F = 0.99$, $\alpha_R = 0.3$

They consist of the exponential functions, so that HB transistor module can be developed with the diode HB module shown by Fig.1(b).

2.4 Comparisons between Volterra series and HB methods We consider a simple base-modulator circuit as shown in Fig.2(a) [13]. Firstly, let us solve it with the Volterra series method.

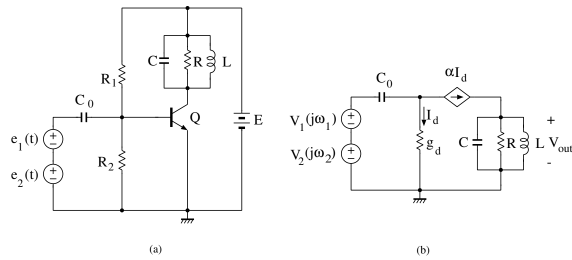

Fig.2(a) Base modulation circuit, (b) 1st order Volterra equivalent circuit.

$R_1 = 50[k\Omega]$, $R_2 = 10[k\Omega]$, $R = 0.2[k\Omega]$

$C = 1[nF]$, $C_0 = 1[nF]$, $C_0 = 1[nF]$, $L = 10[\mu F]$, $E = 10[V]$

$v_1 = 0.02 \sin\omega_1 t$, $v_2 = 0.02 \sin\omega_1 t$, for $\omega_2 = 8.2 \times 10^6 [rad/sec]$

The transistor is replaced by the Ebers-Moll model, and the operating point is firstly calculated and we have $v_{d0} = 0.5822[V]$. Next, the nonlinear characteristic is approximated by the 3rd order Taylor expansion at the point v_{d0} as follow:

$$i_d = 0.0130 + 0.52v_d + 10.40v_d^2 + 138.60v_d^3. \qquad (12)$$

Note that, for these parameters, we have $v_{BC} < 0$ and $I_s \exp(v_{BC}/V_T) = 0$, so that 1st order Volterra kernel is given by Fig.2(b). Next, the 2nd and 3rd Volterra kernels are calculated according to the method of ref. [3]. The frequency-domain inter-modulation phenomena as shown in Fig.2(c) is easily obtained by the application of MATLAB.

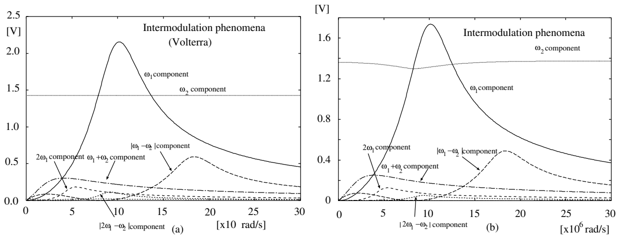

Fig.2(c),(d) The intermodulation phenomena with Volterra series method and HB method.

Next, we analyze the circuit with HB method. Firstly, the transistor is modeled by Ebers-Moll model and two diodes are replaced by diode HB modules as shown in Fig.1(b), where the diodes are approximated by the 3rd order Taylor expansion and it is expanded in the Fourier series with MATLAB. After then, we formulated the "Sine-Cosine circuits"[9,10] corresponding to the frequency components in (6). Their sub-circuits are coupled with controlled sources in each others that corresponds to the determining equation of Spice. Therefore, the frequency-domain intermodulation can be calculated by the DC analysis of Spice as shown in Fig.6(d).

Note that although the above two response curves obtained by Volterra series and HB methods are similar forms in each other, the amplitudes are largely different around the resonant point. We have also calculated the transient responses, where transistors are modeled by the exponential functions. The ω_1 and ω_2 frequency-components are calculated with DFT from the steady-state waveforms as shown in Table 3.

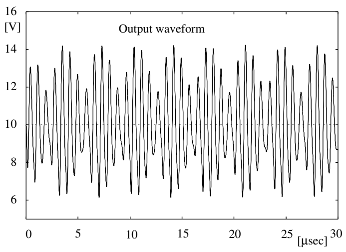

Fig.2(e) Steady-state wave form of base modulator circuit.

Table 3 Comparisons of amplitudes
($\omega_1 = 10^7[dad/s]$, $\omega_2 = 8.2 \times 10^6[rad/s]$)

	Volterra		Harmonic balance		DFT	
$V_1 = V_2$	ω_1	ω_2	ω_1	ω_2	ω_1	ω_2
0.01	0.950	0.693	0.929	0.691	0.913	0.698
0.02	2.20	1.41	1.72	1.31	1.68	1.29
0.03	3.91	2.32	2.50	1.92	2.17	1.72
0.04	6.25	3.35	3.35	2.59	2.50	2.11
0.05	9.80	4.55	4.31	3.34	2.75	2.38

Note that the results from transient analysis may be most accurate, because the transistor is modeled by Ebers-Moll model. On the other hand, the Volterra method approximate the Ebers-Moll model in 3rd order polynomial at fixed operating point. Our HB method approximate it as an unknown operating point decided by HB analysis. Other differences are come from the analytical methods as follows; For smaller input, the intermodulation phenomena both the Volterra and HB methods are almost same features. However, the amplitudes of the Volterra method become much larger than those of HB, especially around the resonant point as shown in Table 3. Therefore, it seems from the results that the Volterra series method becomes erroneous for the strong nonlinear circuits having large inputs. Next, we will investigate the modulations and get the following results:

Table 4 Modulation results for $R = 1k\Omega$.

	ω_1	$omega_2$	$\lvert\omega_1 - \omega_2\rvert$	$\lvert\omega_1 + \omega_2\rvert$
3rd order	20.33	4.92	7.15	3. 37
7th order	10.38	2.81	9.73	4.58
Exact	8.45	2.21	7.58	3.86

Although both the frequency response curves have almost the same phenomena, the peak values to the 3rd order Taylor approximations are larger than those of 7th order.

3. Illustrative examples

We consider a mixer circuit containing 3 transistors as shown in Fig.3(a) [13]. Note that it works as a modulator if we set $v_1(t)$ carrier signal, $v_2(t)$ input signal and choose the LC resonant frequency equal to the carrier frequency. Now, we analyze the intermodulation phenomena with our HB method. Firstly, the transistors are modeled by the Ebers-Moll model, where

$$i_d = I_s e^{\lambda v_d}, \text{ for } I_s = 10^{-12}[A], \ \lambda = 40, \text{ and } \alpha = 0.99 \quad (13)$$

and they are approximated by 3rd order Taylor series as follows:

$$\equiv k_0(v_{d0}) + k_1(v_{d0})v + k_2(v_{d0})v^2 + k_3(v_{d0})v^3 \quad \text{for } v_d = v_{d0} + v \quad (14)$$

In this case, we applied 3rd order Taylor approximation method. Firstly, all the transistors are replaced by the corresponding HB transistor modules, and RLC circuit is transformed into "Sine-Cosine circuits" [9,10]. Thus, the circuit is transformed into the 13 HB sub-circuits coupled with controlled sources, which can solved by the DC analysis of Spice.

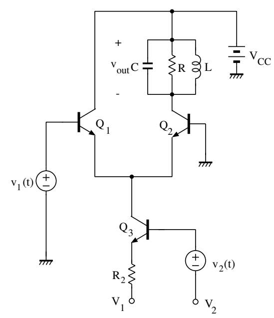

Fig.3(a) Mixer circuit.
$R_2 = 10[k\Omega]$, $R = 100[\Omega]$, $L = 10[\mu H]$,
$V_{CC} = 10[V]$, $V_1 = -10[V]$, $V_2 = -5[V]$

For the mixer circuit, we had the intermodulation result as shown in Fig.3(b), and the steady-state waveforms in Fig.3(c)

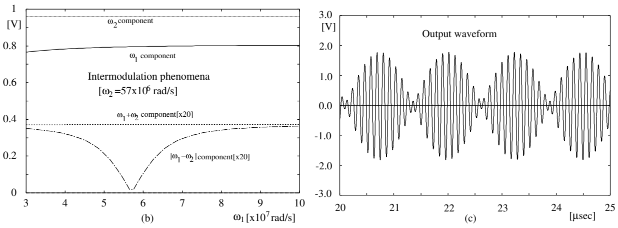

Fig.3(b) Frequency response of mixer circuit, $C=1[pF]$, $v_1=0.002\sin\omega_1 t$, $v_2=0.2\sin\omega_2 t$, for $\omega_2=57\times10^6[rad/s]$.
(c) Inputs $v_1(t)=0.002\sin 50\times10^6 t$, $v_2(t)=0.2\sin 57\times10^6 t$

From DFT analysis of the waveform, we have the following results:

$$\left.\begin{array}{l} V_{out,1}=0.7919[V](\text{HBmethod})\ \ V_1=0.8311[V](\text{DFT}) \\ \qquad\qquad \text{for component}\ \ \omega_1=57\times10^6[rad/s] \\ V_{0ut,2}=0.9603[V](\text{HBmethod})\ \ V_2=0.9585[V](\text{DFT}) \\ \qquad\qquad \text{for component}\ \ \omega_2=50\times10^6[rad/s] \end{array}\right\}$$

The mixer components are $\omega_1+\omega_2$ and $|\omega_1-\omega_2|$, and the $|\omega_1-\omega_2|$-component become zero at $\omega_1=\omega_2$. The computation time is 102.05[s].
Next, we analyze the circuit as modulator. In this case, the resonant frequency is equal to $110^7[rad/s]$. The inptermodulation frequency phenomena is shown by Fig.3(d) and the steady-state waveform by Fig.3(e).

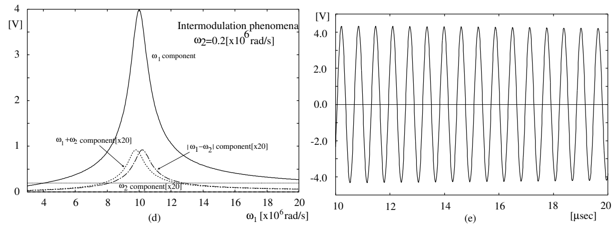

Fig.3(d) Frequency response of modulator. $C=1[nF]$, $v_1=0.001\sin\omega_1 t$, $v_2=0.1\sin\omega_2 t$, for $\omega_2=0.2\times10^6[rad/s]$.
(e) Input $v_1(t)=0.001\sin 10^7 t$, $v_2(t)=0.1\sin 0.2\times10^6 t$

We have the following results:

$$\left.\begin{array}{l} V_{out,1}=3.987[V](\text{HBmethod})\ \ V_1=4.218[V](\text{DFT}) \\ \qquad\qquad \text{for component}\ \ \omega_1=10^7[rad/s] \\ V_{out,2}=0.307[V](\text{HBmethod})\ \ V_2=0.337[V](\text{DFT}) \\ \qquad\qquad \text{for component}\ \ \omega_1=0.2\times10^6[rad/s] \\ V_{out}=85.5[mV]\quad \text{at}\ \ \omega_1-\omega_2,\omega_1-\omega_2 \end{array}\right\}$$

The computation time is 209.30[s].

4. Conclusions and remarks

In this paper, we compared Volterra series method with our Spice-oriented HB method for the analysis of intermodulation phenomena, and we found the following results:

1. Both methods belong to the symbolical analysis, in the meaning that the nonlinear devices must be described by the power series. Volterra method must be firstly found the operating point, and use by Taylor expansion. Our HB method also use Taylor expansion in symbolical form. The determining equation is given by the equivalent circuit so that our HB method is a kind of symbolic method.
2. Both method give good results for small input.
3. Volterra series method can be only applied to weakly nonlinear circuits. Our HB method can be applied to relatively strong nonlinear circuits with a few harmonics.

As the future work, we need to apply our HB method to much more strong nonlinear elements, and develop of HB modules of MOSFETs.

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] R.Telichevesky, K.S.Kundart and J.K.White, "Efficient steady-state analysis based on matrix-free Krylov-subspace methods," *ACM,* pp.480-485, 1995.

[5] K.Kundart "Accurate Fourier analysis for circuit simulators," *CICC94,* pp.25-32, 1994.

[6] R.Telichevesky, K.S.Kundart,I El-Fadel and J.K.White, "Fast simulation algorithms for RF circuits," *CICC96,* pp.1-8, 1996.

[7] Y.Ueda, *The Road to Chaos-II*, Aerial Press. Inc., 2001.

[8] 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. Jour. of Microwave and Millimeter-Wave Computer-Aided Eng.* vol.1, pp.22-37, 1991.

[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] A.Ushida, J.Kawata, Y.Yamagami and Y.Nishio, " Intermodulation analysis of nonlinear circuits using two-dimensional Fourier transformation," *NOLTA'05*, Budges, pp.282-285, 2005.

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

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

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