2009 International Workshop on Nonlinear Circuits and Signal Processing
NCSP'09, Waikiki, Hawaii, March 1-3, 2009

Spice-Oriented Nonlinear Circuit Analysis Using Harmonic Balance Method

Takaaki Kinouchi† Yoshihiro Yamagami† Yoshifumi Nishio† and Akio Ushida‡

†Tokushima University
2-1 Minami-Josanjima, Tokushima, Japan
Phone:+81-88-656-7470
FAX:+81-88-656-7471
Email: {t-kino, yamagami, nishio}@is.tokushima-u.ac.jp

‡Tokushima Bunri University
1314-1, Sido, Sanuki, Kagawa, Japan
Email: ushida@fe.bunri-u.ac.jp

1. Abstract

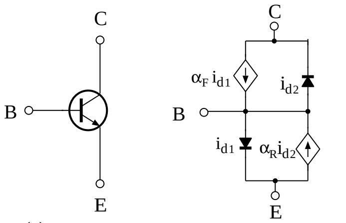
To analyze RF circuits and modulators is very important for designing integrated and communication circuits. They are driven by multiple frequencies, one of which is usually very high carrier frequency compared to the other. To know the transient behaviors, we need to calculate many carrier waveforms, so that the transient analysis is very time-consuming. Hence, we propose an efficient *envelope analysis* for calculating the asymptotic behaviors of the amplitudes, which is based on the harmonic balance (HB) method with the slowly varying coefficients. In order to develop the Spice-oriented simulators, the Fourier expansions of nonlinear devices such as bipolar transistors are executed with MATLAB, and the Fourier modules should be stored in our computer library. Thus, we can easily formulate the *determining equations of HB* method called *Sine-Cosine circuit*.

2. Introduction

For designing integrated circuits and communication systems, it is very important to analyze the characteristics of nonlinear circuits. In these systems driven by multi-frequency inputs. There are various method to solve the characteristics. In this paper, we propose an efficient envelope analysis. The frequency components of communication systems usually consist of very high carrier frequency and low signal. Then, the transient analysis based on the numerical integration technique is very time-consuming, because we need to calculate many carrier waveforms to know the asymptotic transient behaviors. Our envelope analysis is a modification of HB (harmonic balance) method to the high carrier frequency components, where we assume that the Fourier coefficients are slowly varied by the low frequency signal. The harmonic balance (HB) is a well-known method for the frequency domain analysis, which gives good results even for relatively strong nonlinear circuits. In our HB, which combining MATLAB and Spice, it is largely improved such that it can be easily applied to large scale nonlinear circuits, where the determining equation called *Sine-Cosine circuit* can be given in the form of schematic and/or net-list of Spice. The circuit can be solved by Spice simulator and the frequency characteristics are easily obtained.

3. Nonlinear devices of HB combining with MATLAB

In this section, we propose our HB devices combining Spice and MATLAB. Analog integrated circuits usually consist of many kinds of nonlinear devices such as diodes, bipolar transistors and MOSFETs, whose Spice models are described by the several special functions such as exponential, square-root, piecewise-continuous functions and so on. For these special functions, the Fourier expansions cannot be executed in the analytical forms. Therefore, we need to approximate them with the Taylor expansions. For an example of NPN bipolar transistor shown by Fig.1(a), the Ebers-Moll model is given by (b).



(a) NPN Transistor (b) Ebers-Moll model

Fig.1 NPN transistor Ebers-Moll model.

The characteristics of two diodes with Ebers-Moll models are given by

$$i_d = I_S \exp(\frac{v_d}{V_T}), \tag{1}$$

whose parameters are given, for example, $I_S = 10^{-12}$, $V_T = 0.025$. Applying the Taylor expansion, we have

$$i_d = I_S \exp(\frac{v_{d0}}{V_T})\left(1 + \lambda v_d + \frac{\lambda^2}{2!}v_d^2 + \frac{\lambda^3}{3!}v_d^3 + \ldots\right) \tag{2}$$

for a small variation v_d at the operating point v_{d0}, and $\lambda =$

$\exp(\frac{1}{V_T})$. We assume the input waveform as follows;

$$v_d(t) = V_{0d} + \sum_{k=1}^{K} \left(V_{d,2k-1} \cos k\omega t + V_{d,2k} \sin k\omega t \right), \quad (3.1)$$

where K is the maximum higher harmonic component to take into account. Then, the output waveform of (1) is calculated. However, these calculation is very large and complex. Therefore, we use MATLAB and calculated as follows;

$$i_d(t) = I_{0d} + \sum_{k=1}^{K} \left(I_{d,2k-1} \cos k\omega t + I_{d,2k} \sin k\omega t \right), \quad (3.2)$$

in the symbolic form. Namely, $(I_{0d}, I_{d2}, \ldots, I_{d,2K})$ are functions of $(V_{0d}, V_{d2}, \ldots, V_{d,2K})$, and the input-output relations are realized by ABMs of Spice;

$$\begin{array}{l} I_{k,C} = \hat{I}_{k,C}(V_{0C}, \ldots, V_{2K,C}, V_{0B}, \ldots, V_{2K,B}, V_{0E}, \ldots, V_{2K,E}) \\ I_{k,B} = \hat{I}_{k,B}(V_{0C}, \ldots, V_{2K,C}, V_{0B}, \ldots, V_{2K,B}, V_{0E}, \ldots, V_{2K,E}) \\ I_{k,E} = \hat{I}_{k,E}(V_{0C}, \ldots, V_{2K,C}, V_{0B}, \ldots, V_{2K,B}, V_{0E}, \ldots, V_{2K,E}) \\ k = 0, 1, 2, \ldots, 2K \end{array} \quad (4)$$

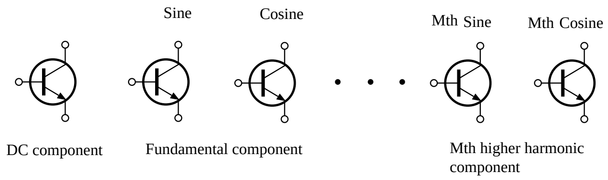


Fig.2 Symbol of NPN transistor Fourier module.

It is convenient to store the Fourier modules in our computer library. Replacing each element by the corresponding HB model, we can formulate the "*Sine-Cosine circuit* that is corresponding to*the HB determining equation*. Resulting circuit is efficiently solved by Spice.

4. Envelope analysis

4.1 RLC circuit Now, consider a simple example RLC circuit Fig.3 driven by

$$v(t) = V \sin(\omega_s t) \sin(\omega_c t), \quad \omega_c \gg \omega_s. \quad (6)$$

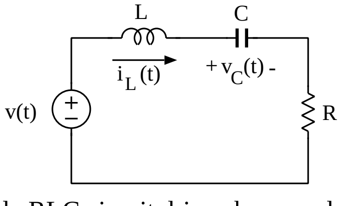


Fig.3 A simple RLC circuit driven by a modulated signal.

We assume the inductor current and capacitor voltage as follows;

$$\begin{array}{l} i_L(t) = I_C(t) \cos \omega_c t + I_S(t) \sin \omega_c t, \\ v_C(t) = V_C(t) \cos \omega_c t + V_S(t) \sin \omega_c t \end{array} \quad (7)$$

Then, the inductor voltage and capacitor current are given by

$$v_L(t) = L\frac{di_L}{dt} = \omega_c L I_S(t) \cos \omega_c t - \omega_c L I_C \sin \omega_c t$$
$$+L\frac{dI_C}{dt} \cos \omega_c t + L\frac{dI_S}{dt} \sin \omega_c t, \quad (8.1)$$
$$i_C(t) = C\frac{dv_C}{dt} = \omega_c C V_S(t) \cos \omega_c t - \omega_c C V I_C \sin \omega_c t$$
$$+C\frac{dV_C}{dt} \cos \omega_c t + C\frac{dV_S}{dt} \sin \omega_c t. \quad (8.2)$$

Using (8.1) and (8.2), we set the Cosine-component and Sine-component of Fig.3 in the manner that they respectively satisfy the circuit equations. Thus, the Sine-Cosine circuit are obtained as shown in Fig.4.

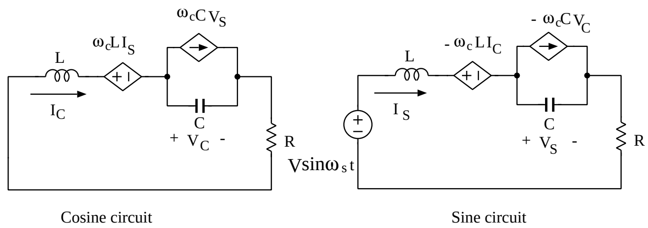
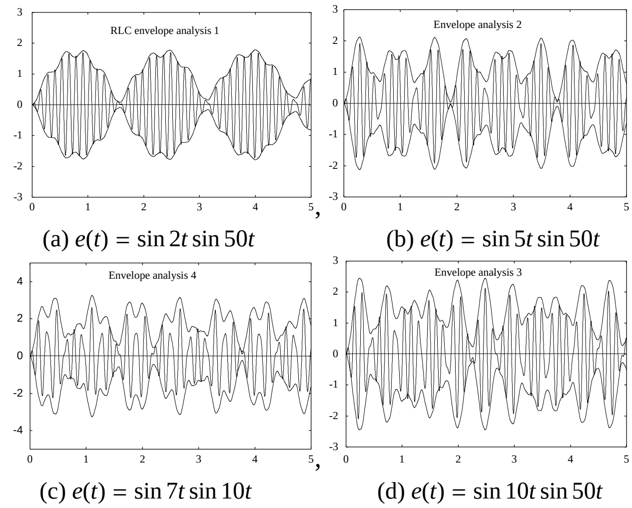


Fig.4 Sine-Cosine circuit of RLC.

They are coupled with the voltage-controlled current sources and current-controlled voltage sources in each other. From the relation (6), the voltage $V \sin \omega_s t$ in Fig.4 corresponds to the envelope of the forced input which is slowly varying because of $\omega_c \gg \omega_s$. Now, we show that our envelope approach is validity when the difference between two input frequencies is large. The parameter values are set as follow;
$L = 0.1, \ C = 0.01, \ R = 0.01, \ \omega_c = 50, \ V = 1,$
and the other frequency is changed to $\omega_s = 2, \ 5, \ 7, \ 10$.

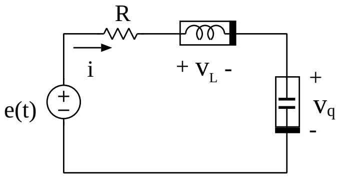


(a) $e(t) = \sin 2t \sin 50t$ (b) $e(t) = \sin 5t \sin 50t$

(c) $e(t) = \sin 7t \sin 10t$ (d) $e(t) = \sin 10t \sin 50t$

Fig.5 Envelopes of currents in RLC circuit of Fig.4.

4.2 Nonlinear RLC circuit Next, we consider nonlinear RLC circuit driven by frequency modulated (FM) signal shown Fig.6.



$e(t) = E\cos\omega t$, $E = 1[V]$, $R = 1[\Omega]$, $\omega = 0.01t$[rad/sec]
Fig.6 Nonlinear RLC circuit

Let the nonlinear capacitor and inductor be are

$$v_q = \alpha q + \beta q^3, \tag{9.1}$$
$$i = \gamma\phi + \delta\phi^3 \tag{9.2}$$

$$\alpha = 1, \quad \beta = 7, \quad \gamma = 0.142857, \quad \delta = 0.$$

Assume the waveforms as follow;

$$q = Q_c\cos\omega t + Q_s\sin\omega t, \tag{10.1}$$
$$\phi = \Phi_c\cos\omega t + \Phi_s\sin\omega t \tag{10.2}$$

The characteristic of nonlinear capacitor is shown in Fig. 7.

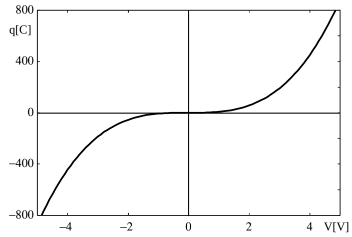


Fig. 7 Characteristic of nonlinear capacitor

Next, substituting (10.1) into (9.1) and (10.2) into (9.2) respectively, we have fundamental harmonic components as follows,

$$\begin{aligned} v_q = (\alpha + 0.75\beta(Q_c^2 + Q_s^2))Q_c\cos\omega t \\ + (\alpha + 0.75\beta(Q_c^2 + Q_s^2))Q_s\sin\omega t, \end{aligned} \tag{11.1}$$

$$\begin{aligned} i = (\gamma + 0.75\delta(\Phi_c^2 + \Phi_s^2))\Phi_c\cos\omega t \\ +(\gamma + 0.75\delta(\Phi_c^2 + \Phi_s^2))\Phi_s\sin\omega t. \end{aligned} \tag{11.2}$$

neglecting the higher harmonic components. We have from (10.1) and (10.2).
On the other hand,

$$i = \frac{dq}{dt} = (\dot{Q}_c + \omega Q_s)\cos\omega t + (\dot{Q}_s - \omega Q_c)\sin\omega t, \tag{12.1}$$

$$v_L = \frac{d\Phi}{dt} = (\dot{\Phi}_c + \omega\Phi_s)\cos\omega t + (\dot{\Phi}_s - \omega\Phi_s)\sin\omega t. \tag{12.2}$$

Terefore, sine and cosine components are as follows;

$$\begin{aligned} I_c = \dot{Q}_c + \omega Q_s, \quad I_s = \dot{Q}_s - \omega Q_c, \\ V_c = \dot{\Phi}_c + \omega\Phi_s, \quad V_s = \dot{\Phi}_s - \omega\Phi_s \end{aligned} \tag{13}$$

Q_c, Q_s, Φ_c, and Φ_s are unknown parameters, therefore it is necessary to make a circuit to satisfy equation (13). Thus, we have Sine-Cosine circuit as show in Fig.8. So these parameters are given by Fig.8(c)-(f).

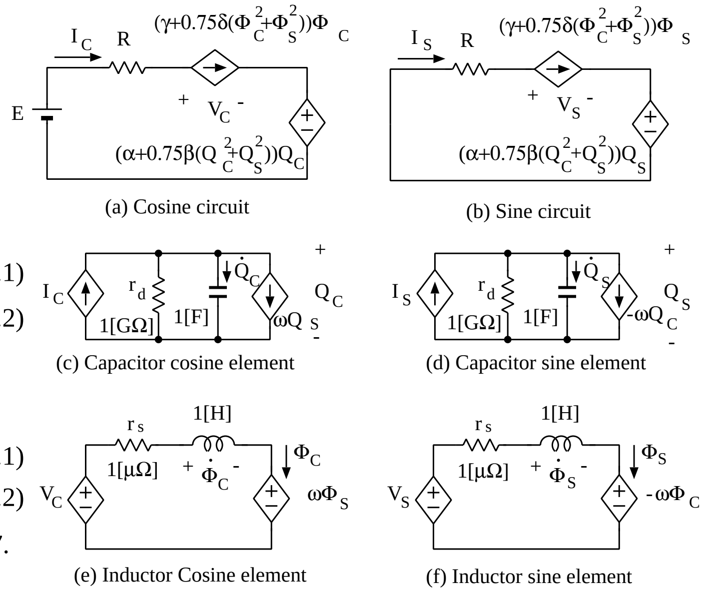


Fig.8 Sine-Cosine circuit of nonlinear RLC

For the capacitor element, equations (13) are realized by Fig.8 (c), (d). Where we assume the voltages of two capacitors 1[F] are Q_c and Q_s. In the same way, the sine-cosine inductor circuits are given by Fig.8 (e), (f). Where we substitute small dummy resistors r_s and large Amy resistors r_d to avoid C-E loops and L-J cut sets. Fig.8 is corresponds to the determining equation of the HB method.

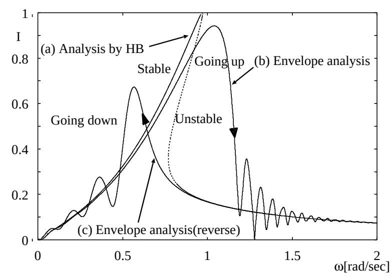


Fig.9 Comparison between the HB and Envelope analysis.

Thus, we obtain the envelope analysis by solving the circuit with transient analysis of Spice shown in Fig.9. The curve (a) is the jump phenomena where the dotted line is unstable and solid line stable. When the frequency is changed by ω=0.01t, we have phenomena (b) which has oscillations after jumping down. When the frequency is changed ω=2-0.01t, we have phenomena (c) which has oscillations after jumping up.

4.3 RF amplifier Consider a simple RF amplifier shown in Fig.10 whose input is assumed by

$$v(t) = V_1\cos\omega_1 t + V_2\cos\omega_2 t, \quad \text{for} \quad \omega_2 = \omega_1 + \Delta\omega, \tag{14.1}$$

for a small $\Delta\omega$. Then, it is written by

$$\begin{aligned} v(t) &= (V_1 + V_2 \cos\Delta\omega t)\cos\omega_1 t - (V_2 \sin\Delta\omega t)\sin\omega_1 t \\ &= V_C(t)\cos\omega_1 t + V_S(t)\sin\omega_1 t \end{aligned} \tag{14.2}$$

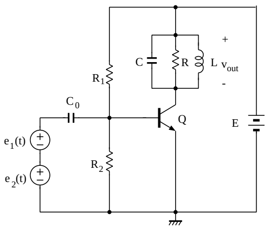


Fig.10 Simple emitter follower amplifier.
$C_0 = 1[nF],\quad C = 1[nF],\quad L = 10[\mu F],\quad R_1 = R_2 = 50[k\Omega]$
$R = 1[k\Omega],\quad E = 10[V],\quad \omega_1 = 10^7[rad/sec]$

We have used the Ebers-Moll model for the transistor,

$$i_d = 10^{-12}\exp[40v_d],\quad \alpha_F = 0.99,\ \alpha_R = 0.3, \tag{15}$$

and approximate it by Taylor expansion. Then, the Fourier module can be obtained by MATLAB in the symboric forms as way of section 2. Thus, the Sine-Cosine circuits of RF circuit can be given in Fig.10.

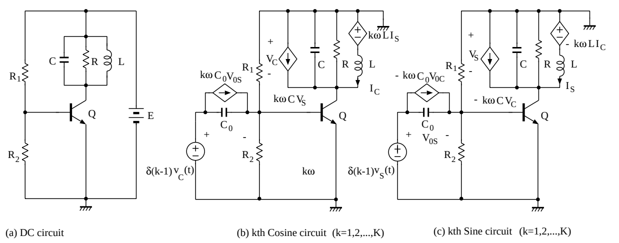
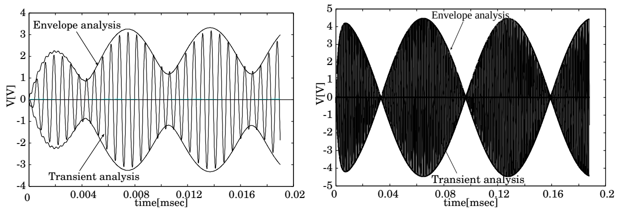


Fig.11 Sine-Cosine circuit of the amplifier shown by Fig.10.

The DC circuit shown by Fig.11(a) only has a DC voltage source, where the input source is removed by the short-circuit. The Sine-Cosine circuits have slowly varying inputs $\delta(k-1)V_C(t)$, and $\delta(k-1)V_S(t)$ to the Cosine and Sine sub-circuits, respectively. Observe that all the circuit configurations corresponding to the kth harmonic components are the same topology. Therefore, we can easily formulate the circuits by "copy and paste" with Spice simulator. Ih this study, we consider maximum harmonic component K=3. Thus, DC circuit and each three of Sine-Cosine circuit, total number of severn circuits are created. All the sub-circuits are coupled with the controlled sourced and the Fourier transistor modules. The DC circuit is also time-varying. Note that the resultant Sine-Cosine circuits correspond to the *determining equations of HB method* and it can be solved by the transient analysis of Spice.



(a) $\Delta\omega = 0.1$, (b) $\Delta\omega = 0.01$
Fig.12 Comparisons between the transient and envelope analysis.

Thus, the asymptotic behavior of the amplitudes can be calculated by transient analysis of Spice as shown in Fig.12.

Table 1 Comparison calcurational time between Envelope and Transient analysis

$\Delta\omega$[Mrad/sec]	Envelope[sec]	Transient[sec]
0.1	3.75	20.45
0.07	4.83	29.48
0.05	5.5	40.55
0.03	5.53	65.05
0.01	11.12	189.86

And calculational time as shown Table 1. Observe that between transient and envelope. When $\Delta\Omega = 0.1$, our envelope analysis is faster about 5 times. However, when $\Delta\omega = 0.01$, faster than about 17 times. Therefore, when ω_2 nearly to ω_1 namely, $\Delta\omega$ become more small, envelope analysis is more equal as shown in Fig.12. And calculational time of our envelope analysis is faster than direct transient analysis shown in Table 1.

5. Conclusions

In this paper, we propose a Spice-oriented envelope analysis which is based on Harmonic Balance method. The coefficients of HB set to slowly varying. Thus, we can obtained efficient asymptotic analysis result. Fourier expansion applies to nonlinear devices by use MATLAB, and the determining equation of HB is replaced Sine-Cosine circuit. Therefore, calculation time of our envelope analysis is faster than direct transient analysis. Also we can analyze the circuit consists nonlinear inductor and capacitor. So, it is promising to analysis more expansion for nonlinear circuits.

References

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

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

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

[4] T. Kinouchi, Y. Yamagami, Y. Nishio, J. Kawata, and A. Ushida, "Spice-Oriented Harmonic Balance and Volterra Series Methods," Proc. of NOLTA'07, pp.513-516, 2007.