PAPER

Analysis of Reactance Oscillators Having Multi-Mode Oscillations

Yoshihiro YAMAGAMI[†a)], **Yoshifumi NISHIO**[†b)], ***Members***, ***and*** **Akio USHIDA**[††c)], ***Fellow***

SUMMARY We consider oscillators consisting of a reactance circuit and a negative resistor. They may happen to have multi-mode oscillations around the anti-resonant frequencies of the reactance circuit. This kind of oscillators can be easily synthesized by setting the resonant and anti-resonant frequencies of the reactance circuits. However, it is not easy to analyze the oscillation phenomena, because they have multiple oscillations whose oscillations depend on the initial guesses. In this paper, we propose a Spice-oriented solution algorithm combining the harmonic balance method with Newton homotopy method that can find out the multiple solutions on the homotopy paths. In our analysis, the *determining equations* from the *harmonic balance method* are given by modified equivalent circuit models of "DC," "Cosine" and "Sine" circuits. The modified circuits can be solved by a simulator STC (solution curve tracing circuit), where the multiple oscillations are found by the transient analysis of Spice. Thus, we need not to derive the troublesome circuit equations, nor the mathematical transformations to get the determining equations. It makes the solution algorithms much simpler.

key words: *oscillator, multiple oscillations, harmonic balance method, Newton homotopy method, SPICE*

1. Introduction

Analysis of oscillator circuits is very important for designing communication circuits such as modulators and mixers. Generally, oscillators have both zero amplitude unstable solutions and non-zero amplitude stable and/or unstable orbits. In the analysis, it is very difficult to find out the multiple oscillations, because the oscillations depend on the initial guesses in the transient simulation. Furthermore, the transient analysis sometimes becomes very time-consuming especially for high Q oscillators. Nowadays, many kinds of coupled oscillators have been proposed [1]–[3], which have many interesting phenomena such as multiple oscillations, quasi-periodic oscillations, chaos and so on. Thus, in order to analyze these phenomena, it will be useful to develop simulators to find out the multiple oscillations.

For circuits having a unique non-zero amplitude oscillation, there are Newton-like shooting algorithms [4]–[7] solving the following relation;

$$\mathbf{F}(\mathbf{v}(0), T) = \mathbf{v}(0) - \mathbf{v}(T) = \mathbf{0}, \tag{1}$$

where $\mathbf{v}$ corresponds to the state-variables of the circuit, and T is the period to be determined. Since the algorithms are based on the Newton-Raphson method, we need to start the iterations from a suitable initial guess. The convergence is guaranteed only when the initial guess is located near the steady-state solution. W. Ma, L.J. Trajkovic and L. Mayaram [7] have proposed an elegant algorithm combining a homotopy method with the above shooting method. The approach has a global convergence property, so that it can be safely applied to find out the solution satisfying (1). However, it is still difficult to choose a suitable initial guesses converging to the multiple oscillations.

In this paper, we propose an elegant Spice-oriented harmonic balance method for solving multiple oscillations. We apply it to an oscillator consisting of a reactance circuit coupled with a negative resistor. At first, the circuit is designed by specifying the resonant and the anti-resonant frequencies of a reactance circuit such as Cauer or Foster circuit. After then, the oscillator is realized by introducing a negative resistor. We found that the oscillations may happen around the anti-resonant frequencies [8].

Now, we will outline our solution algorithm as follow:

1. For a given reactance oscillator, the waveform of the negative resistance is assumed as follows;

$$v(t) = V_0 + V_1 \cos\omega t + \sum_{k=2}^{M}[V_{2k-1}\cos k\omega t + V_{2k}\sin k\omega t] \tag{2}$$

   Note that we can set $V_2 \sin\omega t = 0$ in (2) because of an autonomous system [11], and ω is chosen as an additional variable.

2. The current of negative resistor is given by

$$i(t) = I_0(\mathbf{V}) + \sum_{k=1}^{M}[I_{2k-1}(\mathbf{V})\cos k\omega t + I_{2k}(\mathbf{V})\sin k\omega t] \tag{3}$$

   where $I_0(\mathbf{V}),\ I_1(\mathbf{V}),\ I_2(\mathbf{V}), \ldots, I_{2M}(\mathbf{V})$, are functions of $\mathbf{V} = (V_0, V_1, V_3, \ldots, V_{2M})^T$. Thus, each frequency component $(I_{2k-1}(\mathbf{V}),\ I_{2k}(\mathbf{V}))$ is described by the nonlinear voltage-controlled current sources in our harmonic balance method.

   For the linear inductances, they are transformed into

Manuscript received April 7, 2005.
Manuscript revised August 8, 2005.
Final manuscript received November 30, 2005.

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

DOI: 10.1093/ietfec/e89–a.3.764

the linear current-controlled voltage sources, and the linear capacitances into the linear voltage-controlled current sources. These transformation algorithms using ABMs (Analog Behavior Models) of Spice are shown in Sect. 3. This modified equivalent circuit corresponds to the *determining equations* of the harmonic balance method.

3. Introducing the Newton homotopy method, the modified circuit is solved by a transient analysis of Spice. In Sect. 2, we propose an solution algorithm to find out the multiple oscillations which consists of a curve tracing algorithm based on the arc-length method [9].

Thus, in our method, we need not derive any troublesome circuit equations, determining equations of the harmonic balance method nor the Jacobian matrices to solve the equations, which makes the solution algorithm much easier. We will show an interesting illustrative example of a Cauer oscillator in Sect. 4.

2. Harmonic Balance Method Combining with Newton Homotopy Method

In order to show the ideas of our solution algorithm, let us consider a reactance oscillator having a negative resistor as shown in Fig. 1. Assume that the circuit equation of reactance oscillator is written by

$$\mathbf{h}(v, \dot{\mathbf{w}}, \mathbf{w}) = \mathbf{0}, \quad \mathbf{h}(.,.,.) : R^{2n+1} \mapsto R^{2n+1}, \tag{4}$$

where v is the nonlinear resistor's voltage, and $\mathbf{w} \in R^n$ corresponds to the circuit variables of the linear reactance circuit. Now, let the nonlinear characteristic of the negative resistor be described in the following form;

$$i = \hat{g}(v). \tag{5}$$

We assume the voltage waveform of v as given in (2). Applying the *harmonic balance method* to the circuit equation (4), and eliminating the variables from $\mathbf{w}$, the *determining equations* for the harmonic components can be described as follows;

$$\left.\begin{array}{l} f_0(V_0, V_1, V_3, \cdots, V_{2M}, \omega) = 0 \ \cdots \ \text{DC} \\ f_1(V_0, V_1, V_3, \cdots, V_{2M}, \omega) = 0 \ \cdots \ \cos\omega t \\ f_2(V_0, V_1, V_3, \cdots, V_{2M}, \omega) = 0 \ \cdots \ \sin\omega t \\ \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots \\ f_{2M-1}(V_0, V_1, V_3 \cdots, V_{2M}, \omega) = 0 \ \cdots \ \cos M\omega t \\ f_{2M}(V_0, V_1, V_3, , \cdots, V_{2M}, \omega) = 0 \ \cdots \ \sin M\omega t \end{array}\right\}, \tag{6}$$

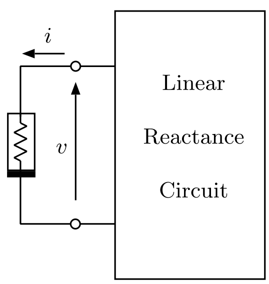

Fig. 1 Reactance oscillator.

where the fundamental frequency component "ω" should be determined as an additional variable, and M denotes the highest frequency component. Thus, the determining equations are described by a set of algebraic equations, which consist of $(2M + 1)$-equations and the same number of variables. However, it is not easy to solve the equations, because they may have the multiple solutions.

For simplicity, we set the relations (6) as follow;

$$\mathbf{f}(\mathbf{V}) = \mathbf{0}, \quad \mathbf{V} \in \mathbf{R}^{2M+1}, \ \ \mathbf{f}(.) : \mathbf{R}^{2M+1} \mapsto \mathbf{R}^{2M+1}, \tag{7}$$

where $\mathbf{V} = [V_0, V_1, V_3, \cdots, V_{2M}, \omega]^T$. Applying the *Newton homotopy method* [9] to solve (7), we have the following relation;

$$\mathbf{F}(\mathbf{V}, \rho) = \mathbf{f}(\mathbf{V}) + (\rho - 1)\mathbf{f}(\mathbf{V}_0) = \mathbf{0}, \tag{8}$$

where ρ is an additional variable, and $\mathbf{V}_0$ is the initial guess. Then, the homotopy path[†] starting from $(\mathbf{V}_0, \rho = 0)$ satisfies the relations (6) at $\rho = 1$. In this way, we can find out the multiple solutions on the paths, whose curves can be efficiently traced by the application of the *arc-length method* as follows;

$$\left.\begin{array}{l} \mathbf{F}(\mathbf{V}, \rho) = \mathbf{0} \\ \displaystyle\sum_{\substack{i=1 \\ i \neq 2}}^{2M+1} \left(\frac{dV_i}{ds}\right)^2 + \left(\frac{d\rho}{ds}\right)^2 = 1 \end{array}\right\}, \tag{9}$$

where s denotes the arc-length from a starting point. Equation (9) is a set of nonlinear algebraic-differential equations, and their circuit models can be realized by the use of ABMs (analog behavior models) of Spice [12]. The circuit diagram is shown in Fig. 2(a) and (b), where "VCCS" is a voltage-controlled current source. The current sources $\hat{f}_i(\mathbf{V}, \rho)$ and I_s, I_ρ in Fig. 2 are given by[††]

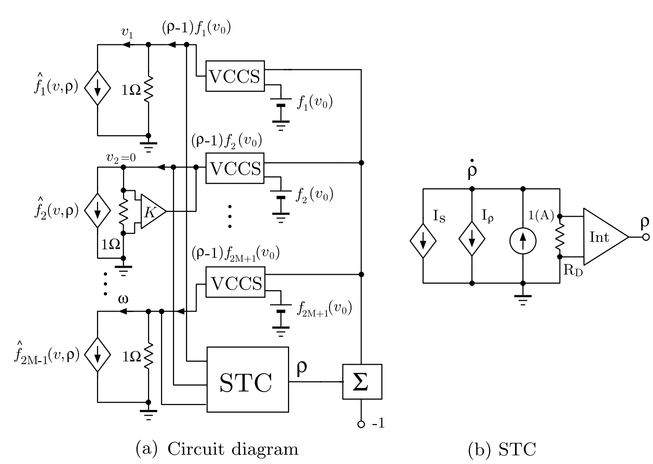

Fig. 2 Circuit diagram of the Newton homotopy method.

[†]The solution curve satisfies the relation (8).

[††]Spice [10] has many useful analog behavior models, so we can easily realize the current sources $\hat{f}_i(\mathbf{V}, \rho)$. In this circuit, $V_2 = 0$ is realized by setting $K = \infty$.

$$\left.\begin{array}{l}\hat{f}_i(\mathbf{V},\rho) = F_i(\mathbf{V},\rho) - V_i,\ \ i = 1,2,\ldots,2M+1,\\ \text{for}\ \ V_2 = 0\\ I_s = \displaystyle\sum_{\substack{i=1\\ i\neq 2}}^{2M+1}\left(\frac{dV_i}{ds}\right)^2,\ \ I_\rho = \left(\frac{d\rho}{ds}\right)^2.\end{array}\right\} \quad (10)$$

The second term of (10) is realized by STC (Solution Curve Tracing Circuit) as shown in Fig. 2(b), where R_D is a sufficiently large dummy resistance to avoid the L-J cut set. Furthermore, $f_1(\mathbf{V}_0), f_2(\mathbf{V}_0), \ldots, f_{2M+1}(\mathbf{V}_0)$ in Fig. 2(a) denote the initial guesses. Thus, we can easily trace the homotopy path starting from an arbitrarily chosen initial guess ($\mathbf{V}_0$). The circuit shown in Fig. 2 is solved by transient analysis of Spice, and the multiple oscillations are found at $\rho = 1$ on the homotopy paths.

3. Equivalent Circuit Models

3.1 Fourier Transfer Circuit Model

Oscillator circuits usually consist of many linear and/or nonlinear reactive and resistive elements. When the harmonic balance method is applied to the analysis, the determining equations are described by a set of algebraic equations as shown in Sect. 2. Firstly, we need to expand the responses of nonlinear elements into the Fourier series. Let us assume the input and output waveforms as follows;

$$\left.\begin{array}{l} v(t) = V_0 + \displaystyle\sum_{k=1}^{M}[V_{2k-1}\cos k\omega t + V_{2k}\sin k\omega t]\\ i(t) = I_0 + \displaystyle\sum_{k=1}^{M}[I_{2k-1}\cos k\omega t + I_{2k}\sin k\omega t]\end{array}\right\} \quad (11)$$

where M denotes the highest harmonic component to take account in the analysis. The output Fourier coefficients are described as follows;

$$\left.\begin{array}{l} I_0 = g_0(V_0, V_1, \cdots, V_{2M})\\ I_1 = g_1(V_0, V_1, \cdots, V_{2M})\\ \cdots\cdots\cdots\cdots\cdots\cdots\\ I_{2M} = g_{2M}(V_0, V_1, \cdots, V_{2M})\end{array}\right\} \quad (12)$$

In this case, these coefficients can be given by explicit forms only if the nonlinear characteristics are described by power series. For general nonlinear elements, we need to introduce some other techniques for the Fourier transformation. When the input-output relation is given by

$$i = g(v), \quad (13)$$

the Fourier coefficients are calculated by the following formulas;

$$\left.\begin{array}{l} I_0 = \dfrac{1}{T}\displaystyle\int_0^T g(v)dt\\ I_{2k-1} = \dfrac{2}{T}\displaystyle\int_0^T g(v)\cos k\omega t dt,\ \ I_{2k} = \dfrac{2}{T}\displaystyle\int_0^T g(v)\sin k\omega t dt\\ k = 1,2,\ldots,M\end{array}\right\}. \quad (14)$$

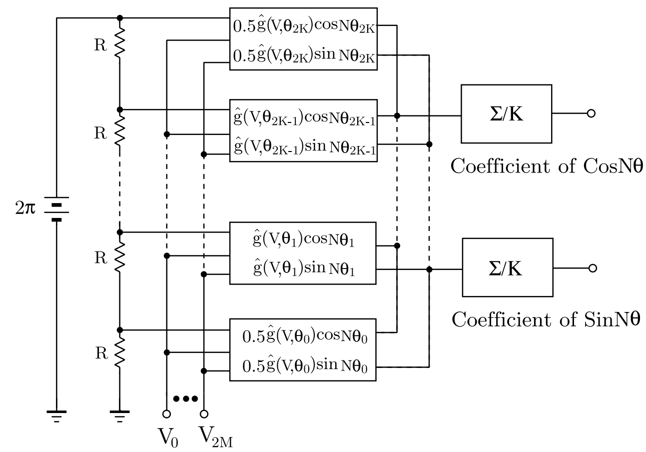

Fig. 3 Fourier transfer circuit model.

Now, let us apply the trapezoidal integration formula to (14) given as follow;

$$\int_a^b g(v)dt = \frac{h}{2}(g_0 + g_n) + h(g_1 + g_2 + \cdots + g_{n-1}), \quad (15)$$

where the step-size of the integration is $h = (a-b)/n$ for "n" divisions. Then, the truncation error is given by $g^{(2)}h^2/12n$. Using this formula, we can realize the equivalent circuit model satisfying (15) with the ABMs of Spice. To understand the circuit model, we assume that the input is given by (11) with $\theta = \omega t$. Then, the *Fourier transfer circuit model* for calculating the Nth higher harmonic component is shown by Fig. 3. $2K + 1$ blocks calculate the Nth components (I_{2N-1}, I_{2N}) in (14). On the other hand, the integration interval $[0, 2\pi]$ for $\theta = \omega t$ is divided by $2K$ sections using the resistors, so that the kth input voltage of the ABM blocks is given by kth term in (15) with $\omega t(= \theta_k = 2\pi k/2K)$ at the kth node, and the values (14) are calculated by the use of ABMs of Spice. Thus, the resultant outputs are given by $\{g(\mathbf{V}, \theta_k)\cos N\theta_k, g(\mathbf{V}, \theta_k)\sin N\theta_k, k = 0, 1, 2, \ldots, 2K\}$. Summing all of them and dividing by $2K$, the outputs are equal to the coefficients of $\cos N\theta$ and $\sin N\theta$, respectively.

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

$$I_N(x) = \frac{1}{2\pi}\int_{-\pi}^{\pi} e^{x\cos\theta}\cos N\theta d\theta. \quad (16)$$

The simulation results with $h = 2\pi/20$ are shown in Fig. 4. The value $I_1(10) = 2761$ for $N = 1,\ x = 10$ is exactly equal to the result from the Table of Bessel function [13]. *Remark* that Fourier expansions for exponential functions are very important to the analysis of the circuit containing diodes and bipolar transistors [12]. Next, we apply the transfer circuit to the Fourier expansion of *MOSFET*, whose characteristic in Spice model is described by a piecewise continuous functions [12] as follows:

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

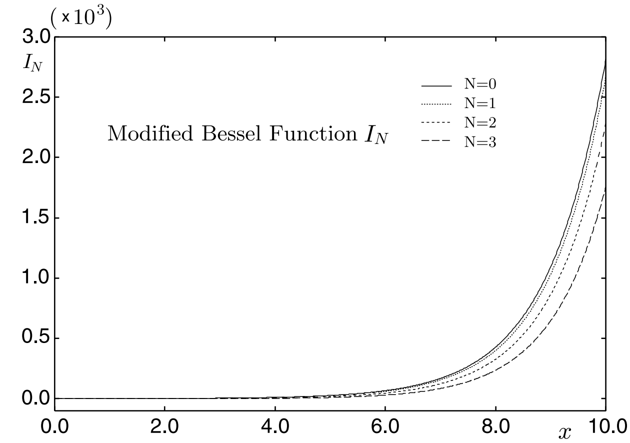

Fig. 4 Fourier transformation for modified Bessel function.

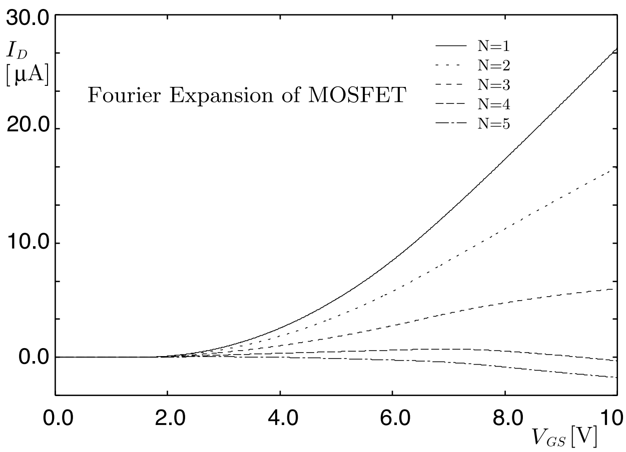

Fig. 5 Fourier transformation for MOSFET. V_{DS} = 3 [V], K = 3.87 [μA], λ = 0.01605, W = 2 [μm], L = 2 [μ], V_T = 0.827 [V].

$$I_D = \frac{KW}{L}\left[(V_{GS} - V_T) - \frac{V_{DS}}{2}\right]V_{DS} \\ (1 + \lambda V_{DS}). \tag{17a}$$

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

$$I_D = \frac{KW}{L}(V_{GS} - V_T)^2(1 + \lambda V_{DS}). \tag{17b}$$

The result of Fourier expansion for an input $V_{GS}\cos\omega t$ is shown in Fig. 5. Thus, the Fourier transfer circuit model given by Fig. 3 can be efficiently applied to any kind of circuit elements contained in analog ICs.

3.2 Fourier Expansion of Linear and Nonlinear Inductors

Let an inductor flux be the current-controlled as follow;

$$\phi_L(t) = \hat{\phi}(i_L) \tag{18}$$

If the characteristic is described by a power series, the Fourier coefficients will be described in explicit forms. Otherwise, we need to use the Fourier transfer circuit model given in Fig. 3. Thus, the Fourier expansion is described in the following form;

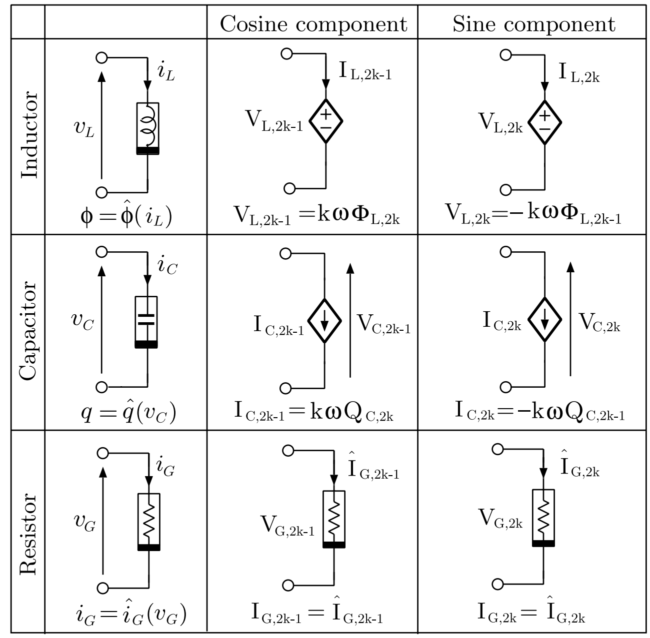

Fig. 6 Equivalent "Cosine" and "Sine" circuits.

$$\phi_L(t) = \Phi_{L,0} + \sum_{k=1}^{M}[\Phi_{L,2k-1}\cos k\omega t + \Phi_{L,2k}\sin k\omega t]. \tag{19a}$$

Differentiating $\phi_L(t)$, we have

$$v_L(t) = \sum_{k=1}^{M} k\omega\,[-\Phi_{L,2k-1}\sin k\omega t + \Phi_{L,2k}\cos k\omega t]. \tag{19b}$$

Thus, "Sine" and "Cosine" components for the kth higher harmonic components are respectively given by

$$V_{L,2k} = -k\omega\Phi_{L,2k-1},\ V_{L,2k-1} = k\omega\Phi_{L,2k}, \\ k = 1, 2, \ldots, K, \tag{20a}$$

Thus, the inductor is replaced by the coupled current-controlled voltage sources as shown in Fig. 6.
For a linear inductor, we have

$$v_L = L\frac{di}{dt} \\ \Rightarrow V_{L,2k} = -k\omega LI_{L,2k-1},\ \ V_{L,2k-1} = k\omega LI_{L,2k} \tag{20b}$$

3.3 Fourier Expansion of Linear and Nonlinear Capacitors

Let a capacitor charge be the voltage-controlled characteristic as follow;

$$q_C(t) = \hat{q}_C(v_C) \tag{21}$$

The Fourier expansion can be carried out with the same technique as the inductor.

$$q_C(t) = Q_{C,0} + \sum_{k=1}^{M}[Q_{C,2k-1}\cos k\omega t + Q_{C,2k}\sin k\omega t], \tag{22a}$$

Differentiating $q_C(t)$, we have

$$i_C(t) = \sum_{k=1}^{M} k\omega[-Q_{C,2k-1}\sin k\omega t + Q_{C,2k}\cos k\omega t], \quad (22b)$$

Thus, "Sine" and "Cosine" components for the kth high harmonic component are respectively given by

$$I_{C,2k} = -k\omega Q_{C,2k-1},\ I_{C,2k-1} = k\omega Q_{C,2k}, \quad k = 1, 2, \ldots, K, \quad (23a)$$

For a linear capacitor, we have

$$i_C = C\frac{dv}{dt} \Rightarrow I_{C,2k} = -k\omega C V_{C,2k-1}, \quad I_{C,2k-1} = k\omega C V_{C,2k} \quad (23b)$$

Thus, each capacitor is replaced by the coupled voltage-controlled current sources as shown in Fig. 6.

3.4 Equivalent Circuit Model of the Determining Equations

All the reactance elements in the linear reactance circuit and nonlinear resistors can be replaced by the equivalent circuit models in the table of Fig. 6. Thus, the reactance oscillator shown in Fig. 1 is transformed into the "Cosine" and "Sine" circuits as shown in Fig. 7, where we assume the voltage and current of nonlinear resistor as follows;

$$v(t) = V_0 + \sum_{k=0}^{M}[V_{2k-1}\cos k\omega t + V_{2k}\sin k\omega t] \quad (24a)$$

$$i(t) = I_0 + \sum_{k=0}^{M}[I_{2k-1}\cos k\omega t + I_{2k}\sin k\omega t] \quad (24b)$$

For the analysis of autonomous systems, the time origin can be arbitrarily chosen satisfying a relation "$V_2 \sin\omega t = 0$." The equivalent circuit model is shown by Fig. 7(a); namely,

$$V_2 - \omega = -\omega \quad (25)$$

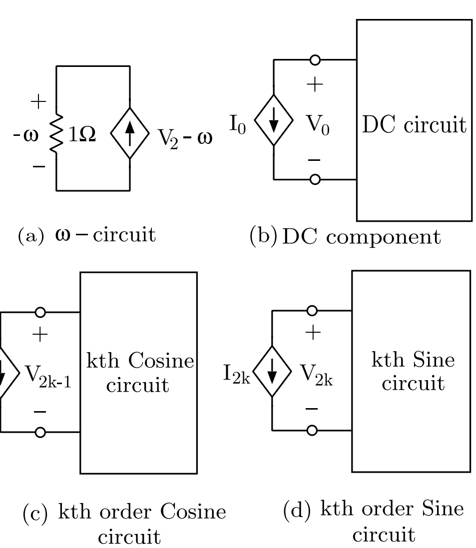

Fig. 7 Equivalent "Cosine" and "Sine" circuit models of the determining equations. $k = 1, 2, 3, \ldots, M$.

which means $V_2 = 0$. Although the equivalent circuit in Fig. 7 is composed of $2M + 2$ sub-circuits, one of them corresponds to the $V_2 \sin\omega t = 0$ or $V_2 = 0$. Therefore, the solution of the circuit decides the $2M + 1$ variables $\{\omega, V_0, V_1, V_3, \ldots, V_{2M-1}, V_{2M}\}$. The modified circuit shown in Fig. 7 can be efficiently solved by the Newton homotopy method shown in Sect. 2 and STC (Solution curve Tracing Circuit).

4. An Illustrative Example

Consider a reactance Cauer oscillator having a negative resistor as shown in Fig. 8, whose nonlinear characteristic is given by

$$i_G = -C_1 v_G + C_3 v_G^3,\ \ C_1 = 1.\ \ C_3 = 1. \quad (26)$$

Assume the resonant and ant-resonant frequencies as follows;

$$\text{Anti-resonant frequencies}: \omega_{10} = 1,\ \omega_{30} = 4,\ \omega_{50} = 6$$
$$\text{Resonant frequencies}: \ \omega_{20} = 2,\ \ \omega_{40} = 5 \quad (27)$$

Then, we have the circuit parameters as shown in Fig. 8, where we have added a small resistor ($r = 0.05$) to each inductor.

It is well-known from the experimental results [6], [14], [15] that, when a weakly nonlinear resistor (26) has a symmetric characteristic to the origin, the resultant waveforms will have only odd number of the higher harmonic components. Thus, we can set

$$v_G(t) = V_{1,1C}\cos\omega t + \sum_{k=1}^{M}[V_{2k+1,1C}\cos(2k+1)\omega t + V_{2k+1,1S}\sin(2k+1)\omega t], \quad (28)$$

where the term $V_{1,1S}\sin\omega t$ in (28) was deleted because of the autonomous system.

Now, we will develop "Cosine" and "Sine" circuits to Fig. 8 which correspond to the determining equations of the harmonic balance method.

In this case, for linear inductors, we have

$$V_{L,2k} = -k\omega L I_{L,2k-1},\ V_{L,2k-1} = k\omega L I_{L,2k}, \quad k = 1, 2, \ldots, M. \quad (29)$$

For linear capacitors, we have

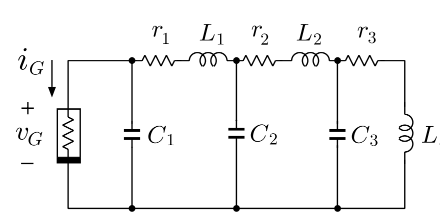

Fig. 8 Cauer oscillator. $C_1 = 0.1$, $C_2 = 0.343$, $C_3 = 0.439$, $L_1 = 0.417$, $L_2 = 0.262$, $L_3 = 1.058$, $r_1 = \cdots = r_3 = 0.01$.

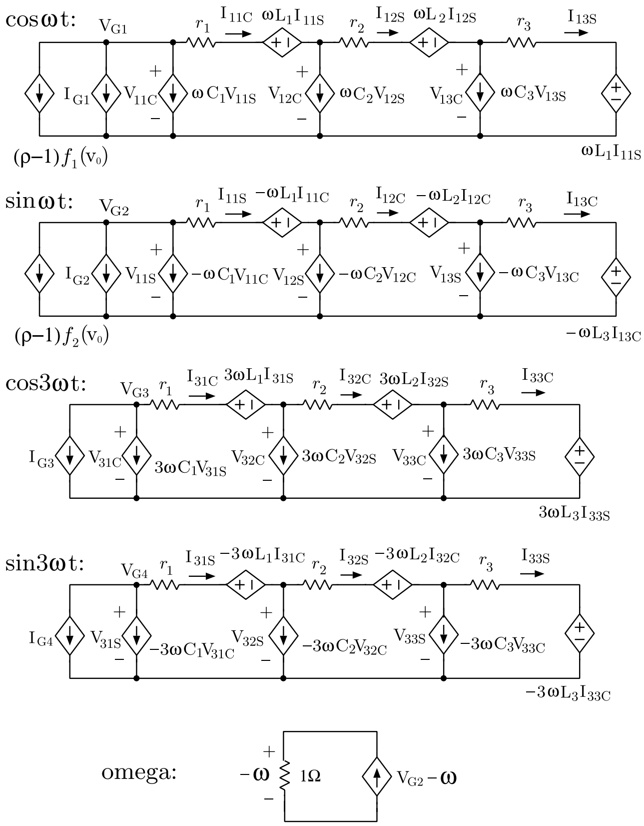

Fig. 9 Equivalent circuit describing the determining equations.

$$I_{C,2k} = -k\omega C V_{C,2k-1},\ I_{C,2k-1} = k\omega C V_{C,2k},\quad k = 1, 2, \ldots, M. \tag{30}$$

Thus, the inductors and capacitors in the modified "Cosine" and "Sine" circuits are replaced by the simple coupled linear current-controlled voltage sources, and coupled voltage-controlled current sources, respectively. Whole the circuit configuration until the 3rd higher harmonic components is shown in Fig. 9. In order to trace the multiple solutions, it also contains the Newton homotopy sub-circuits denoted by $\{(\rho - 1)f_i(\mathbf{V}_0), i = 1, 2\}$ in the first two sub-circuits. Observe that each of the "Cosine" and "Sine" circuits has the same topology as the original circuit shown in Fig. 8. It makes easier to construct the equivalent circuits with the schematic editor of Spice.

Combining STC (solution curve tracing circuit) shown in Fig. 2(b) with Fig. 9, we can trace the homotopy paths starting from $\{f_i(\mathbf{V}_0), i = 1, 2\}$, and the multiple solutions are found at $\rho = 1$. We have traced both sides of the paths by setting the initial conditions $\dot{\rho}(0) > 0$, and < 0 of STC. The results are shown in Fig. 10(a) for $s < 0$ and (b) for $s > 0$, where the solid line corresponds to ω-curve. Thus, we have found 6 solutions at $\rho = 1$ as follows:

Stable solutions : $\omega_1 = 1.0878,\ \omega_3 = 3.6770,\ \omega_5 = 5.722$
Unstable solutions : $\omega_0 = 0,\ \omega_2 = 2.0001,\ \omega_4 = 4.999$

Observe that the amplitudes of oscillations at resonant frequencies $\omega = 0,\ 2,\ 5$ are all the zeros. It means that the

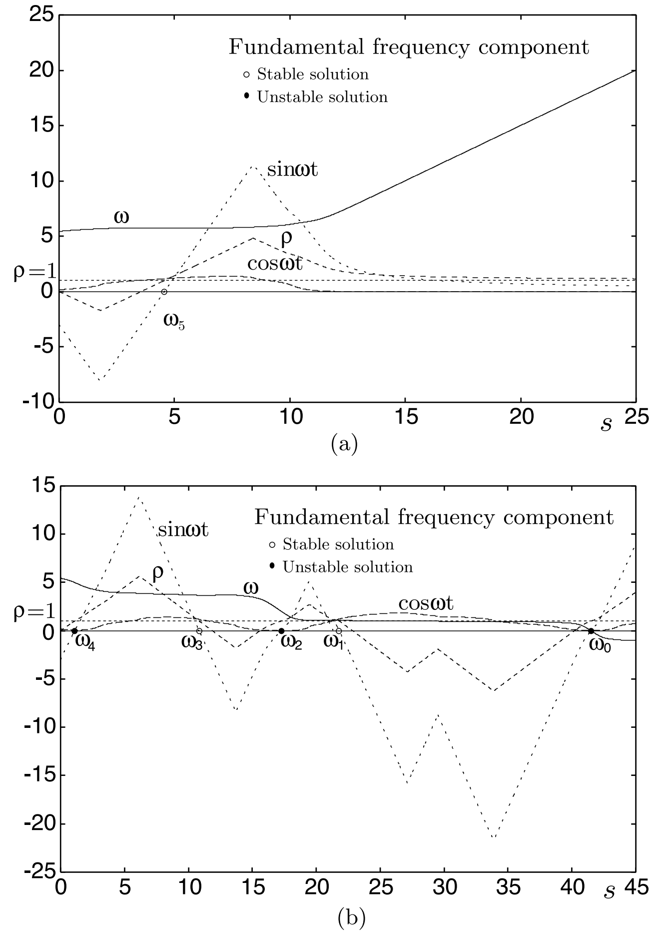

Fig. 10 Solution curves, (a) $s < 0$, (b) $s > 0$.

reactance oscillator does not oscillate at the resonant frequencies of the reactance circuit [8], and the non-zero oscillations will happen around the anti-resonant frequencies. We found from Fig. 10 that the oscillator has totally 6 solutions, and that those around the anti-resonant frequencies are stable† Next, we have calculated the steady-state waveforms starting from the initial conditions estimated by the above harmonic balance methods. The results are shown in Figs. 11(a), (b) and (c). The frequency spectra with FFT are shown in Figs. 12(a), (b) and (c), respectively. The waveforms at $\omega = 1.0556$ is largely distorted and contains many higher harmonic components.

We have confirmed from the results that, although the Cauer oscillator happens to oscillate around the anti-resonant frequencies of the reactance circuit, their frequen-

†Although we can arbitrarily choose the initial guesses, it may happen to trace all the solutions or some of them depending on the guesses. In this example, when we have traced both directions of solution curves of $s > 0,\ s < 0$ with $\omega_0 = 6$, most of our cases found all of the solutions on the homotopy paths. It is known that the curve tracing algorithm [9] is failed at the point of $\mathrm{rank}(\mathbf{F}(\mathbf{V})) < n$ for an $n + 1$ variables. However, it is impossible to know the geometries of curves in $n + 1$-dimensional space. Hence, to find all the solutions, we will recommend to choose many initial guesses in the space around the expected solutions.

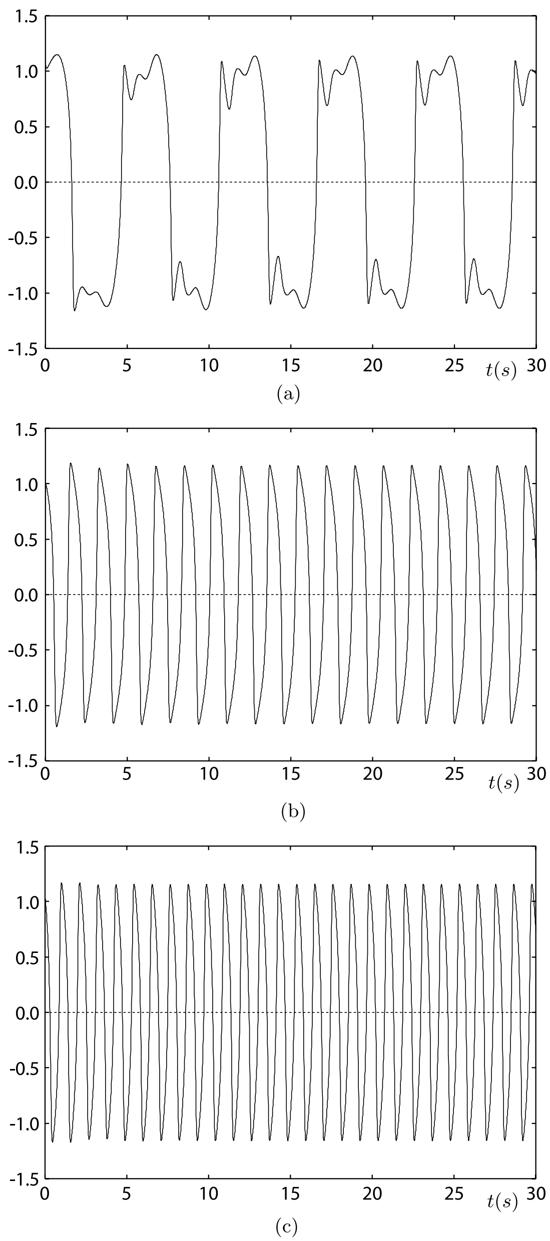

Fig. 11 The steady-state waveforms. (a) ω_1 = 1.0556, (b) ω_3 = 3.619, (c) ω_5 = 5.68.

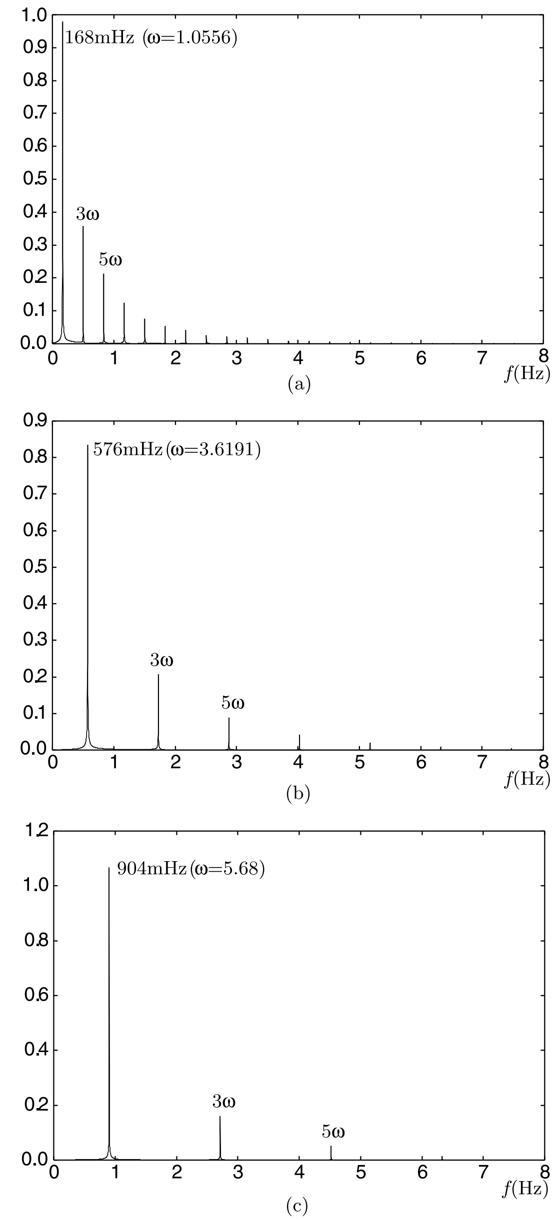

Fig. 12 Frequency spectrum for Fig. 11. (a) ω_1 = 1.0556, (b) ω_3 = 3.6191, (c) ω_5 = 5.68.

cies are little different from the designed frequencies because of the nonlinearity of the resistor.

5. Conclusions and Remarks

It is a difficult task to analyze oscillators having multi-mode oscillations, because each of the oscillations depends on the corresponding initial condition. In this paper, we have proposed a Spice-oriented harmonic balance method, where the determining equations can be described by the equivalent "DC," "Cosine" and "Sine" circuits. The modified circuits are solved by the Newton homotopy method using Spice. Thus, in our algorithm, we need not to derive any troublesome circuit equations, determining equations nor the Jacobian matrices. As an example, we considered a Cauer oscillator having multi-mode oscillations which is composed of a reactance Cauer circuit and a negative resistor. We found from the results that the oscillations happen to appear around the ant-resonant frequencies of the reactance circuit. The solution algorithm shown in this paper will be applied to any kind of reactance oscillators.

The Cauer oscillators sometimes happen quasi-periodic oscillations depending on the circuit parameters. We will also apply our algorithm to electronic circuits such as Hartley and Colpitis oscillators. These are the future problems.

References

[1] T. Endo and S. Mori, "Mode analysis of a multimode ladder oscillators," IEEE Trans. Circuits Syst., vol.CAS-23, no.2, pp.100–113, 1976.
[2] S. Moro, Y. Nishio, and S. Mori, "On coupled oscillators networks for cellular neural networks," IEICE Trans. Fundamentals, vol.E80-A, no.2, pp.214–222, Feb. 1997.
[3] Y. Nishio and A. Ushida, "Multimode chaos in two coupled chaotic oscillators with hard nonlinearities," IEICE Trans. Fundamentals, vol.E79-A, no.2, pp.227–232, Feb. 1996.
[4] K. Kundert, J. White, and A. Sangiovanni-Vincentelli, Steady-State Methods for Simulating Analog and Microwave Circuits, Kluwer Academic Publishers, Boston, 1990.
[5] T.J. Aprille, Jr. and T.N. Trick, "A computer algorithm to determine the steady-state response of nonlinear oscillators," IEEE Trans. Circuit Theory, vol.CT-19, no.4, pp.354–360, 1972.
[6] K. Matsuo, T. Matsuda, Y. Nishio, Y. Yamagami, and A. Ushida, "Steady-state analysis of reactance oscillators having multiple oscillations," Proc. ITC-CSCC 2000, vol.1, pp.203–206, 2000.
[7] W. Ma, L.J. Trajkovic, and K. Mayaram, "HOMSPICE: A homotopy-based circuit simulator for periodic steady-state analysis on oscillators," IEEE ISCAS 2002, vol.1, pp.645–648, Phenix, Arizona, 2002.
[8] R.K. Brayton and K. Moser, "A theory of nonlinear networks," I and II, Q. Appl. Math., vol.22, pp.1–33 and pp.81–104, 1964.
[9] 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. Comput.-Aided Des. Integr. Circuits Syst., vol.21, no.3, pp.337–348, 2002.
[10] MicroSim, PSpice A/D Circuit Analysis User Guide, MicroSim Co., 1995.
[11] A.H. Nayfeh and D.T. Mook, Nonlinear Oscillations, John Wiley & Sons, 1979.
[12] A. Ushida and M. Tanaka, Electronic Circuit's Simulations, Korona Pub., 2002.
[13] K.K. Clarke and D.T. Hess, Communication Circuits: Analysis and Design, Addison-Wesley, 1971.
[14] E. Ikeno and A. Ushida, "The arc-length method for computation of characteristic curves," IEEE Trans. Circuits Syst., vol.23, no.3, pp.181–183, 1976.
[15] Y. Ueda, Some Problems on the Theory of Nonlinear Oscillations, Nippon Printing, 1969.

Yoshihiro Yamagami graduated from the Department of Electrical Engineering, Anan Technical College, Tokushima, Japan, in 1982. He joined the Department of Electrical and Electronic Engineering of Tokushima University, Tokushima, Japan, in 1984. His research interests include computer-aided simulations of nonlinear circuits and high-speed interconnect circuits.

Yoshifumi Nishio received B.E., M.E., and Ph.D. degrees in electrical engineering from Keio University, Yokohama Japan, in 1988, 1990, and 1993, respectively. In 1993, he joined the Department of Electrical and Electronic Engineering at Tokushima University, Tokushima Japan, where he is currently an Associate Professor. From May 2000 he spent a year in the Laboratory of Nonlinear Systems (LANOS) at the Swiss Federal Institute of Technology Lausanne (EPFL) as a Visiting Professor. His research interests include analysis and application of chaos in electrical circuits, analysis of synchronization in coupled oscillators circuits, development of analyzing methods for nonlinear circuits and theory and application of cellular neural networks. He was the Chair of the IEEE CAS Technical Committee on Nonlinear Circuits and Systems (NCAS) during 2004–2005 and is currently the Secretary of the IEICE Research Society of Nonlinear Theory and its Applications (NOLTA). He is serving as an Associate Editor for the IEEE Transactions on Circuits and Systems - Part I, for the area of Theory and Systems. He is a member of the IEEE and the RISP.

Akio Ushida received the B.E. and M.E. degrees in Electrical Engineering from Tokushima University in 1961 and 1966, respectively, and the Dr. degree in Electrical Engineering from University of Osaka Prefecture in 1974. He was an Associate Professor from 1973 to 1980 at Tokushima University. From 1980 to 2003, he was a Professor in the Department of Electrical and Electronic Engineering at Tokushima University. Since 2003, he has been with the Department of Mechanical-Electronic Engineering, Tokushima Bunri University. From 1974 to 1975, he spent one year as a visiting scholar at the Department of Electrical Engineering and Computer Sciences at the University, Berkeley. His current research interests include numerical methods and computer-aided analysis of nonlinear circuits. Dr. Ushida is a member of the IEEE.