Frequency response analysis for switching converters in SPICE without averaging

Haifei Deng, Alex Q. Huang, Yan Ma

Center for Power Electronics Systems, Department of Electrical and Computer Engineering
Virginia Polytechnic Institute and State University, Blacksburg, VA 24060-0179, USA
Phone: +1(540) 231-7204, Fax: +1(540) 231-6390, Email: hdeng@vt.edu

Abstract— **Stability is the key requirement in designing switching converter ASIC. Achieving this requires an accurate loop gain design. Since DC-DC switching converters are time-varying system, traditional small signal analysis in SPICE cannot be directly used to simulate the loop gain of this kind of system. A new method--Periodic small signal analysis is proposed to analyze and simulate DC-DC switching converter inside a SPICE like simulator without the need for averaging [1]. This general method is suitable for any switching regulators. The results are accurate comparing with average modeling and experiment results even beyond half switching frequency. A general procedure to design loop gain is demonstrated in a peak current control buck.**

I. INTRODUCTION

Stability is the key requirement in designing switching converter ASIC. Achieving this requires an accurate loop gain design. Most numerical methods in SPICE are developed for linear circuit. In traditional AC analysis, SPICE simulator linearizes circuit around a DC point and calculates the small signal response of the circuit around that DC bias point. Since DC-DC switching converters are mixed-signal time-varying system, some circuit blocks in the converter like PWM modulator, driver, switches and diodes are non-linear and working in large signal mode, traditional small signal analysis in SPICE cannot be directly used to simulate the loop gain of this kind of system. This causes a lot of trouble in converter design, especially if one is doing power management IC design. In recent years, some methods were proposed to simulate DC-DC switching converters. James Groves proposed a periodic operating trajectory concept and developed a simulation tool for the frequency analysis of DC-DC switching converters [2]. In [3], Richard P.E. Tymerski developed "PWM switch" concept and proposed large signal and small signal model for "PWM switch". In [4], "software network analysizer" method is proposed using MATLAB and PSPICE to simulate loop gain by frequency sweep. SIMPLIS may be the best software on the market that can be used to simulate loop-gain of DC-DC converter directly. However, the model in SIMPLIS is piece-wise linear model. It doesn't work for SPICE model. None of these methods can be directly used for SPICE like simulator, which causes a lot of trouble for power management ASIC designers. The normal approach to design a DC-DC switching converter ASIC in industry is to do modeling first by MATHCAD or MATLAB, then design the transistor level circuit to match modeling results. Since a lot of ideal assumptions are made in modeling, modeling results may not be close to real circuit. For example, modeling can't take into account the delay in real circuit, which may cause some additional phase drop. If a method can be directly used to simulate the loop gain of switching converters by SPICE like simulator, ASIC designers can get accurate loop gain that takes care of all the delay and non-linear effect at transistor circuit level. In this paper, a new method—periodic analysis is proposed to simulate DC-DC switching converter directly inside a SPICE like simulator without the need for averaging. This general method is suitable for any switching regulators. The results are accurate comparing with average modeling and experiment results even at high frequency. A general procedure to design loop gain is demonstrated by the design of a peak current control buck.

II. PROPOSED PERIODIC METHOD FOR ANALYSIS AND SIMULATION OF DC-DC SWITCHING CONVERTERS

Fig.1 is a typical single loop PWM DC-DC converter. Most of the blocks in this system are linear except switches, PWM modulator and drivers. As shown in Fig. 2, if we take a closer look at the detail waveform at every node in this system, we will find that the whole system is working in a periodic steady state with some perturbation. V_{OUT}, V_C, V_G, I_L are the DC components. V_{out}, V_c, Vg, I_l are the real signals. If we choose the periodic steady state as a new "DC bias" point, then the system acts like a "linear" system. The whole system can be linearized around the periodic steady state point; the periodic small signal analysis can be used to get the frequency response of the switching converter system. That is the basic concept of "periodic method" proposed in this paper. Fig. 3 is the corresponding frequency spectrum of the signals at different nodes around the control loop. We can see that: PWM modulator works like a "phase" modulator except the "phase" is duty cycle. Driver works like a power amplifier. Switches work like a "phase" demodulator or a square wave sampler. If we consider PWM modulator, driver and PWM switch as one non-linear block, it looks like a "phase" modem. The DC gain of this block is V_g / V_p from V_c to V_{out} and $D(t)$ from V_g to V_{out} as shown in Fig 4. We call it "duty cycle modulator". If we compare "duty cycle modulator" with a mixer as shown in Fig. 4, we will see that they are quite similar. Now come to the point, is it possible to use the same analytic method as that used for mixer to analyze the "duty cycle modulator"? The answer is "yes". From Fig.3, we can also see that, although PWM modulator, driver and PWM switch will generate high frequency harmonics, only if these harmonics are higher than $1/2f_s$, otherwise there will be no aliasing and output low pass filter will filter them out finally. We only need to consider the fundamental frequency signal, i.e. "0" sideband signal in periodic small signal analysis when only loop gain is considered. The loop gain will be the products of "0" sideband small signal gain of every block around control loop. The same conclusion can be applied for any switching converter.

As a conclusion, based on periodic steady state "DC bias" and "duty cycle modulator" concepts, if we consider only the fundamental frequency component of the small signal gain and discard all of the harmonic sidebands in the control loop, we can use periodic analysis method to get the system loop gain directly in SPICE.

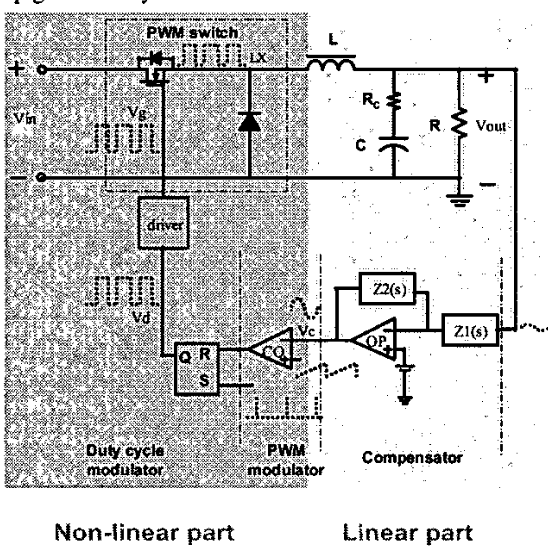


Fig. 1 single loop DC-DC buck converter

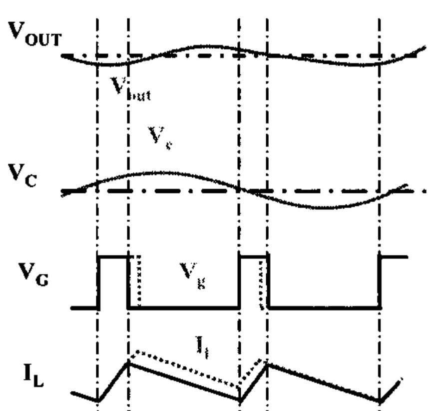


Fig. 2 periodic steady state and new "DC" bias concept

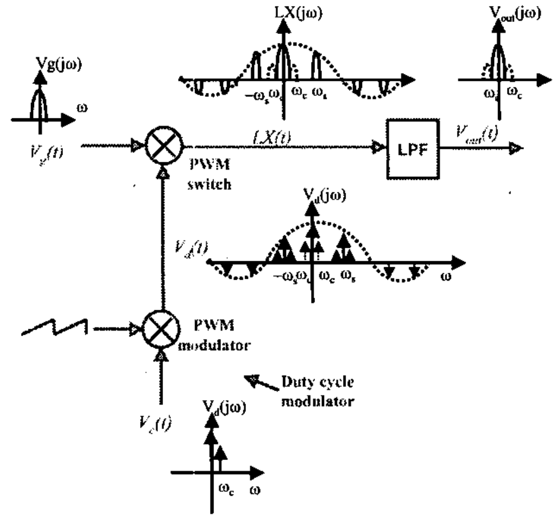
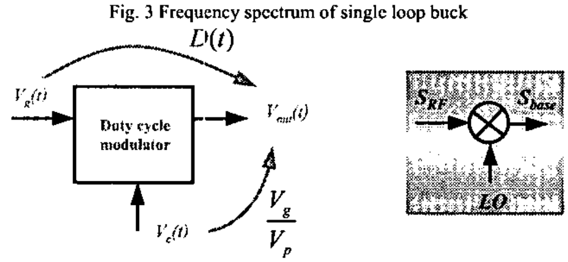


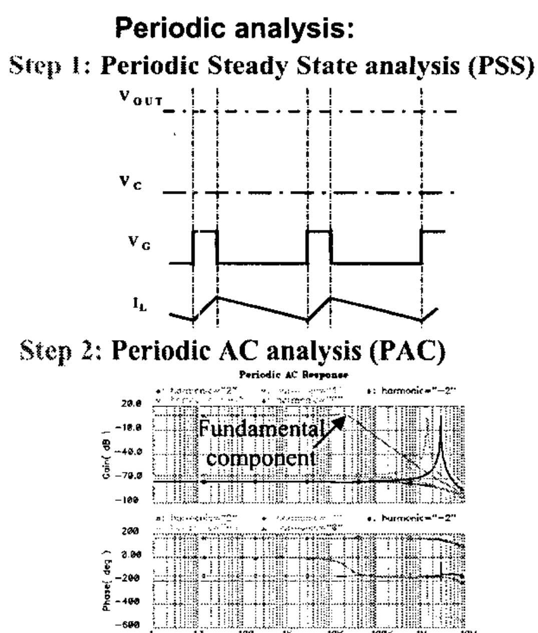
Fig. 3 Frequency spectrum of single loop buck



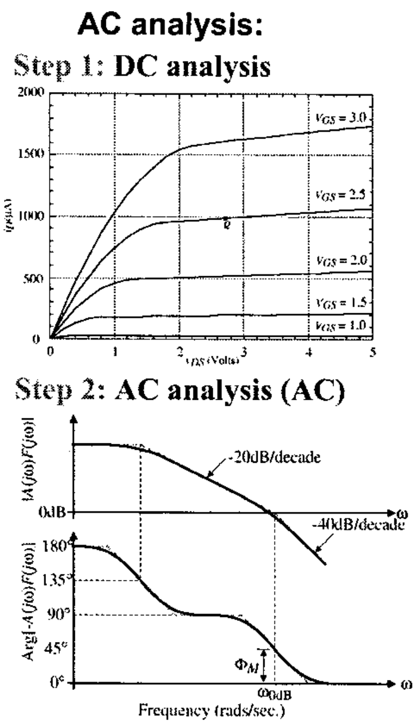
(a) Duty cycle modulator (b) Mixer

Fig. 4 Comparison between duty cycle modulator and mixer

III. ALGORITHMS FOR PERIODIC SMALL SIGNAL ANALYSIS



(a) Periodic analysis



(b) Traditional AC analysis

Fig. 5 Periodic analysis and traditional AC analysis

Fig. 5 is the comparison between periodic analysis and traditional AC analysis. Both of them are a two-step process. For traditional AC analysis, a DC analysis is needed first to build the operating points for all components like the Q point in Fig. 5 (b). After DC analysis, AC analysis will do the frequency sweep and calculate the small signal gain for the circuit at different frequency. The result of AC analysis is a bode plot with only one band. For periodic analysis, a PSS analysis and a PAC analysis are needed [5]. PSS analysis will calculate a periodic trajectory for the periodic system. PAC analysis will linearize the whole system around this periodic trajectory and calculate periodic small signal gain for different frequency. The result of PAC analysis is a plot with both fundamental frequency component and some sidebands. For loop gain analysis, only fundamental frequency component is needed.

In this section, the basic algorithm of PSS and PAC will be introduced briefly.

A. Periodic Steady State analysis

For nonautonomous system and autonomous system, the algorithms used for PSS analysis are different. Normally, nonautonomous system is easy to converge. In this paper, only nonautonomous system is discussed. Assume clock is generated by an independent source, a DC-DC switching system becomes a nonautonomous system. Three different methods can be used for PSS analysis: finite difference method, shooting method and harmonic balance method. The first two methods are time domain method. The last one is in frequency domain. Harmonic balance is suitable for mildly nonlinear circuits. Shooting methods are suitable for drastically nonlinear circuits [6]. Shooting method are chosen for DC-DC converter analysis. Consider a nonautonomous circuit whose equations are given by the standard form

$$\frac{dq(x(t))}{dt} + f(x(t)) + b(t) = 0 \qquad (1)$$

The independent sources are assumed to be periodic with period T. Since the circuit is nonautonomous, the circuit steady-state response $x(t)$ will also be periodic with period T. Now the problem becomes to obtain an initial condition $x(t_0)$ and optionally the trajectory $x(t)$ so that $x(t_0) = x(T)$. The solution trajectory can be viewed as a function of both time t and the initial condition $x(t_0)$, that is

$$x(t) = \phi(t, x(t_0))$$

Then the shoot equation can be written as

$$F_{sh} = \phi(T, x(t_0)) - x(t_0) = 0$$

This equation can be viewed as a nonlinear equation with *m (m is the size of the circuit)* variables $x(t_0)$ and therefore can be solved using Newton's method. The detail algorithm based on Krylov subspace method is out of the scope of this paper [7] [8] [9].

B. Periodic AC simulation

Now consider that a "small" input signal $D(x)\xi(t)$ is added to equation (1), i.e.,

$$\frac{dq(x(t))}{dt} + f(x(t)) + b(t) + D(x)\xi(t) = 0 \qquad (2)$$

Assume $x_s(t)$ is the steady-state T-periodic solution of this system and the solution of above equation is $x_s(t) + x_p(t)$ where $x_p(t)$ is small. Substituting this form into (2), we have

$$0 = \frac{d\left(\left.\frac{\partial q}{\partial x}\right|_{x_s} \cdot x_p(t)\right)}{dt} + \left.\frac{\partial f}{\partial x}\right|_{x_s} \cdot x_p(t) + D(x_s(t))\xi(t)$$

All of the coefficients in above equation are T-periodic. This equation can be solved in time domain or frequency domain. The detail algorithm based on Krylov subspace method is not the focus of this paper.

IV. Experimental Verification

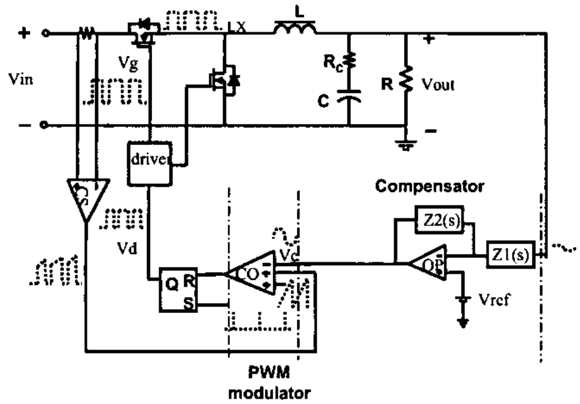


Fig 6. Block diagram of experimental structure

TABLE I
PARAMETERS FOR THE EXPERIMENTAL STRUCTURE

V_{in} (V)	V_{out} (V)	I_{out} (A)	C_{out} (uF)	L (uH)	R_C (Ω)	V_P (V)	R_i (Ω)	S_e	f_s (Hz)
5	1.5	0.2	10	4.7	5m	1	0.22	23k	11.16k

In this section, we will apply periodic analysis method to design a typical peak current control buck DC-DC switching converter. In design, only "0" sideband is considered and all other sidebands are discarded since they have no effect on final system loop gain if only aliasing effect doesn't happen. We divide the whole design procedure into three parts: Power stage design, current loop design and compensator design.

Fig. 6 is the simplified block diagram of the experimental structure. A lot of factors affect power stage design including the real application, performance requirement etc. Assume that all of the factors are considered and we got the power stage shown in Table I. V_p is the peak-to-peak magnitude of the ramp signal. In order to design system loop gain, we need to know the control to output transfer function for the power stage since it is part of the control loop.

Since $D(t)$ is not an electrical variable that can be tested, in the simulation, $\partial V_{out} / \partial V_c$ is simulated instead of $\partial V_{out} / \partial d$. Fig. 7 shows the simulation result, average model result for control to output transfer function $\partial V_{out} / \partial V_c$ of the power stage. From the results, we can see that simulation result is very close to average model result except that the DC gain and Q value are lower in simulation results. The reason is that some of parasitic components are not included in average model! This difference will directly affect the final loop gain.

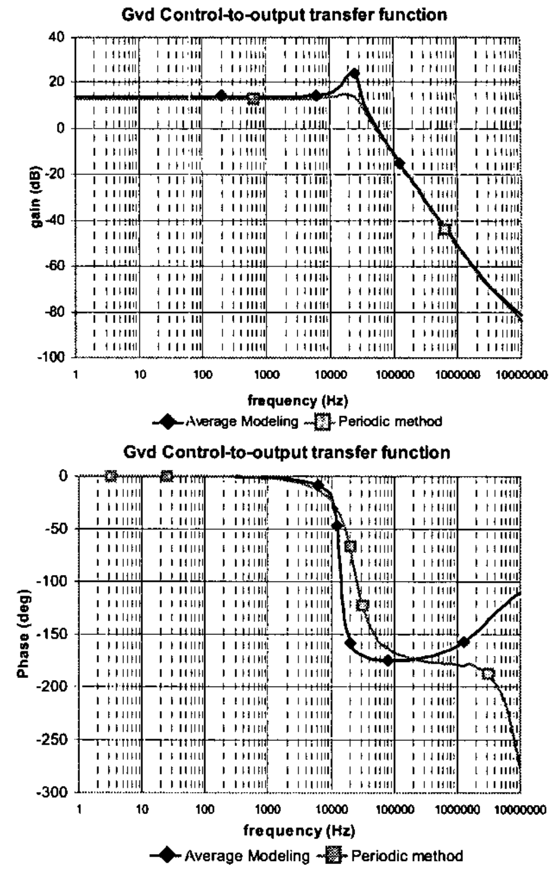


Fig. 7 Comparison of control to output transfer function for buck converter between periodic analysis and average model

After power stage design is done, we need to design current loop gain. With current loop closed, the system is degraded to one-order system. The design variables for current loop are current sensor gain R_i, external ramp slope S_e. R_i will affect the DC gain of control-to-output transfer function with current loop closed. External ramp slope will stabilize the system when duty cycle is larger than 50% and damp the peak at $1/2f_s$. By periodic small signal analysis, we can tune the value of R_i and external ramp slope and finally get the control-to-output transfer functions G_{oc} with current loop closed shown in Fig 8 for simulation and average modeling based on Dr. Ridley's model [10].

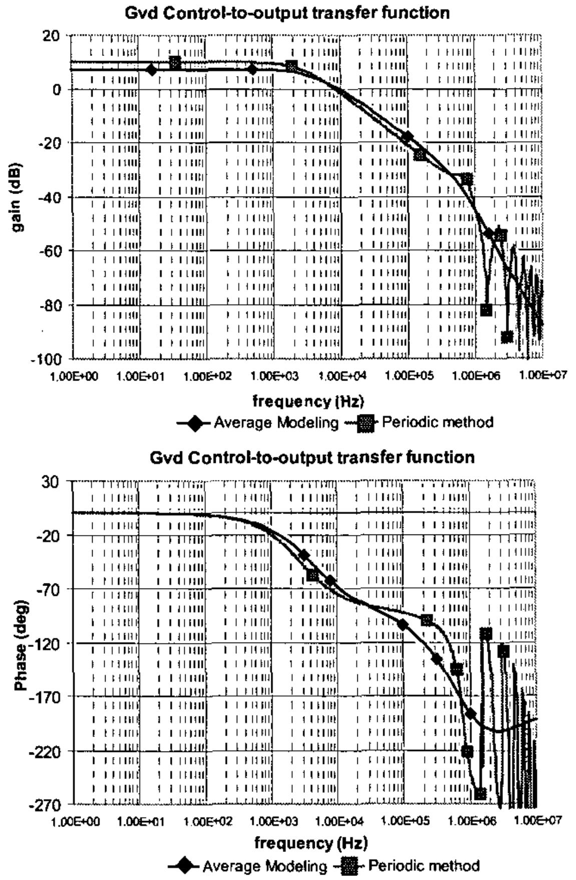
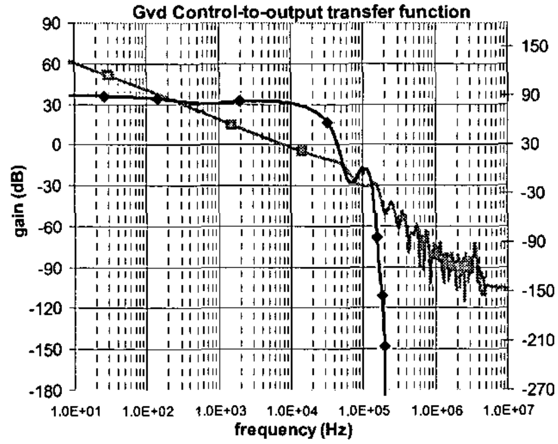


Fig 8 Comparison of control to output transfer function with current loop closed for buck converter between Dr. Ridley's model and periodic analysis

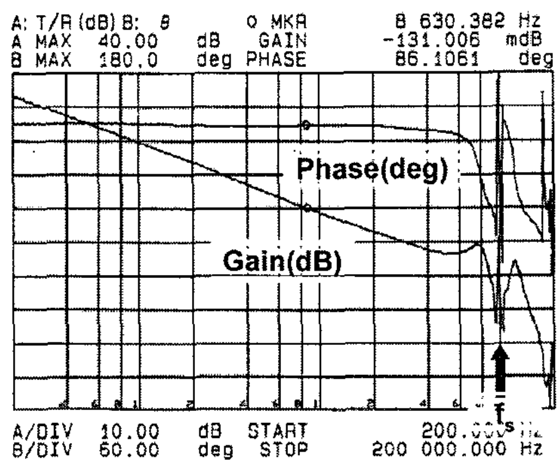
Since compensator is linear circuit, it is not difficult to get the transfer function by traditional AC analysis. Normally, we use two-pole-one-zero network for the compensator. By doing AC analysis, we can tune the position of zero and poles to get the desired loop gain. Combine compensator with the control-to-output transfer functions $\partial V_{out} / \partial V_c$ with current loop closed discussed above, we will get the open loop gain for the whole system, which is

$$T = A(s) \cdot \frac{\partial V_{out}}{\partial V_c}$$

Fig 9 shows the transfer function by periodic small signal analysis and real test results. We can see that simulation results are very close to test results. Figure 10 shows the experiment results of power on start up and the waveforms for a typical application.



(a) Simulated open loop gain of system loop



(b) Tested open loop gain of system loop

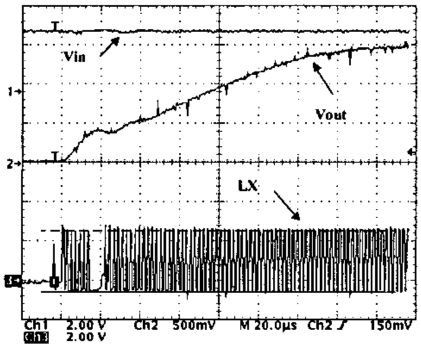
Fig 9 Comparison between experiment results and simulation results

V. CONCLUSION

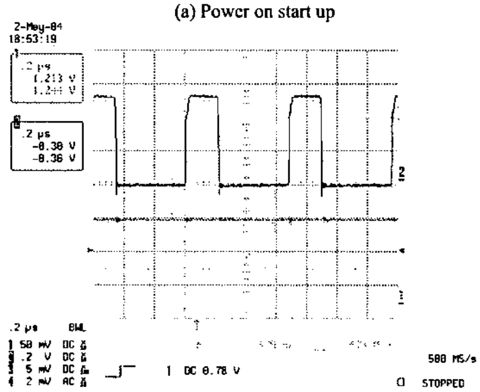
In this paper, a new method--Periodic small signal analysis is proposed to simulate switching converter directly inside a SPICE like simulator without the need for averaging. The basic concept and algorithm are discussed. The results are accurate comparing with average modeling and experiment results.

ACKNOWLEDGMENT

The authors would like to thank Ken Saller from national semiconductor for the support of this work.



(a) Power on start up



(b) PWM steady state waveform

Fig. 10 Tested results for experiment buck (Vin=3, Vout=1.5, Iload = 200mA)

REFERENCE:

[1] S. Cuk and R. D. Middlebrook, "A General Unified Approach to Modelling Switching DC-to-DC Converters in Discontinuous Conduction Mode," *IEEE Power Electronics Specialists Conference Record*, 1977.

[2] James Groves,"Small Signal Analysis of Nonlinear Systems with Periodic Operating Trajectories", Ph.D Dissertation,Virginia Tech, 04 1995

[3] Richard P.E. Tymerski., "Topology and analysis in power conversion and inversion", Ph. D Dissertation, Virginia Tech, April 22, 1988

[4] Jason H. Ly, Kasemsan Siri., "Frequency Response Analysis for DC-DC Converters Without Small-Signal Linearization", APEC 2003, P1008-1013

[5] Toshiji Kato, Wataru Tachibana., "Periodic Steady-State Analysis of an Autonomous Power Electronic System by a Modified Shooting Method", IEEE TRANSACTIONS ON POWER ELECTRONICS, VOL. 13, NO. 3, MAY 1998 P522-527

[6] Kenneth S. Kundert, "Introduction to RF Simulation and Its Application", IEEE JOURNAL OF SOLID-STATE CIRCUITS, VOL. 34, NO. 9, SEPTEMBER 1999 P1298-1319

[7] Jacob W. van der Woude, Willem L. de Koning, and Yusuf Fuad, "On the Periodic Behavior of PWM DC-DC Converters" IEEE TRANSACTIONS ON POWER ELECTRONICS, VOL. 17, NO. 4, JULY 2002, P585-595

[8] Leon O. Chua, Pen-Min Lin, "Computer-Aided analysis of electronic circuits" Prentice-hall, Inc., Englewood Cliffs, New Jersey, 1975

[9] Vatché Vorpérian, "Fast analytical techniques for electrical and electronic circuits" Cambridge University press, Cambridge, 2002

[10] R.B. Ridley, "A New Small-Signal Model for Current-Mode Control", PhD Dissertation, Virginia Polytechnic Institute and State University, November, 1990