A Software Tool for Learning the Dynamic Behavior of Power Electronics Circuits

Kwok-Tong Chau, *Member, IEEE*

Abstract—**A new software tool for learning the dynamic behavior of power electronics circuits is developed for undergraduate students. This tool incorporates the merits of two well-known software packages, namely, the realistic time-domain simulation of PSpice and the parametric identification process of MatLab. Hence, without going through complicated mathematics, the students can easily obtain the transient response, transfer function, and frequency response of power electronics circuits. The software tool is exemplified using a buck dc-dc converter. Its accuracy is verified by comparing the simulation result with those obtained by the state-space averaging technique and the experimental measurement.**

I. Introduction

THE study of dynamic behavior using both time-domain and frequency-domain simulations has been a major research area in power electronics. The time-domain simulation provides the transient behavior of power electronics circuits such that the maximum overshoot, rise time, settling time, etc. can be evaluated. On the other hand, the frequency-domain simulation provides a useful tool to assess the local stability and to design the feedback control loop of power electronics circuits. In order to carry out the time-domain simulation, the corresponding circuit can be represented by a modified nodal equation [1]. After solving this equation symbolically, the time-domain transient response can be determined progressively according to the switching sequences and threshold conditions. For the frequency-domain simulation, the corresponding circuit is usually divided into a series of topological stages. Each stage is described by a state-space equation. By using the state-space averaging technique, the small-signal transfer function can be determined [2]. Hence, the corresponding frequency response can be simulated.

Although it is widely accepted that the dynamic behavior of power electronics circuits should be introduced to the undergraduate students, it suffers from the problem that undergraduate power electronics curriculum involve substantial mathematics. As a result, some other basic contents may be sacrificed, which is why we aim to provide an introduction to various power electronics circuits, including dc-dc converters, ac voltage controllers, controlled rectifiers, cyclo converters, and inverters. Thus, it is my intention that the students should be able to learn the dynamic behavior of power electronics circuits without going through complicated mathematics.

PSpice is one of the most popular circuit simulators [3]. It is very suitable for the undergraduate students to perform the simulation of both analogue and digital electronics circuits because it is user-friendly and incorporates the realistic model of most electronics components. Increasingly, it provides built-in commands to study their dynamic behavior. Recently, it has also been used to study the time-domain steady-state behavior of power electronics circuits, hence aiding in the teaching of their principles of operation [4]. However, due to the switching nature of power electronics circuits, the use of PSpice to study their dynamic behavior encounters two major problems: the dc analyses available in PSpice are ill-suited to determine the steady-state operating point, and the frequency response cannot be obtained using the built-in ac analysis [5].

MatLab is a well-known software package for computer simulation [6] that is also commonly used for teaching and learning in undergraduate courses. It consists of a series of standard routines and software toolboxes, such as the signal processing, control system, and system identification, which enable the students to perform the simulation efficiently. In order to study the dynamic behavior of power electronics circuits using MatLab, the circuit needs to be idealized and its operating principles, such as the switching sequences and threshold conditions, must be well defined. Moreover, unless the students are familiar with the circuit-oriented modeling approach, they cannot directly learn the dynamic behavior of power electronics circuits using MatLab.

System identification deals with the problem of building mathematical models of dynamic systems, based on observed data. This area has matured into an established collection of basic techniques that are well understood and known to perform successfully in practical applications [7]. However, the application of system identification in power electronics is surprisingly rare. As compared with the circuit-oriented modeling approach, the modeling of power electronics circuits using system identification possesses a definite advantage of simplicity, which is particularly important for undergraduate students.

It is the purpose of this paper to present a new software tool for the undergraduate students to learn the dynamic behavior of power electronics circuits. The software tool incorporates the realistic time-domain simulation provided by PSpice as well as the parametric identification and frequency-domain simulation provided by MatLab. The power of the proposed tool lies in the ability to study the dynamic behavior of power electronics circuits in the absence of complicated mathematics.

Manuscript received February 7, 1994; revised July 26, 1994. This work was supported by the Hong Kong Polytechnic under Research Grant 0340.744.A3.410.

The author is with the Department of Electrical and Electronic Engineering, The University of Hong Kong, Hong Kong.

Publisher Item Identifier S 0018-9359(96)02131-0.

It should be noted that this paper has no intention of diminishing the importance of analytical skills that have been widely used for learning the dynamic behavior of power electronics circuits. In fact, it provides an alternative to learn the dynamics by combining the time-domain simulation and parametric identification.

First, the time-domain simulation using PSpice is described in Section II. Secondly, the basic theory of parametric identification is described in Section III. Section IV is devoted to instructing how to obtain the frequency-domain simulation from the time-domain simulation using MatLab. Finally, an application example with verification is given in Section V.

II. Time-Domain Simulation

The time-domain simulation of power electronics circuits is an important tool to assess the dynamic behavior. During the simulation, observing the voltage and current waveforms in a circuit can provide an insight that is not always possible with pencil and paper analysis. Although numerous approaches for the time-domain simulation of power electronics circuits have been proposed [1], the use of PSpice is a natural choice for the undergraduate students.

PSpice uses the same algorithms as SPICE2, which is a standard tool in the industry for circuit simulations. The input syntax for PSpice is a free-format style that does not require data to be entered in fixed column locations. A circuit is described by statements that are stored in a file. This circuit file can be created using the text editor contained within the control shell of PSpice, or any text editor that creates a text file without any embedded control characters. Each statement in the circuit file is self-contained and has no interaction with the others. PSpice also performs a considerable amount of error checking to ensure that the circuit has been entered correctly. Therefore, the students just need to spend a short time to create the circuit file of a power electronics circuit.

The time-domain simulation of the proposed software tool consists of two parts. Both are based on the transient analysis using PSpice. First, the start-up transient response is simulated until the steady-state is reached. Second, based on the resulting steady-state operating condition, the time-domain response due to an external perturbation is simulated.

Having performed the start-up transient simulation using the .TRAN command, the transient behavior of circuit voltages and currents, such as the maximum overshoot, rise time and settling time, can be obtained. It should be noted that the instantaneous overvoltage and overcurrent may deteriorate the power semiconductor devices of the power electronics circuit. Since the transient response generally extends over many switching cycles, the simulation usually involves a massive database. As the .PROBE command is limited to displaying 8000 data points, the resulting database is printed as a table in the output file using the .PRINT command. Then, the database is plotted by MatLab because it does not suffer from the conventional memory barrier and variable size restriction.

When the steady-state is reached, an additional small random signal is purposely injected to superimpose on the excitation of the circuit. The excitation may be the duty ratio or the supply voltage, respectively, for determining the control-to-output response and the supply-to-output response. Although PSpice does not have a built-in way to generate a random signal in the time-domain, the random signal can be easily simulated by using the piecewise linear (PWL) voltage source model. This PWL source model, which consists of a list of random numbers at specific instants of time, can be easily created using the random number generator available in MatLab. During the simulation, the input excitation and output voltage are recorded according to the pre-defined sampling interval. The resulting database is then printed as a table form in the output file, which will be directly used for parametric identification.

III. Parametric Identification

In this Section, only the basic theory of parametric identification being employed by the proposed software tool is described. A comprehensive description about the theory of system identification can be found in [7].

In general, the input-output configuration of a circuit can be described as

$$y(t) = G(q)u(t) + H(q)e(t) \tag{1}$$

with

$$G(q) = \sum_{k=1}^{\infty} g(k)q^{-k} \tag{2}$$

$$H(q) = 1 + \sum_{k=1}^{\infty} h(k)q^{-k} \tag{3}$$

where $u(t)$, $y(t)$, and $e(t)$ are the input, output, and unmeasurable noise of the circuit. The functions $G(q)$ and $H(q)$ are to be parameterized, most often as rational functions in the delay operator q^{-1}. Thus, the circuit description can be given by a generalized model structure

$$A(q)y(t) = \frac{B(q)}{F(q)}u(t-nk) + \frac{C(q)}{D(q)}e(t) \tag{4}$$

with $A(q)$, $B(q)$, $C(q)$, $D(q)$, and $F(q)$ the polynomials in the delay operator q^{-1}

$$A(q) = 1 + a_1 q^{-1} + \cdots + a_{na} q^{-na} \tag{5}$$

$$B(q) = b_1 + b_2 q^{-1} + \cdots + b_{nb} q^{-nb+1} \tag{6}$$

$$C(q) = 1 + c_1 q^{-1} + \cdots + c_{nc} q^{-nc} \tag{7}$$

$$D(q) = 1 + d_1 q^{-1} + \cdots + d_{nd} q^{-nd} \tag{8}$$

$$F(q) = 1 + f_1 q^{-1} + \cdots + f_{nf} q^{-nf} \tag{9}$$

where na, nb, nc, nd, and nf are the orders of the respective polynomials, and nk is the number of delays from input to output.

The model structure given by (4) is too general for most practical purposes. When $nd = nf = 0$, the corresponding model is so-called the autoregressive moving average with exogenous input (ARMAX) realization. In fact, the ARMAX model has become a standard tool for system identification.

Having observed the input-output data, the most common parametric identification method is to determine G and H by minimizing the loss function

$$V_N(G,H) = \sum_{t=1}^{N} e^2(t) \tag{10}$$

where N is the length of the data record. Ideally, the structure that has the smallest loss function should be selected. However, if the model is validated on the same data set that it was estimated from, the fit will always improve as the model structure increases. In deal with this circumstance, a technique, the so-called the Akaike's final prediction error (FPE) criterion, is adopted to compensate for the automatic decrease of the loss function. The FPE is formed as

$$\text{FPE} = \left(\frac{1+n/N}{1-n/N}\right) V_N \tag{11}$$

where n is the total number of estimated parameters. The theory says that in a collection of different models, the one with the smallest FPE should be chosen.

It should be noted that the above theory has been implemented in the system identification toolbox of MatLab [8]. Although the students can easily handle the identification process without going through any mathematical derivation, they are not encouraged to use the tool as a pure "black-box" approach. The students should atleast understand the basic principle of parametric identification described in this section. In fact, a deep understanding of basic principles can facilitate the convergence rate of identification by selecting good initial values for the iterative search.

Moreover, since the proposed approach possesses the advantages of minimum knowledge of circuit operation and free from circuit idealization, it is a powerful tool for circuit design. However, these advantages may not be too beneficial to the undergraduate students because an understanding of circuit operation and idealizing approximations is particularly helpful to them.

IV. Frequency-Domain Simulation

Having sampled the input–output data during the time-domain simulation using PSpice, the procedure to obtain the frequency-domain simulation from the time-domain simulation using MatLab is described in this section.

Before the identification process, the levels in the raw input and output sequences should be subtracted. Otherwise, the models will waste some parameters in order to correct the levels. With the *detrend* command, the levels or trends can be removed. Moreover, as the frequency bands below the switching frequency are particularly interested, filtering the data before the identification process can improve the fit in the interesting region. With the *filter* command, both FIR and IIR digital filters can be employed.

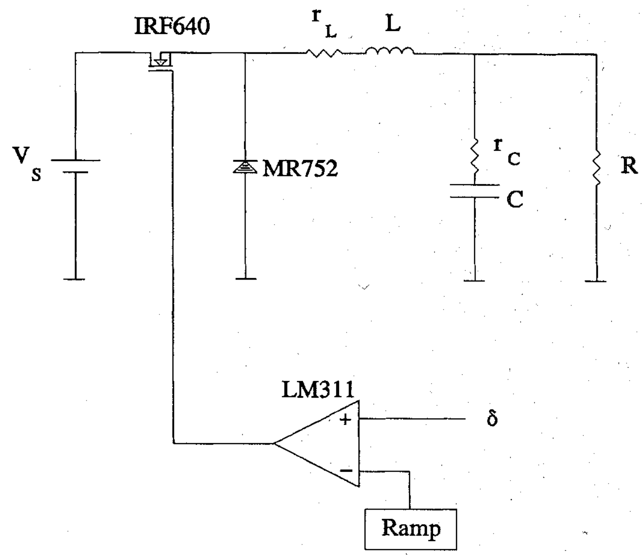


Fig. 1. Buck dc-dc converter.

With the *armax* command, an ARMAX model is fitted to the data set for a particular model structure. By varying the model structures with different orders and delays, the structure that minimizes the FPE is selected. The selection of this optimal structure can be performed automatically using the *selstruc* command. As the resulting parametric model is in the time-domain format, the corresponding transfer function can simply be determined using the *trf* command. Hence, with the *zpplot* and *bodeplot* commands, the pole-zero location and Bode diagrams can be plotted automatically.

V. Application Example

A. Simulation

A buck dc-dc converter operating at the continuous conduction mode is used for exemplification. As shown in Fig. 1, the power converter consists of a dc supply V_s, a power MOSFET IRF640, a diode MR752, an inductor L with an equivalent series resistor r_L, a capacitor C with an equivalent series resistor r_C, and an output resistor R. The values of the circuit components are $V_s = 20$ V, $L = 500$ μH, $r_L = 1.5$ Ω, $C = 100$ μF, $r_C = 0.3$ Ω, and $R = 10$ Ω. In this power converter, the switching frequency f_s is kept constant at 50 kHz. The output voltage is controlled by adjusting the turn-on period of the power MOSFET, the so-called duty-ratio control or pulsewidth modulation (PWM). This control signal is the output of a comparator LM311, which is produced by comparing the ramp generator's sawtooth signal ψ and the duty-ratio δ. The peak value of ψ is 1 V, and δ is 0.5. Initially, both inductor current i_L and capacitor voltage v_C are zero.

The corresponding PSpice circuit file can be easily created with any text editor, as long as it creates a text file without any embedded control characters. The models of the power MOSFET IRF640, diode MR752, and comparator LM311

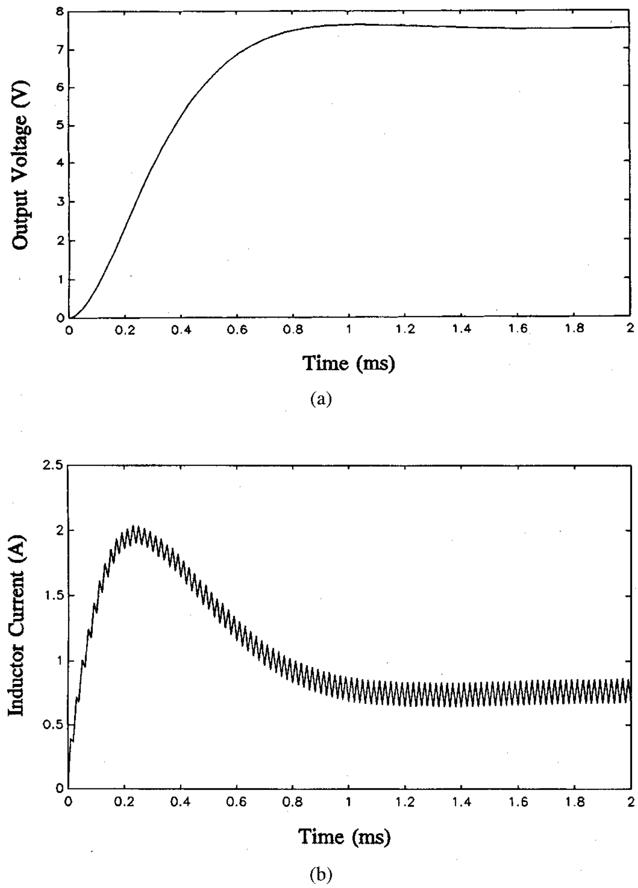


Fig. 2. Start-up transient waveforms. (a) Output voltage. (b) Inductor current.

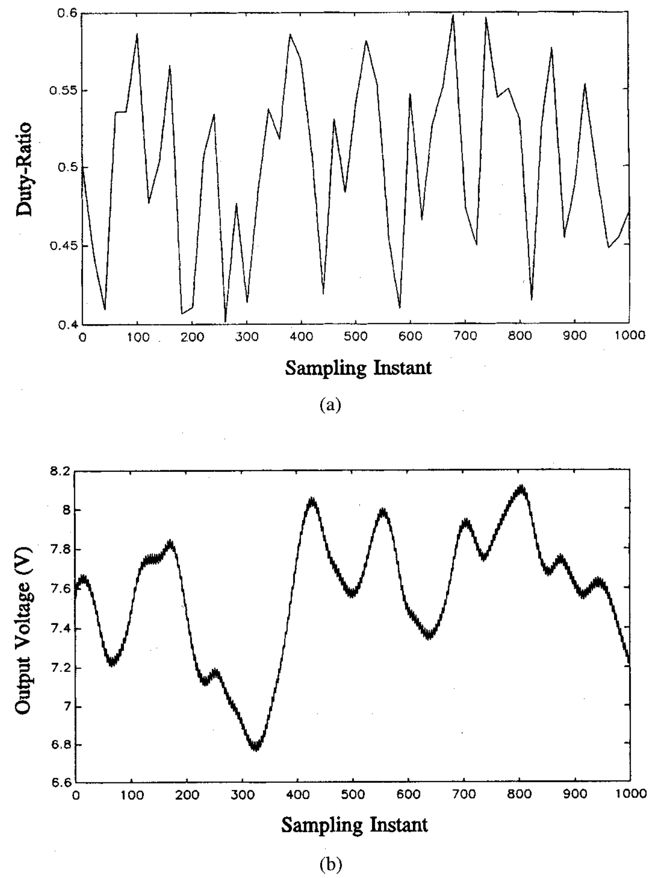


Fig. 3. Control-to-output time-domain response. (a) Duty-ratio. (b) Output voltage.

can be obtained from the PSpice libraries PWRMOS.LIB, DIODE.LIB, and LINEAR.LIB, respectively. The circuit file for the start-up transient time-domain simulation is listed in the Appendix.

Having performed the start-up transient simulation, the steady-state operating condition can be obtained, namely $i_L = 0.8595$ A and $v_C = 7.521$ V. The resulting transient waveforms are shown in Fig. 2. It can be found that there is a dangerous current overshoot that may deteriorate the power semiconductor devices. Moreover, the settling time is about 1 ms, which is equivalent to about 50 switching cycles.

In order to obtain the control-to-output transfer function $G_{cz}(z)$ and its frequency response, the duty-ratio is represented by a small random signal that is uniformly distributed around 0.5. The resulting output voltage and duty-ratio are recorded as shown in Fig. 3. Making use of this set of data, the control-to-output transfer function can be expressed as

$$G_{cz}(z) = \frac{0.0979 - 0.0534z^{-1}}{1 - 1.9093z^{-1} + 0.9120z^{-2}}. \tag{12}$$

Hence, the corresponding pole-zero location and frequency response can be obtained, which are shown in Figs. 4 and 5, respectively.

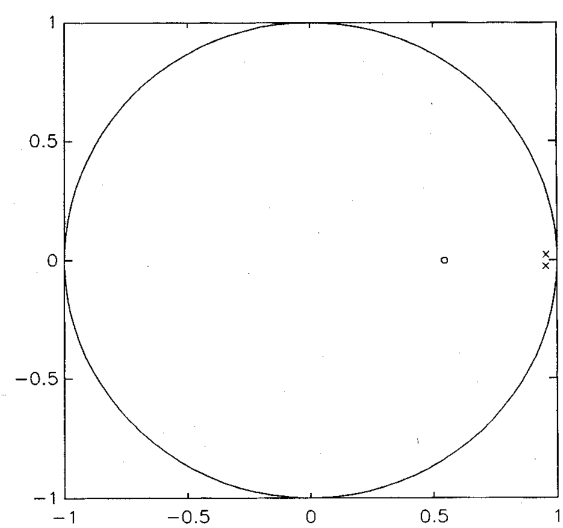
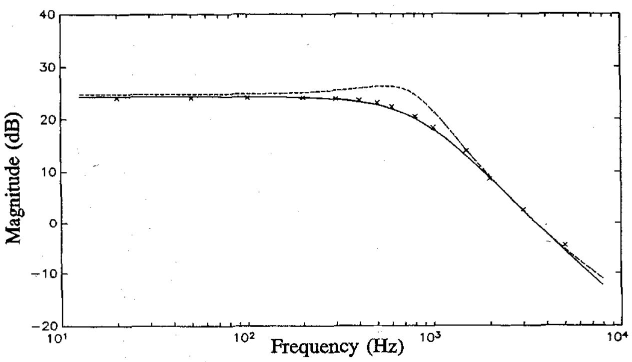


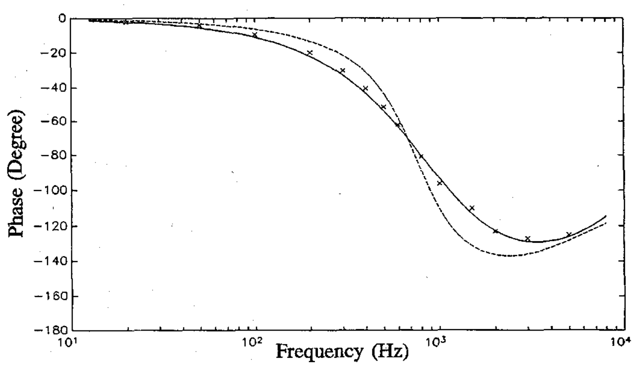
Fig. 4. Control-to-output pole-zero location.

B. Verification

To verify the proposed approach, the simulation result is compared with the experimental measurement. In order to further testify its accuracy, the result is compared with that



(a)



(b)

Fig. 5. Control-to-output frequency-domain response: - - - simulation result using the state-space averaging, ——— simulation result using the proposed approach, and x experimental result. (a) Magnitude. (b) Phase.

obtained by using a well-known circuit-oriented modeling approach, namely the state-space averaging technique. It should be noted that the transfer function obtained by using the state-space averaging technique is in the s-domain.

After a tedious mathematical derivation [9], the control-to-output transfer function $G_{cs}(s)$ of the buck converter is given by

$$G_{cs}(s) = V_s\left[\frac{R}{R+r_L}\right] \times \left[\frac{1+sr_CC}{1+s\left(r_CC+\frac{Rr_LC}{R+r_L}+\frac{L}{R+r_L}\right)+s^2LC\left(\frac{R+r_C}{R+r_L}\right)}\right]. \tag{13}$$

Similar to $G_{cz}(z)$, $G_{cs}(s)$ can be easily expressed as Bode plots using MatLab. For comparisons, the frequency responses obtained from the proposed approach and the state-space averaging technique and measured experimentally are shown in Fig. 5. It can found that the response obtained from the proposed approach closely agrees with the experimental result. The discrepancy of the response obtained from the state-space averaging technique is due to its basic assumptions that the power semiconductor devices and their switching characteristics are idea. Therefore, the proposed approach possesses the advantages of high accuracy, and free from complicated mathematics.

VI. Conclusion

This paper presents a new software tool for learning the dynamic behavior of power electronics circuits. By employing the realistic time-domain simulation of PSpice, and the parametric identification of MatLab, the corresponding transient response, transfer function, and frequency response can be obtained easily. Since the proposed approach possesses the advantages of high accuracy and no mathematical derivation, it is especially suitable for the undergraduate students. Although only a buck dc-dc converter is adopted for exemplification, the tool can readily be applied to other power converters.

Appendix

```
BUCK CONVERTER
* Power Circuit:
VS 1 0 DC 20
M 1 6 2 2 IRF640
.LIB PWRMOS.LIB
D 0 2 MR752
.LIB DIODE.LIB
RL 2 3 1.5
L 3 4 500U IC = 0
RC 4 5 0.3
C 5 0 100U IC = 0
R 4 0 10
*Comparator:
XCOMP 10 9 8 0 7 0 LM311
.LIB LINEAR.LIB
VCC 8 0 DC 5
R1 8 7 1K
R2 7 0 1K
EVGS 6 2 7 0 1
* Ramp & Duty-Ratio Generators:
VRAMP 9 0 PULSE (0 1 0 19.8U 0.1U 0.1U 20U)
VDUTY 10 0 DC 0.5
.OPTIONS ITL5 = 0 LIMPTS = 0 NOPAGE NOMOD NOECH(
.TRAN 5U 2M 0 5U UIC
.PRINT TRAN V(4) I(L)
.PROBE
.END
```


Acknowledgment

The author would like to thank the reviewers for their valuable comments.

References

[1] C. C. Chan and K. T. Chau, "A fast and exact time-domain simulation of switched-mode power regulator," *IEEE Trans. Ind. Electron.*, vol. 39, no. 4, pp. 341–350, 1992.

[2] K. T. Chau, Y. S. Lee, and A. Ioinovici, "Computer-aided modeling of quasiresonant converters in the presence of parasitic losses by using MISSCO concept," *IEEE Trans. Ind. Electron.*, vol. 38, no. 6, pp. 455–462, 1991.

[3] M. H. Rashid, *SPICE for Circuits and Electronics Using PSpice.* Englewood Cliffs, NJ: Prentice-Hall, 1990.

[4] D. W. Hart, "Circuit simulation as an aid in teaching the principles of power electronics," *IEEE Trans. Educ.*, vol. 36, no. 1, pp. 10–15, 1993.
[5] *PSpice Reference Manual Version 5.0*, MicroSim Corp., Irvine, CA, 1991.
[6] *PC-MatLab User's Guide*, The MathWorks, 1989.
[7] L. Ljung, *System Identification: Theory for the User.* Englewood Cliffs, NJ: Prentice-Hall, 1987.
[8] L. Ljung, *System Identification Toolbox for Use With MatLab User's Guide*, The MathWorks, 1988.
[9] R. P. Severns and G. Bloom, *Modern dc-to-dc Switchmode Power Converter Circuits.* New York: Van Nostrand Reinhold, 1985.

Kwok-Tong Chau (M'89) received the first-class honors B.Sc. (Eng.), M.Phil., and Ph.D. degrees, all in electrical and electronic engineering, from the University of Hong Kong in 1988, 1991, and 1993, respectively.

Since 1990 he has been with Hong Kong Polytechnic, where he currently works as University Lecturer in the Department of Electrical Engineering. His research interests include power electronics, circuits and systems, and motor drives. He has published more than 35 international journal and conference papers, as well as several industrial reports.

Dr. Chau received the Sir Edward Youde Memorial Fellowships in 1988–1989 and 1989–1990.