Modeling and Simulation of Boost Converter based on Circuit Averaging Model

Guo Chen, Yejun He, and Yong Li

School of Mechatronic Engineering, Zhongshan Polytechnic, Zhongshan 528400, China

Abstract

The principle and equivalent circuit of circuit averaging model are introduced in this paper. The PWM switching converters is a strong nonlinear time-invariant system, exact time domain simulation using PSpice needs a long time and is hard to converge. The method based on PSpice ABM (analogy behavioral modeling) and switched inductor model to model and simulate the Boost converter is also introduced. By using the averaged switch model, the time domain simulation of the Boost converter is simplified and with less time.

Keywords

Boost; Modeling and Simulation; Switched Inductor Model; ABM.

1. Introduction

As we all know, the PWM switching converters is a strong nonlinear time-invariant system, so it is difficult to find the analytical solution, this hampers the analysis and design of converter systems. The state-space averaging model introduced by R.D.Middlebrook and Slobodan Cuk in 1976, is a well-known modeling method in power electronic. The following method, such as averaged switch modeling, switched inductor and equivalent circuit model, all belong to circuit averaging. The circuit averaging method is not to average the state space equation, but to average the operating waveform or characteristics related to the nonlinear components in the converter. This method is simple to operate, and the topology obtained is similar to the original circuit, so it can be easily simulated by PSpice. The average circuit model is used instead of the switch model, so that the simulation process avoids solving the circuit state equation at each switch time, and ignores the details of the circuit process in the switch cycle, thus improving the simulation speed. The key step of the circuit averaging method is to replace the switching elements in the converter with voltage and current sources in order to obtain a time invariant circuit topology [1]. The waveforms of voltage and current sources are defined to be consistent with the switching waveforms before homogenization. After the time invariant network is determined, the circuit waveform can be averaged in a switching cycle, thus removing the switching ripple [2-4]. Then, any nonlinear element in the average circuit can be perturbed and linearized to establish a large signal simulation model. In this paper, a large signal simulation model of the continuous conduction mode current mode control Boost main circuit is established by using the switched inductance model method [5-7].

2. Modeling of Boost Converter

2.1. State-space Averaging Modeling

The principle of this method is time averaging, its objective is to converter switched circuit into time-variant circuit, and so we can use linear circuit theory to analyze small signal character. It is easy to see that the following state-space model describes the CCM converter, which has two working mode in one period:

$$\frac{dx}{dt} = q(t)(A_1x(t) + B_1U_S(t)) + (1 - q(t))(A_2x(t) + B_2U_S(t)) \tag{1}$$

$$q(t) = \begin{cases} 1 & 0 < t \le dT \\ 0 & dT < t \le T \end{cases} \tag{2}$$

where x is state vector containing all of state variables, that is, the inductor current, capacitor voltage, etc. U_S is input vector containing the independent inputs to the system, such as the input voltage source, A and B are the respective state equation matrices, d is duty cycle, T is switching period, q is defined switching function.

Upon averaging the equation (1) and (2) over one switching period one obtains:

$$\frac{dx}{dt} = (dA_1 + d'A_2)x + (dB_1 + d'B_2)U_S \tag{3}$$

$$d = \frac{1}{T}\int_{t-T}^{t} q(\tau)d\tau \tag{4}$$

where $d' = 1 - d$. This continuous-time state-space model is referred to as the famous state-space averaged model. It is obvious that, the time-variant equation (1) is converted into a time-invariant equation (3), which can be used to solve for steady-state operating point and small signal transfer function, etc. The disadvantages are that stability analysis is inaccurate, and it cannot be used to analyze switching ripple and resonant converter, etc

2.2. Circuit Averaging Modeling

Circuit averaging is another well-known technique for derivation of converter equivalent circuit. Rather than averaging the converter state equations, with the circuit averaging technique we average the converter waveforms directly. All manipulations are performed on the circuit diagram, instead of on its equations, and hence the circuit averaging technique gives a more physical interpretation to the model. By substituting the averaged circuit for switched circuit, the numerical calculation for time domain simulation has to be carried out in much smaller steps, so the computer time can be reduced. The key step in circuit averaging is to replace the converter switches with voltage and current sources, to obtain a time-invariant circuit topology. The waveforms of the voltage and current generators are defined to be identical to the switch waveforms of the original converter. Once a time-invariant circuit network is obtained, then the converter waveforms can be averaged over one switching period to remove the switching harmonics. Any nonlinear elements in the averaged circuit model can then be perturbed and linearzed, leading to the small-signal ac model.

Averaged switch model and switched inductor model (SIM) are two typical circuit averaging methods. Their basic principles are the same, the only difference is the choice of nonlinear switching element assembly. The large-signal simulation model of boost converter operating in the continuous conduction mode (CCM) will be derived by employing the two modeling approaches.

2.2.1. Averaged Switch Modeling

The averaged switch modeling is effective for modeling the low-frequency portions of the converter waveforms. The basic assumption is made that natural time constants of the converter are much longer than the switching period, so the converter has the low-pass filter character. This assumption coincides with the requirement for small switching ripple. One may average the waveforms over a time interval which is short compared to the system natural time constants, without significantly altering the system response. Hence, when the basic assumption is satisfied, it is a good approximation to average the converter waveforms over the switching period. The resulting averaged model predicts the low-frequency behavior of the system, while neglecting the high-frequency switching harmonics.

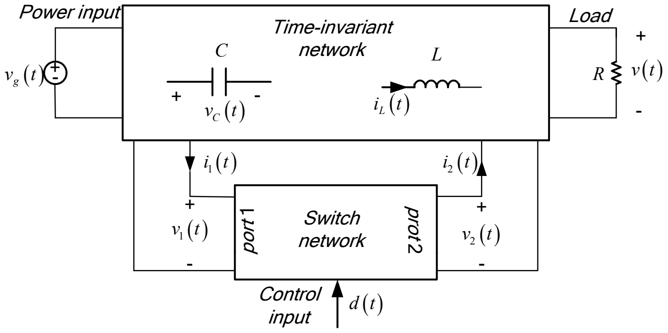
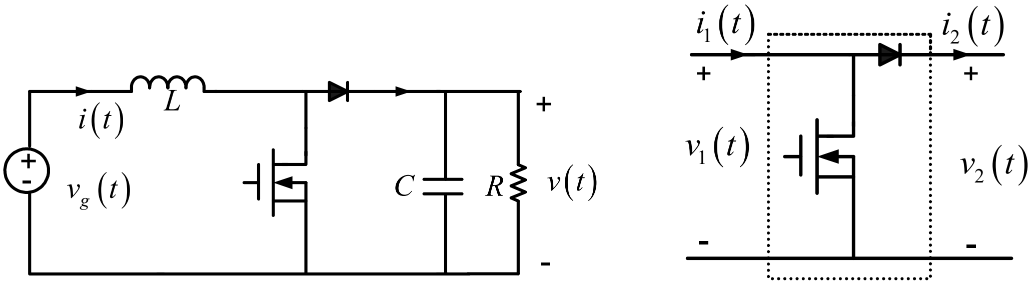


Fig. 1 System structure of switching converter

In Fig. 1, the switching elements are separated from the remainder of the converter. The converter therefore consists of a switch network containing the converter switching elements, and a time-invariant network, containing the reactive and other remaining elements. Fig. 1 illustrates the simple case in which there are two switcher; the switches can then be represented using a two-port network.

The central idea of the averaged switch modeling approach is to find an averaged circuit model for the switch network in Fig. 1. The resulting averaged switch model can then be inserted into the converter circuit to obtain a complete averaged circuit of the converter. An important advantage of the averaged switch modeling approach is that the same model can be used in many different converter configurations. It is not necessary to rederive an averaged circuit model for each particular converter. Furthermore, in many case, the averaged switch model simplifies converter analysis and yields good intuitive understanding of the converter steady-state and dynamic properties.



(a) (b)

Fig. 2 Structure of Boost converter. (a) An ideal boost converter. (b) The definition of the switch network.

Considering the ideal boost converter of Fig. 2(a), the switch network contains the transistor and the diode, as shown in Fig. 2(b). The switch network terminal quantities are illustrated in Fig. 2(b) for CCM operation. Since i_1 and v_2 coincide with the converter inductor current and capacitor voltage, it is convenient to choose these waveforms as the independent inputs to the switch network. The switch network terminal waveforms are shown in Fig. 3.

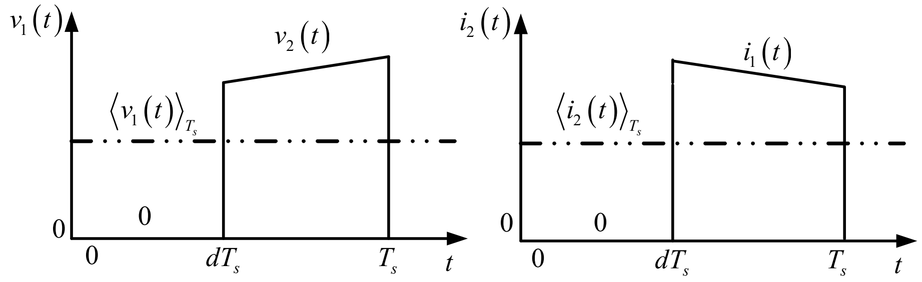


Fig. 3 Terminal waveforms of the switch network

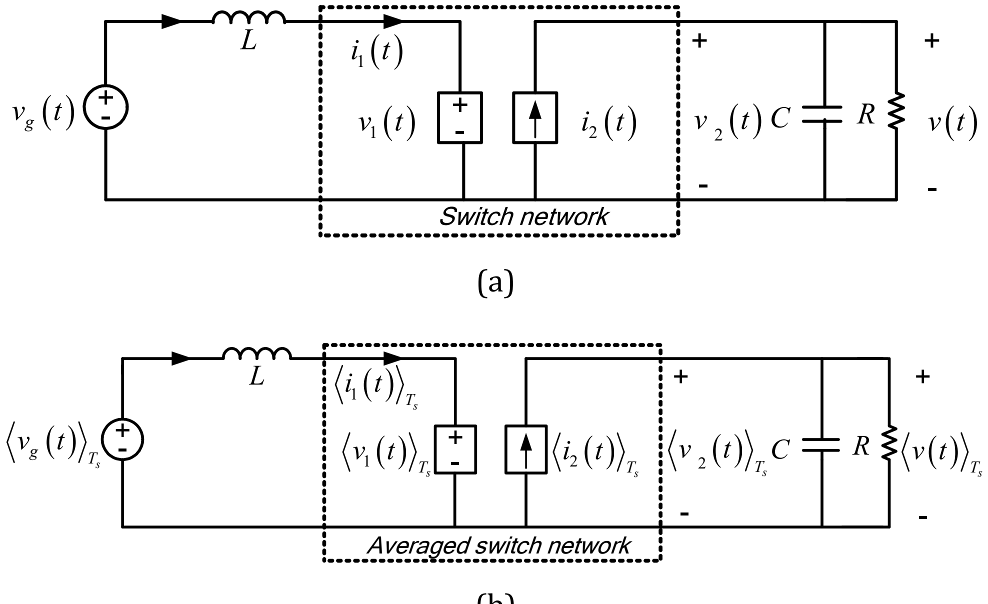


Fig. 4 Derivation of the averaged switch model of the CCM boost

First, we replace the switch network with dependent voltage and current as illustrated in Fig. 4(a). The voltage generator v_1 models the dependent voltage waveform at the input port of the switch network, i.e., the transistor voltage. As shown in Fig. 3(a), v_1 is zero when the transistor conducts, and is equal to v_2 when the diode conducts:

$$v_1(t)=\begin{cases} 0, & 0<t<dT_s \\ v_2(t), & dT_s<t<T_s \end{cases} \tag{5}$$

When v_1 is defined in this manner, the inductor voltage waveform is unchanged. Likewise, i_2 models the dependent current waveform at port 2 of the network, i.e., the diode current. As illustrated in Fig. 3(b), i_2 is equal to zero when the transistor conducts, and is equal to i_1 when the diode conducts:

$$i_2(t)=\begin{cases} 0, & 0<t<dT_s \\ i_1(t), & dT_s<t<T_s \end{cases} \tag{6}$$

With i_2 defined in this manner, the capacitor current waveform is unchanged. Therefore, the original converter circuit shown in Fig. 2(a), and the circuit of Fig. 4(a), are electrically identical. So far, no approximations have been made. Next, we remove the switching harmonics by averaging all signals over switching period, as in equation (7).

$$\left\langle x_1(t)\right\rangle_{T_s} = \frac{1}{T_s}\int_t^{t+T_s} x(t)dt \tag{7}$$

The results are:

$$\left\langle v_1(t)\right\rangle_{T_s} = d'(t)\left\langle v_2(t)\right\rangle_{T_s} \tag{8}$$

$$\left\langle i_2(t)\right\rangle_{T_s} = d'(t)\left\langle i_1(t)\right\rangle_{T_s} \tag{9}$$

Here, $d'(t) = 1 - d(t)$.

Here we have assumed that the switching ripples of the inductor current and capacitor voltage are small, or at least linear functions of time. The averaged switch model of Fig. 5 is now obtained. This is a large-signal and nonlinear model circuit model of the converter. The switching harmonics have been removed from all converter waveforms, leaving only the dc and low-frequency ac components.

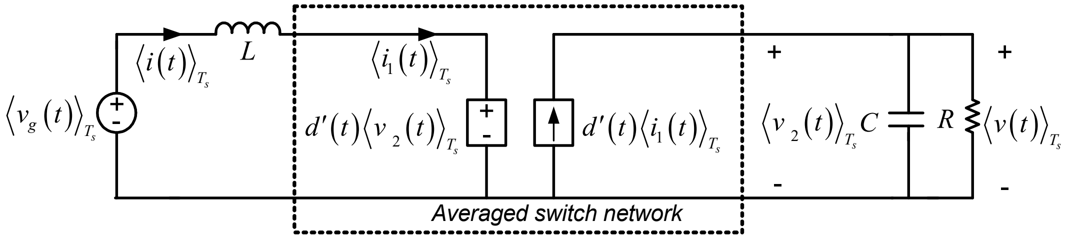
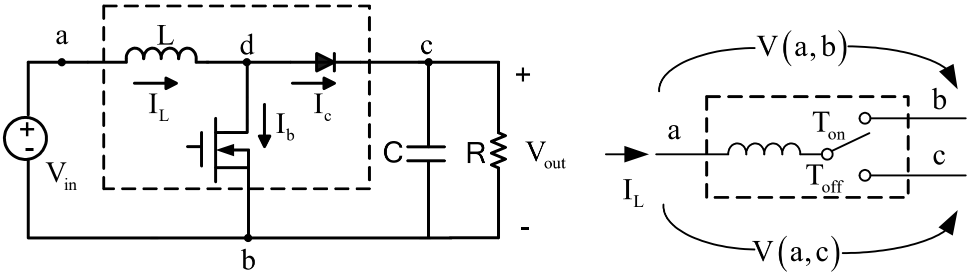


Fig. 5 Large-signal and nonlinear averaged circuit model of CCM boost converter

2.3. Switched Inductor Model

The central idea of the switched inductor modeling approach is to replace the switching part of the converter by a low frequency, averaged equivalent circuit. As illustrated in Fig. 6(a), close examination of PWM converters (buck, boost, buck-boost)reveals that they all rely on a nonlinear subcircuit: a switched inductor, which serves as a temporary energy storage element between the input and output terminals. Hence, modeling and simulation of switch mode converters can be simplified if the nonlinear part is replaced by a equivalent circuit that is compatible with PSpice. The basic switched inductor assembly (Fig. 6(b)) consists of an inductor that is switched at one end between two terminals b and c at a frequency f_s, and a duty cycle D_{on}, for port b, and D_{off} for port c.



(a) (b)

Fig. 6 Switched inductor assembly

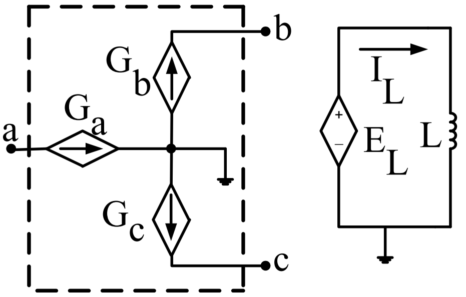


Fig. 7 Switched inductor model

The same as the approach of averaged switch modeling, the equivalent circuit of the switched inductor model (SIM) shown in Fig. 7 is easily developed by considering the average signals of the ports a, b and c under the assumption that the switching period is much smaller than the basic time constant of the converter system. Hence, the objective of the average equivalent circuit approach would be to replace this switched inductor by an dependent source circuit, such that the average voltages seen across the inductor and the average currents flowing through three ports, will remain the same as in the physical system. As illustrated in Fig. 7, G_a, G_b and G_c are dependent current sources, which are used to represent the average terminal currents of switched inductor assembly; E_L is dependent voltage source, which is used to represent the average voltage across the inductor; I_L is average inductor current, i.e., the current of terminal a; D_{on} is switch conduction duty cycle; D_{off} is diode conduction duty cycle. V(a,b) is the voltage across the terminals a and b, V(a,c) is the voltage across the terminals a and c. The expression for the dependent sources for the general case of continuous and discontinuous conduction modes are as follows:

$$G_a = I_L \tag{10}$$

$$G_b = \frac{I_L * D_{on}}{D_{on} + D_{off}} \tag{11}$$

$$G_c = \frac{I_L * D_{off}}{D_{on} + D_{off}} \tag{12}$$

$$E_L = V(a,b) * D_{on} + V(a,c) * D_{off} \tag{13}$$

For the CCM case equations (10) to (13) can be simplified by substituting :

$$D_{off} = 1 - D_{on} \tag{14}$$

And D_{off}, in this case, can be removed from the equations.

By replacing the switched inductor assembly with switched inductor model in proper orientation as in Fig. 6, We can obtain the equivalent circuit for the boost converter based on switched inductor model.

3. Modeling and Simulation of Boost Converter based on SIM

In the above two sections, we have introduced two circuit averaging modeling approaches, and obtain the equivalent circuit of boost converter. In the following section, we will use the PSpice behavioral parts to develop a PSpice-compatible simulation model of boost converter.

As illustrated in equations (10) to (13), the dependent sources of the SIM module presented above are a function of the voltages across its ports , the average current of the inductor and the duct cycle (D_{on} and D_{off}). Except for the latter, all other variables can be sensed within the module itself. To operate the SIM, an external excitation of the duty cycle must be provided. So, the behavioral boost converter includes two blocks: the SIM, discussed above and the Duty Cycle Generator.

In general case, D_{on} is normally an externally supplied signal, either from an independent generator (for open loop systems) or from a controller that generates a D_{on} as a function of the error signal.

The D_{on} generator: 1) Voltage mode. For a given primary control voltage of the PWM modulator (V_E), the definition of the dependent voltage source ($E_{D_{on}}$) that generates the duty cycle signal ($V_{D_{on}}$) is :

$$E_{D_{on}} = K_M V_E \tag{15}$$

where K_M is the modulator's proportionality constant.

Current mode. Applying the expression for the duty cycle generator for the continuous mode case, we define the duty cycle dependent voltage source as:

$$E_{D_{on}} = \frac{V_E - K_S \dfrac{|I_L|}{V_{D_{on}} + V_{D_{off}}}}{T_S \left(M_C + \dfrac{K_S V(a,b)}{2L} \right)} \tag{16}$$

where K_S is the current loop gain, M_C is the slope of the compensating ramp, I_L is average inductor current, L is the value of inductor.

D_{off} generator: In the CCM case, the relationship of D_{on} and D_{off} for both voltage and current modes are as below:

$$D_{off} = 1 - D_{on} \tag{17}$$

Hence, we have:

$$E_{D_{off}} = 1 - V_{D_{on}} \tag{18}$$

For the case of DCM mode, the relationship of *D_{on}* and *D_{off}* is:

$$E_{D_{off}} = \frac{2I_L L f_s}{V(a,b)V_{D_{on}}} - V_{D_{on}} \tag{19}$$

In the CCM case, equation (19) yields equation (20):

$$D_{off} \ge 1 - D_{on} \tag{20}$$

So, for the case of combined DCM/CCM mode, the unified equation of D_{off} is:

$$V_{D_{off}} = \min\left\{\left(1 - V_{D_{on}}\right), \left(\frac{2I_L L f_s}{V(a,b)V_{D_{on}}} - V_{D_{on}}\right)\right\} \tag{21}$$

Both the average simulation circuit based and the cycle-by-cycle simulation circuit is developed by the PSpice behavioral parts to demonstrate the power of the proposed SIM. Fig. 8 shows a cycle-by-cycle circuit model developed by PSpice part library. The inductor winding resistance R_2 is included to model the inductor copper losses. The pulsating voltage source V_2 models the driver signal and has the pulse amplitude equal to 5V. The period is 20us, the rise and fall times are 0.02ns, and the pulse width is 7us. Hence, the duty cycle is 0.35. The output capacitor is small to easily observe the voltage ripple.

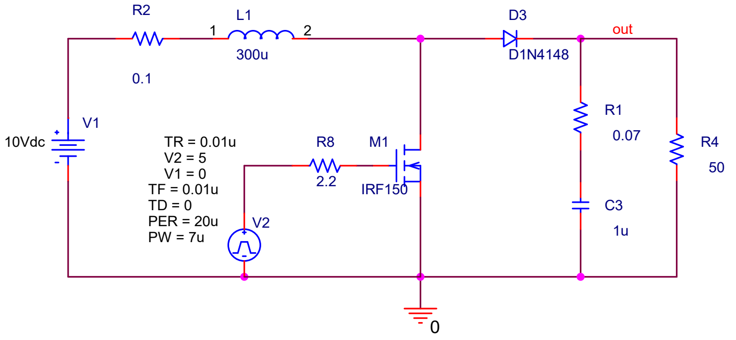


Fig. 8 Cycle by cycle simulation circuit of PWM boost converter

An averaged circuit model of the boost converter is shown in Fig. 9 developed by PSpice Analogy Behavioral Modeling part. The inductor and switched are replace by the SIM equivalent circuit. For a open loop simulation, the switch duty cycle (D_{on}) is generated by independent voltage source Vdon set to 0.35V; the diode duty cycle (D_{off}) is generated by behavioral part Etable. The EXPR property of this part is set to equation (21) to emulate the practical duty cycle output, and in order to limit the output range of the duty cycle generator, the TABLE property of this part is set to (0.01,0.01) and (0.99,0.99), so the duty cycle ranges from 0.01 to 0.99. The SIM equivalent circuit is emulated by behavioral parts GVALUE and EVALUE, where the EXPR property of Ga, Gb, Gc and EL is set according to equations (10) to (13).

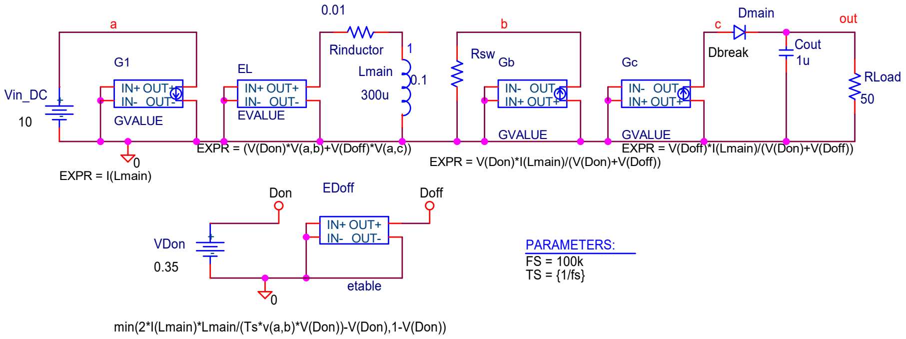


Fig. 9 Average simulation circuit with SIM of PWM boost converter

By carrying out transient simulation of PSpice, the inductor current and the output voltage waveforms during the start-up transient are shown in Fig. 10 and Fig. 11. For comparison, the waveforms obtain by transient simulation of the cycle by cycle simulation circuit shown in Fig. 8, and by simulation of the averaged circuit model of Fig. 9 are shown. Switching ripples can be observed in the waveforms obtained by simulation of the cycle by cycle circuit model. The converter transient response is governed by the converter natural time constants. Since these time constants are much longer than the switching period, the converter start-up responses in Fig. 8 and Fig. 9 take many switching cycles to reach the steady state. In the results obtained by simulation of the averaged circuit model, the switching ripples are removed, but the low-frequency portions of the converter transient responses, which are governed by the natural time constants of the converter network, match very closely the responses obtained by simulation of the cycle by cycle simulation.

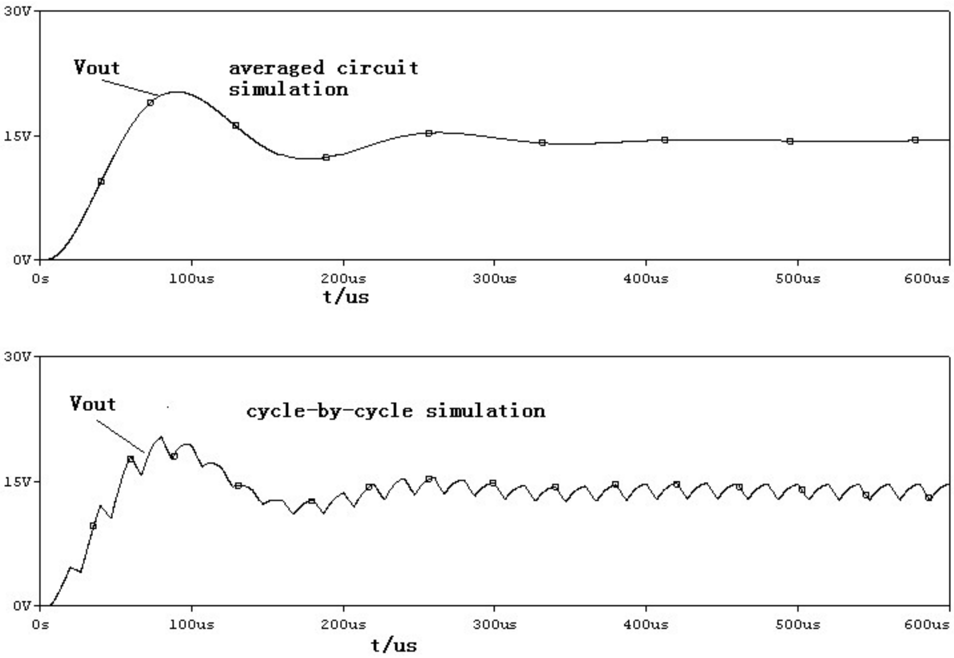


Fig. 10 Output voltage waveforms obtained by transient simulation of the circuit shown in Fig. 8 and Fig. 9

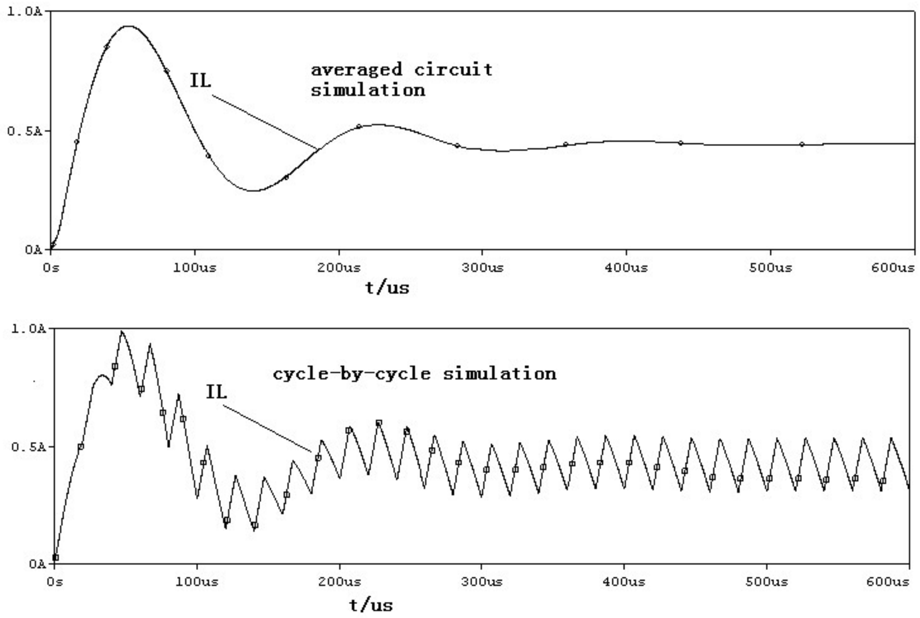


Fig. 11 Inductor current waveforms obtained by transient simulation of the circuit shown in Fig. 8 and Fig. 9

This simulation example illustrates how an averaged circuit model can be used in place of a switching circuit model to investigate converter large-signal transient responses. An advantage of the averaged circuit model is that transient simulations can be completed much more quickly because the averaged model is time invariant, and the simulator does not spend time computing the details of the fast switching transitions. This advantage can be important in simulations of large electronic systems that include switching power converters. Another important advantage also comes from the fact that averaged circuit model is nonlinear but time-invariant: ac simulations can be used to linearize the model and generate small-signal frequency response of interest. This is not possible with switching circuit models. Examples of small-signal ac simulations will be studied in the other papers.

4. Summary

This paper analyzes the fundamental principles and modeling methods of the circuit averaging model, which presents a general approach for modeling boost circuits based on the switched inductance model. The simulation model established using this method can analyze the time-domain characteristics of the system. The approach is simple and straightforward, significantly improving simulation speed. Simulation results validate the correctness of the modeling method.

Acknowledgments

This work was supported by the Guangdong Provincial University Characteristic Innovation Project (Grant No. 2023KTSCX368) and School Level Scientific Research Project of Zhongshan Polytechnic (Grant No. KYB2405).

References

[1] Y.Amran,F.Hulidhel and S.Ben-Yaakov. A unified SPICE compatible average model of PWM converters.IEEE Trans.on Power Electronics, vol.6, p. 585-594, 1991.

[2] X. Li and A. K. Bhat. Analysis and design of high-frequency isolated dual-bridge series resonant dc/dc converter. IEEE Trans. on Power Electronics, vol. 25, no. 4, pp. 850–862, 2010.

[3] S.Ben-Yaakov and Z.Gaaton. Unified SPICE compatible model of current feedback in switch mode converters.IEE Electronics Letters,vol.28, 1356-1357,1992.

[4] V.Vorperian. Simplified analysis of PWM converters using the model of the PWM switch:parts Ⅰ and Ⅱ. IEEE Trans. on Aerospace and electronic systems, vol. AES-26, p. 490-505,1990.

[5] Mahafzah, K.A.; Obeidat, M.A.; Al-Shetwi, A.Q., et al. A Novel Synchronized Multiple Output DC-DC Converter Based on Hybrid Flyback-Cuk Topologies. Batteries 2022, 8, 93.

[6] S.Ben-Yaakov and D.Adar. Average models as tools for studying the dynamics of switch mode DC-DC converters. IEEE Trans. PESC Rec., p. 1218-1225, 1994.

[7] R.W.Erickson and D.Maksimovic. Fundamentals of Power Electronics, 2nd ed. Dordrecht, The Netherlands: Kluwer, 2001: p. 226-256.