Accurate, High-Speed Simulation of Transient Response and Frequency Characteristics of Switching Converters

Shoko Sugimoto
Advanced Intelligence Research Inc.
Tokyo 102-0094, Japan
Email: sugimoto@adin.co.jp

Masahiro Suzuki
Graduate School of E.E. & C. Eng.
Chuo University
Tokyo 112-8551, Japan
Email: suzu@sugi.elect.chuo-u.ac.jp

Yasuhiro Sugimoto
Deptartment of E.E. & C. Eng.
Chuo University
Tokyo 112-8551, Japan
Email: sugimoto@elect.chuo-u.ac.jp

***Abstract*—This paper proposes a fast, precise transient response and frequency characteristics simulation method for switching converters. This method uses a behavioral simulation tool (MATLAB/Simulink) without using a SPICE-like analog simulator. The nonlinear operation of the circuit is considered, and the nonlinear function is realized by defining the formula based on the circuit operation and by applying feedback. The transient response and frequency characteristics of a current-mode boost DC-DC converter and a charge pump that were designed using a 0.18-um CMOS process were simulated by SPICE and the proposed program on a behavioral simulation tool, which we named NSTVR (New Simulation Tool for Voltage Regulators). Simulation results showed good agreement, yet NSTVR was more than 85 times faster than SPICE in CPU time in calculating frequency characteristics of a current-mode boost DC-DC converter.**

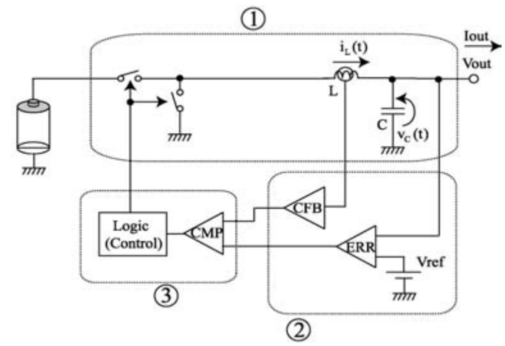


Fig. 1. Block diagram of a DC-DC converter in general.

I. Introduction

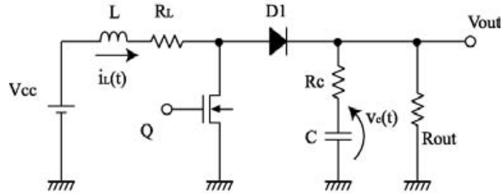
The use of DC-DC converters has become common in electronic devices because a stable supply voltage for circuits and LSIs must be produced by converting the voltage without consuming power. In the circuit design of a DC-DC converter, the performance and characteristics are commonly evaluated by SPICE simulation. As the DC-DC converter is inherently a negative feedback system with mixed analog and digital circuits, an analog simulator such as SPICE is typically used to simulate loop dynamics and stability.

Figure 1 is a block diagram of the DC-DC converter in general. Blocks in Figure 1 are divided into three parts: the switching and output part (circle number 1), the voltage and current feedback part (circle number 2) and the digital control part (circle number 3). The switching and output part consists of a smoothing filter and switching power transistors and/or diodes. The voltage and current feedback part has an error amplifier, a reference voltage, a current sensing circuit and a current-to-voltage converter. Among them, the smoothing filter in the switching and output part, and the voltage and current feedback part are analog with some nonlinear functions while the switching power transistors and/or diodes in the switching and output part, and the digital control part are digital. Because the output voltage and current are controlled in every interval of the system clock which is supplied by the oscillator, voltages and currents at various nodes in the analog part change drastically at every clock cycle, although they become periodic when a DC-DC converter operates in a steady-state condition.

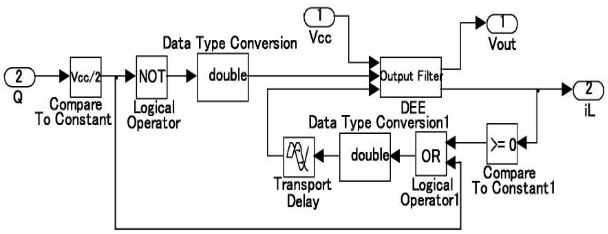
Under these circumstances, SPICE has to simulate the transient response in very small time steps to trace the drastic value changes in the analog part and in the digital part; as a result, the simulation time increases. To solve the problem, an approximated small signal model [1] was used to model the control and output block as the equivalent analog function. However, it was a linear model and couldn't handle the nonlinearity of the circuit. A periodic small signal analysis was introduced [2] so that the function that performs the periodic analysis in SPICE could be utilized, assuming that the waveform at each node is stably periodic in the steady state of operation. Unfortunately, this change did not accelerate the transient response simulation. A method using behavioral models in a simulator such as MATLAB without using a small signal model has been proposed [3], but it has not yet been verified using actual circuits.

In this paper, we introduce a new simulation method for DC-DC converters that has a faster simulation time than SPICE, that supplies simulation results as precise as those of

978-1-4244-7456-1/10/$26.00



(a) The switching and output part of a current-mode boost DC-DC converter.



(b) Modeling in a behavioral simulation tool.

Fig. 2. Modeling of a current-mode boost DC-DC converter.

SPICE and that is realized by behavioral simulation taking the nonlinearity of circuits into account.

II. Strategies of circuit modeling

Modeling strategies include the Laplace transform, state variable equations, nonlinear formulae and feedback. The switching and output part in Figure 1 is modeled by using state variable equations including parasitic elements and the newly introduced switching function. Nonlinear formulae and feedback are also used. The voltage and current feedback part in Figure 1 is modeled by using the Laplace transform. The voltage gain and frequency bandwidth of an error amplifier in the voltage feedback loop are taken into consideration. The voltage limiter across an error amplifier is modeled by applying the transistor's current equation. The modeling of the circuit to obtain a compensation slope and of the current detection circuit has been done by taking the real circuit operation into account, where the influences of the transistors' transconductances are formulated.

The digital control block in Figure 1 is modeled using the Boolean equation and the delay function. This is based on the finding that stray capacitors of transistors don't have much influence on the time constant for waveforms at various nodes in the circuit because the clock frequency is slow, only several MHz. However, the delay in the digital circuit does affect the minimum pulse width of the control signal for output power transistors, and the delay must be considered adequately. Modeling strategies are that key elements and major non-linearities need to be considered and that the modeling should be based on the circuit operation. They are common to all kinds of DC-DC converters, such as a buck DC-DC converter, a boost DC-DC converter and a charge-pump. Even when the element value changes due to process variation, it is sufficient only to replace the element value.

III. Modeling of the switching and output part

We previously introduced the modeling of a current-mode buck DC-DC converter [4]. In the present paper, we describe the modeling of a current-mode boost DC-DC converter and a charge pump.

A. Current-mode Boost DC-DC converter

Figure 2 shows the circuit of the switching and output part of a current-mode boost DC-DC converter with parasitic elements. R_L and R_C are series resistors of an inductor L and a capacitor C (ESR). R_{out} is the load resistor. During the energy charging interval, the switching power transistor Q comes on and the current in L increases at a rate of approximately V_{CC}/L to store the energy. As the diode D_1 becomes reverse-biased, there is no delivery of energy between L and V_{out}. During this interval, the following state equation holds.

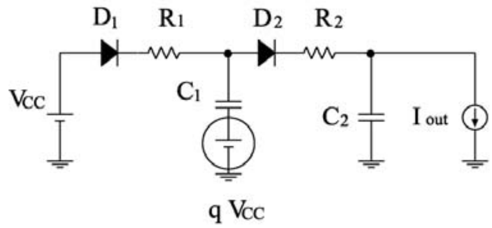
$$\begin{aligned} \frac{dv_C(t)}{dt} &= \frac{1}{C}\left\{-\frac{v_C(t)}{R_{out}+R_C}\right\} \\ \frac{di_L(t)}{dt} &= \frac{1}{L}\left\{V_{CC}-R_L i_L(t)-v_p(t)\right\} \end{aligned} \qquad (1)$$

where $v_p(t)$ is the voltage across the switching transistor when it is turned on. On the other hand, during the energy discharging period, the diode D_1 comes on while Q turns off. In this case, the state equation is,

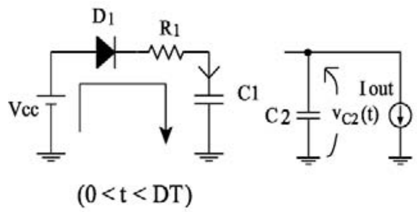
$$\begin{aligned} \frac{dv_C(t)}{dt} &= \frac{1}{C}\left\{i_L(t)-\frac{v_C(t)+i_L(t)R_C}{R_{out}+R_C}\right\} \\ \frac{di_L(t)}{dt} &= \frac{1}{L}\left\{V_{CC}-R_L i_L(t)-\frac{v_C(t)+i_L(t)R_C}{1+\frac{R_C}{R_{out}}}-V_f\right\} \end{aligned} \qquad (2)$$

where V_f is the fixed forward bias voltage of the diode. In order to express the circuit by combining two state variable equations in different phases into one, the switch function $u2$ is introduced, which corresponds to a diode's on and off action. In addition, the latter part of equation 2 is formulated by setting the diode in Figure 2(a) in conduction, and it is likely to occur the reverse current flow in an inductor. As there is no reverse current in the actual diode, it is necessary to protect the reverse current flow which only appears in the equation. Another switching function, $u3$, is introduced for this purpose. $u3$ is 1 when i_L is positive and is otherwise 0. As a result, the state equation of the circuit in Figure 2(a) becomes,

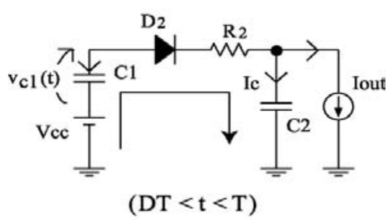
$$\begin{aligned} \frac{dv_C(t)}{dt} &= \frac{1}{C}\left\{u_2 i_L(t)-\frac{v_C(t)+u_2 i_L(t)R_C}{R_{out}+R_C}\right\} \\ \frac{di_L(t)}{dt} &= \frac{1}{L}\Bigg\{V_{CC}-R_L i_L(t)-u_2\left(\frac{v_C(t)+u_2 i_L(t)R_C}{1+\frac{R_C}{R_{out}}}\right) \\ &\qquad -u_2 V_f-(1-u_2)v_p(t)\Bigg\}u_3 \end{aligned} \qquad (3)$$



(a) The switching and output part of a charge pump.



(b) Energy charging phase.



(c) Energy transfer phase.

Fig. 3. Modeling of a charge pump.

The modeling strategy above is the introduction of the switching function for expressing one-way diode operation and for protecting reverse diode current. The $v_p(t)$ in equation (3) is the voltage across the on-resistance of a transistor.

B. Charge pump

Figure 3(a) shows the two-stage charge pump known as the Dickson-type charge pump. The on and off action of V_{CC} is controlled by the parameter q, where $q = 1$ means that V_{CC} connects to $C1$. However, in the energy charging phase of $C1$, which is the time interval between 0 to DT where D is the duty ratio of the control clock and T is the clock period, $C1$ connects to the ground ($q = 0$) and is charged by V_{CC} through diode D_1, as shown in Figure 3(b). The forward bias voltage and the equivalent series resistance of diode D_1, those of which depend on the current flowing in a diode, are $V_{f1}(t)$ and $R_1(t)$, respectively. The state equation becomes,

$$\begin{aligned} \frac{dv_{C1}(t)}{dt} &= \frac{1}{C_1 R_1(t)} \left(V_{CC} - V_{f1}(t) - v_{C1}(t)\right) \\ \frac{dv_{C2}(t)}{dt} &= -\frac{I_{out}}{C_2} \end{aligned} \quad (4)$$

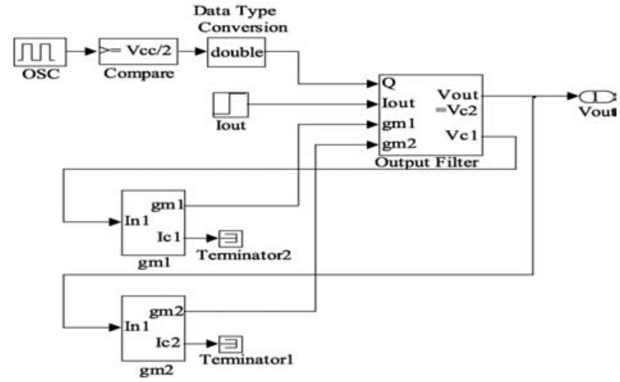


Fig. 4. Modeling of a charge pump.

In the energy transfer phase, which is the time interval from DT to T, q becomes 1 and V_{CC} is connected to $C1$, then $C2$ is charged through diode D_2, as shown in Figure 3(c). The forward bias voltage and the equivalent series resistance of diode D_2 are $V_{f2}(t)$ and $R_1(t)$, respectively. The state equation becomes,

$$\begin{aligned} \frac{dv_{C1}(t)}{dt} &= \frac{1}{C_1 R_2(t)} \left(v_{C2}(t) - v_{C1}(t) + V_{f2}(t) - V_{CC}\right) \\ \frac{dv_{C2}(t)}{dt} &= -\frac{I_{out}}{C_2} - \frac{1}{C_2 R_2(t)} \left(v_{C2}(t) - v_{C1}(t) + V_{f2}(t) - V_{CC}\right) \end{aligned} \quad (5)$$

As a result of these series of operations, the voltage across C_2 becomes approximately twice that of V_{CC}. The state equation in both phases can be combined into one by using parameter q, such as,

$$\begin{aligned} \frac{dv_{C1}(t)}{dt} &= (1-q)\frac{1}{C_1 R_1(t)}\left(V_{CC} - V_{f1}(t) - v_{C1}(t)\right) \\ &\quad + q\frac{1}{C_1 R_2(t)}\left(v_{C2}(t) - v_{C1}(t) + V_{f2}(t) - V_{CC}\right) \\ \frac{dv_{C2}(t)}{dt} &= (1-q)\left(-\frac{I_{out}}{C_2}\right) + q\left\{-\frac{I_{out}}{C_2} \right. \\ &\quad \left. - \frac{1}{C_2 R_2(t)}\left(v_{C2}(t) - v_{C1}(t) + V_{f2}(t) - V_{CC}\right)\right\} \end{aligned} \quad (6)$$

In this way, the state equation has been set up for the circuit shown in Figure 3(a). State equations for multiple stages of a Dickson-type charge pump can be set up in the same way. The modeling in a behavioral simulation program becomes like that shown in Figure 4. The nonlinear effect of the circuit is again considered faithful to the modeling strategy described in section 2. Feedback blocks of gm1 and gm2 contain nonlinear resistors and the bulk effect of transistors that form diodes.

IV. Performance comparison between SPICE and NSTVR

In order to verify the usefulness of the proposed simulation method NSTVR, we compared our simulation results of transient and frequency characteristics with those of SPICE in Figures 5(a) and 5(b) in the case of a current-mode boost DC-DC converter. In the case of a charge pump, we compared the results of transient response with those of SPICE in Figure

6. The simulated circuit by SPICE is the same as the circuit of the IC, which has been designed using a 0.18-um CMOS process.

Figure 5(a) shows the simulation results of the transient response at Vout terminal (+5V) when the 2.5-V step input voltage V_{CC}, in Figure 2(a), is applied. The clock frequency and load current were 1 MHz and 100 mA, respectively. The simulation is from time 0 to 200 us in 1-ns steps by both SPICE and NSTVR, and the measured CPU time is listed in Table 1. Both traces agreed well within 3 mV with each other and NSTVR was 22 times faster than SPICE.

Figure 5(b) compares the gain and phase frequency characteristics of the total feedback loop of the current-mode boost DC-DC converter. In this case, SPICE needs to repeat the transient analysis with an injected signal in a loop one at a time as the signal frequency changes. As shown in Figure 5(b) and Table 1, SPICE took 204 minutes to simulate the gain and phase of the loop at only five different frequencies, denoted by X characters in Figure 5(b), whereas NSTVR took 11 minutes to simulate 23 different frequencies. Thus NSTVR was 85 times faster than SPICE. The good agreement in phase and extremely fast simulation time indicate the effectiveness and usefulness of NSTVR.

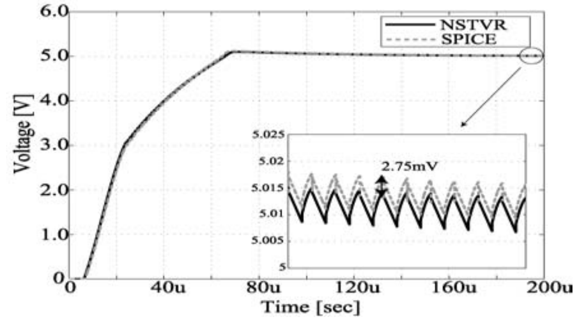
Figure 6 shows the simulation results of the transient response of a charge pump at terminals across capacitor $C2$ when the input voltage V_{CC} is 5-V in Figure 3(a). The output voltage falls to approximately 7.5-V due to the large gate-to-source voltage of a transistor in diode connection with a large bulk effect even when the load current is 0 mA (no load). The clock frequency and capacitor value were 1 MHz and 1 uF, respectively. It is simulated from time 0 to 400 us in 1-ns steps by both SPICE and NSTVR, and the measured CPU time is listed in Table 1. These traces agreed well within 150 mV with each other. NSTVR was only 1.5 times faster than SPICE in this case because the circuit is simple and has only a two-stage configuration.

V. Conclusions

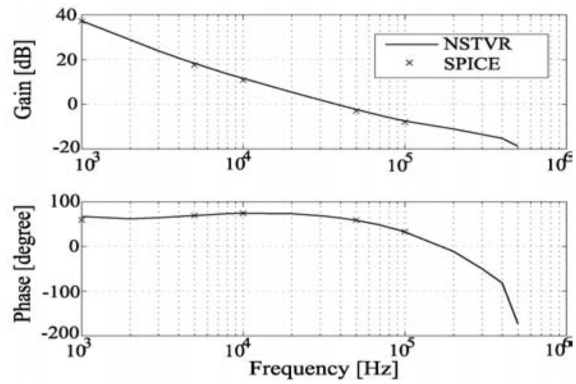
It was verified that the proposed NSVTR which introduced a new modeling method to behavioral simulation, considering only key elements and major non-linearities and taking circuit-based approach, is effective for the fast and precise simulation of DC-DC converters.

TABLE I
CPU time comparison for three different simulations in Figures 5 and 6 (CPU is Sun Blade 2000 UltraSPARC III).

	Figure 5(a)	Figure 5(b)	Figure 6
SPICE	357 sec	204 min. (5 freq. points)	58 sec
NSTVR	16 sec	11 min. (23 freq. points)	38 sec



(a) Transient response at Vout terminal.



(b) Frequency characteristics.

Fig. 5. Performance comparison between SPICE and NSTVR.

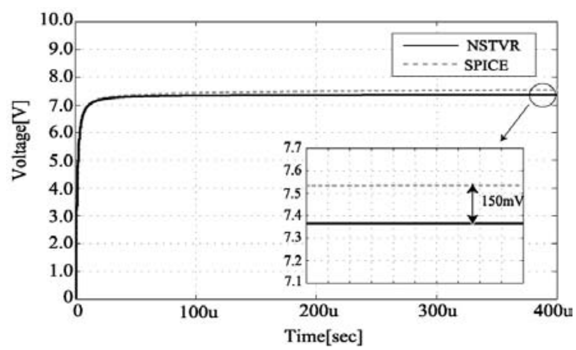


Fig. 6. Transient response of a charge pump (no load).

References

[1] R.B.Ridley, "A New, Continuous-Time Model For Current-Mode Control," IEEE Transactions on Power Electronics, vol. 6, no. 2, pp. 271-280, April 1991.

[2] H.Deng, A.Q.Huang and Y.Ma, "Frequency response analysis for switching converters in SPICE without averaging," in 2004 35th Annual IEEE Power Electronics Specialists Conference, 2004, pp. 861-866.

[3] J.H.Ly, and Kasemsan Siri, "Frequency Response Analysis for DC-DC Converters without Small-Signal Linearization," in 2003 IEEE Applied Power Electronics Conference, 2003, pp. 1008-1013.

[4] Y.Sugimoto, "A Highly Efficient Transient and Frequency-Response Simulation Method for Switching Converters without Using a SPICE-like Analog Simulator," International Conference on Circuits and Systems, pp.1308-1311, June 2010.