Stochastic Analysis of Switching Power Converters via Deterministic SPICE Equivalents

Original

Availability:
This version is available at: 11583/2536295 since:

Publisher:
IEEE

Published
DOI:10.1109/TPEL.2014.2301561

Terms of use:
openAccess
This article is made available under terms and conditions as specified in the corresponding bibliographic description in the repository

Publisher copyright

(Article begins on next page)
Stochastic Analysis of Switching Power Converters via Deterministic SPICE Equivalents

Paolo Manfredi, Member, IEEE, Igor S. Stievano, Senior Member, IEEE, Flavio G. Canavero, Fellow, IEEE

Abstract—This letter addresses the stochastic analysis of nonlinear switching power converters via an augmented circuit equivalent and a single deterministic SPICE simulation. The proposed approach is based on the expansion of the constitutive relations of the circuit elements in terms of orthonormal polynomials within the well-established framework of polynomial chaos. The feasibility and strength of the method are demonstrated on a DC-DC boost converter described by either detailed nonlinear components or via its averaged linear circuit. Excellent modeling accuracy as well as remarkable speed-ups compared to traditional sampling-based approaches are achieved.

Index Terms—Circuit simulation, polynomial chaos, SPICE, stochastic circuits, switched mode power supplies.

I. INTRODUCTION

Switching power converters constitute a standard supply configuration in many electrical and electronic applications. In critical (e.g., biomedical or vehicular) equipment, serious concerns are raised by stability and reliability under the unavoidable presence of large uncertainty on power electrical components [1]. Typical examples are provided by inductors and electrolytic capacitors, that are affected by non-negligible manufacturing tolerances, nonideal behavior or measurement errors, as well as parasitics, source noise and operating temperature and conditions [2].

Statistical analysis of power switching networks therefore became a hot topic in the past decade [3]–[5]. The high computational inefficiency that affects traditional techniques, like the Monte Carlo (MC) method or possible enhanced variations based e.g. on the design of experiments or similar tools [6], paved the way to more efficient circuit simulation approaches, such as those based on the polynomial chaos (PC) framework (see, e.g., [5], [7], [8] and reference therein). The underlying idea of PC is to represent circuit responses in terms of basis expansions that carry fundamental statistical information. Combination with statistical tests, e.g. the analysis of variance (ANOVA), possibly allows a preliminary screening and selection of the most relevant random parameters [9].

The present contribution extends the results of previously published papers in this field (see [5], [7], [8]), providing a framework for the circuit-level stochastic analysis of both linear and nonlinear circuits via an augmented deterministic equivalent. One of the main achievements is that the resulting equivalent circuit can be readily simulated in standard SPICE-type simulators, thus allowing circuit designers to rely on existing and well-consolidated tools, providing hundreds of device models, rather than requiring the creation of a customized software and the re-development of pertinent library models. A very general framework for the inclusion of nonlinear components is also outlined, as such enabling the analysis not only of linearized average models for power converters, but also of their detailed nonlinear description.

II. EXAMPLE TEST CASE

For illustration purposes we focus this letter on the typical DC-DC boost converter shown in Fig. 1. The above example, belonging to a class of switching circuits widely spread in power electronics, offers a representative test case of a closed-loop system involving both linear and strongly nonlinear circuit components, thus allowing to stress the benefits of the proposed methodology and to demonstrate its feasibility and strength.

![Fig. 1. Example test case: DC-DC boost converter with analog feedback control.](image-url)

In Fig. 1, an analog voltage and current feedback control is used. The circuit elements take the following values: $E = 5 \text{ V}$, $r_L = 1 \text{ m}\Omega$, $L = L_0 = 5 \text{ mH}$, $r_C = 5 \text{ m}\Omega$, $C = C_0 = 10 \mu\text{F}$, whilst the switching frequency is $f_s = 45 \text{ kHz}$. As to the feedback network, the configuration in [3] is considered among a number of existing alternative control strategies. The MOS device is driven by a square wave with a suitable duty-cycle $D$, that is provided by the feedback network as a function of $v_C$ and $i_L$. Specifically, the MOS gate control is the output of a comparator, whose non-inverting input is fed by a sawtooth waveform, whereas its inverting input is defined as $d(v_C, i_L) = V_{ref} - k_1 i_L - k_2 v_C$. The coefficients $k_1$ and $k_2$ are designed to optimize the performance of the closed-loop circuit and $V_{ref}$ is the desired output voltage. Here $V_{ref} = 12 \text{ V}$.
where \( P \) is the number of terms defining the truncated series and \( v_k(t) \) are deterministic coefficients to be determined for the nodal or branch voltages. A similar reasoning applies to the circuit currents \( i(t) \). Finally, \( \varphi_k \) are the elements of a polynomial basis orthonormal with respect to the inner product:

\[
\langle f, g \rangle = \int_R f(\xi)g(\xi)w(\xi)d\xi,
\]

where \( w(\xi) \) is the probability distribution of \( \xi \). According to the Wiener-Askey scheme, for standard distributions, the orthogonal polynomials are well-known and correspond to Hermite polynomials for Gaussian random variables, Legendre polynomials for uniform variables, etc. [7]. In order to preserve symmetry and reciprocity in the models, such orthogonal polynomials are normalized so that \( \langle \varphi_k, \varphi_j \rangle = \delta_{kj} \) (Kronecker’s delta). For instance, the orthonormal Hermite polynomials are \( \varphi_0 = 1 \), \( \varphi_1 = \xi \), and \( \varphi_2 = \frac{1}{\sqrt{2}}(\xi^2 - 1) \), etc.

The main advantage of the PC representation (1) is that, once the unknown coefficients \( v_k(t) \) are computed, the statistical information of \( v(t) \) (e.g., its mean or standard deviation) can be effectively computed form (1) via analytical or numerical techniques [7], [8].

The rationale of the proposed simulation approach is to derive deterministic constitutive equations relating the voltage and current coefficients for the circuit elements exhibiting variability. Such relationships are then implemented and simulated in SPICE via equivalent circuits to retrieve the sought-for PC variables. The remainder of this section illustrates the approach by focusing on the derivation of the augmented constitutive equations for two different types of lumped circuit elements: linear and nonlinear.

![Fig. 2. Transient SPICE simulation of the boost circuit in Fig. 1. Gray line: detailed simulation; black line: simplified average simulation.](image)

In order to illustrate and validate the benefits of the proposed technique, a variability of the dynamical elements \( L \) and \( C \) will be considered and parameterized as \( L = L_0(1 + \sigma_1\xi_1) \) and \( C = C_0(1 + \sigma_2\xi_2) \), respectively, where \( \xi_1 \) and \( \xi_2 \) are independent Gaussian random variables with zero mean and unit variance, whereas \( \sigma_1 \) and \( \sigma_2 \) are the normalized standard deviations. The next section discusses how the SPICE simulation models can be suitably modified to account for the effects of variability on linear and nonlinear circuit components.

### III. POLYNOMIAL CHAOS-BASED SPICE EQUIVALENTS

This section summarizes the basic mathematical tools that allow to create SPICE-compatible circuit models accounting for the variability of network elements. For conciseness, the discussion is based on one random variable only, parameterized by the normalized variable \( \xi = \xi_1 \), being the extension to the multivariate case straightforward and available in the literature (e.g., see [8]). In presence of such a random quantity, both the voltages and the currents in the network inherently depend on \( \xi \), thus becoming stochastic variables themselves.

The idea underlying the PC is the expansion of the above electrical variables in terms of polynomials, e.g.

\[
v(t, \xi) \approx \sum_{k=0}^{P} v_k(t)v_k(\xi),
\]

where \( v_k(t) \) and \( v_k(\xi) \) are deterministic coefficients and deterministic polynomials, respectively.

The first example deals with a linear inductor connecting nodes A and B, and whose characteristic is \( v_A(t) - v_B(t) = L \frac{di(t)}{dt} \), where \( v_A(t) \) and \( v_B(t) \) are the pertinent nodal voltages and \( i(t) \) is the current flowing through the inductor (see the left panel of Fig. 3). For conciseness, all the electrical variables are expressed in terms of a two-term expansion, leading to

\[
(v_{A0}(t)v_0 + v_{A1}(t)v_1) - (v_{B0}(t)v_0 + v_{B1}(t)v_1) =
= (L_0\varphi_0 + L_1\varphi_1)\frac{d}{dt}(v_0(t)v_0 + i(t)v_1))
\]

The projection of (3) onto the first polynomial \( \varphi_0 \) via (2) produces:

\[
v_{A0}(\varphi_0, v_0) + v_{A1}(\varphi_1, v_0) - v_{B0}(\varphi_0, v_0) - v_{B1}(\varphi_1, v_0) =
= L_0(\varphi_0v_0, \varphi_0)\frac{d}{dt}i_0 + L_0(\varphi_0v_1, \varphi_1)\frac{d}{dt}i_1\]

\[
+ L_1(\varphi_1v_0, \varphi_0)\frac{d}{dt}i_0 + L_1(\varphi_1v_1, \varphi_0)\frac{d}{dt}i_1,
\]

where \( v_0(t) \) and \( v_1(t) \) are deterministic coefficients.

![Fig. 3. Linear inductor and its corresponding augmented model.](image)
where the dependence on the time variable \( t \) is neglected for brevity of notation and the terms \( \langle \cdot, \cdot \rangle \) reduce to mere real numbers that can be analytically computed from (2).

The above equation, along with the projection of (3) onto the second polynomial \( \varphi_1 \), leads to the following constitutive relations for the expansion coefficients of the stochastic inductor:

\[
\begin{align*}
v_{A0}(t) - v_{B0}(t) &= L_0 \frac{d}{dt} i_0(t) + L_1 \frac{d}{dt} i_1(t) \\
v_{A1}(t) - v_{B1}(t) &= L_1 \frac{d}{dt} i_0(t) + L_0 \frac{d}{dt} i_1(t).
\end{align*}
\]

(5)

It is worthwhile noting that (5) are deterministic because the dependence on the random variable \( \xi \) has been suppressed by the projection procedure. This suggests to represent the inductor in terms of a new multi-terminal circuit element involving the coefficients of the node voltages and branch currents. According to (5), this turns out to be a mutual inductor, as illustrated in Fig. 3. The above interpretation also provides a clever way to implement this new element in any SPICE-type simulator by means of standard circuit elements, as follows:

```
.subckt Laugmented A0 B0 A1 B1 P0=... P1=...
L1 A0 B0 P0
L2 A1 B1 P0
K12 L1 L2 K="P1//P0"
.ends Laugmented
```

The approach is readily extended to expansions with a larger number of terms as well as to the modeling of the stochastic capacitor or other linear elements.

The second example involves a circuit element with nonlinear characteristic. For instance, the output of a comparator in Fig. 1 can be expressed as \( v_B(t) = F(v_A(t)) \), with \( F(\cdot) \) being a suitable (e.g., piecewise) nonlinear function, whereas \( v_A \) and \( v_B \) are the input and output (differential) voltages of the comparator, respectively. Again, a second-order expansion of the electrical variables leads to:

\[ v_{B0}(t) \varphi_0 + v_{B1}(t) \varphi_1 = F(v_{A0}(t) \varphi_0 + v_{A1}(t) \varphi_1). \]

(6)

The projection of (6) onto the first orthonormal polynomial \( \varphi_0 \) yields:

\[ v_{B0}(t) = \int_{\mathbb{R}} F(v_{A0}(t) \varphi_0(\xi) + v_{A1}(t) \varphi_1(\xi)) \varphi_0(\xi) w(\xi) d\xi. \]

(7)

The inner product integral on the right-hand side has been made explicit to highlight that, due to the nonlinearity, no closed-form expression is available. In order to arrive at a deterministic and closed-form expression, the integral in (7) is approximated by means of a quadrature rule, thus leading to:

\[ v_{B0}(t) \approx \sum_{q=1}^{Q} F(v_{A0}(t) \varphi_0(\xi_q) + v_{A1}(t) \varphi_1(\xi_q)) \varphi_0(\xi_q) w_q. \]

(8)

In (8), \( Q \) is the number of quadrature points \( \xi_q \), and \( w_q \) are the corresponding weights, depending on the adopted quadrature rule. It is important to note that (8) is a deterministic nonlinear function, since the terms \( \varphi_0(\xi_q), \varphi_1(\xi_q) \) and \( w_q \) become numerical coefficients. Moreover, a very good modeling accuracy with a low number of points \( Q \) can be achieved by selecting the optimal Gaussian quadrature rule according to the weighting function \( w(\xi) \) (e.g., Gaussian-Hermite, Gauss-Legendre, etc.). For Gaussian variability, a Gauss-Hermite quadrature with \( Q = 3 \) yields for instance \( \xi_q = \{ -\sqrt{3}, 0, +\sqrt{3} \} \) and \( w_q = \{ 1/6, 4/6, 1/6 \} \).

Repeating the procedure for the second polynomial \( \varphi_1 \) provides the sought-for deterministic constitutive equations for the coefficients of the electrical variables. Expressions like (8) are implemented into SPICE by means of standard controlled sources. Also, analogous steps can be readily applied to elements described by a current-voltage characteristic, like diodes, as well as to more complex, possibly multi-terminal, subcircuits, like those modeling MOSFET devices. Fig. 4 provides an illustrative and graphical summary of the proposed method for the general case of a nonlinear multi-terminal device.

The above procedure is applied in an automated fashion to all the linear and nonlinear circuit elements of the stochastic network, and a new augmented circuit is built by suitably interconnecting the augmented counterparts of each element. What is more important, a single deterministic SPICE simulation of the created network allows to compute all the voltage and current coefficients, and subsequently retrieve the statistical information of the circuit response. The proposed procedure does not require dedicated or customized tools for the circuit simulation, thus relying on commercially available SPICE simulators.

Fig. 4. Graphical interpretation of the expected augmented characteristic of a generic multi-terminal circuit element represented as a subcircuit and possibly representing the MOS device or the comparator in the circuit of Fig. 1.

### IV. Numerical Results

This section collects the results of the stochastic simulation of the example test case of Fig. 1. Fig. 5 shows the stochastic simulation of the averaged circuit for the boost converter, carried out with HSPICE by considering \( \sigma_1 = \sigma_2 = 5\% \). The gray area results from the superposition of a limited set of MC responses and provides a qualitative idea of the response fluctuation due to circuit uncertainties. The black lines are the average response and the \( \pm 3\sigma \) limits estimated using 1000 MC samples. Finally, the markers indicate the same statistical information achieved via the PC-based simulation of the augmented circuit, constructed considering \( P = 5 \). From the above comparison, a very good accuracy is established for the proposed technique.

To validate the modeling of nonlinear components, a stochastic simulation is performed also for the complete circuit with the MOSFET switch, the diode and the feedback network. In order to provide a thorough comparison on the detailed
Finally, Tab. I shows the relative difference $\epsilon_\sigma$ between the standard deviations of the output voltage at 2.1 ms (i.e., in the middle of the first window in Fig. 6) estimated with PC and MC, the latter using an increasing number of $N = 10, 100$ and 1000 simulation samples. The table indicates that a large number of MC samples is required to approach the PC accuracy. For further information on the convergence and the accuracy of both MC and PC, the reader is referred to [8]. It is important to point out, however, that the proposed PC-based method still provides an improvement over MC even when smaller numbers of simulation samples are considered.

V. CONCLUSIONS

This letter addresses the stochastic circuit simulation of nonlinear switching networks for power electronics by means of deterministic SPICE equivalents. For a given network with circuit elements defined by uncertain coefficients, the voltage and current variables are expressed in terms of a truncated series of orthonormal polynomials of random variables. The coefficients of the above expansion carry fundamental statistical information and are interpreted as deterministic variables of an equivalent augmented circuit. The polynomial chaos tool, possibly combined with quadrature integration schemes, provides a robust framework for the computation of the SPICE equivalents to be used in the augmented network. A single SPICE simulation of the resulting circuit allows to compute the coefficients of the voltage and current variables and therefore their statistical behavior. An example involving the statistical analysis of a either fully nonlinear or averaged and linearized DC-DC boost converter is used to demonstrate the benefits of the proposed technique in terms of both accuracy and efficiency.

ACKNOWLEDGEMENT

The authors would like to thank Prof. Dries Vande Ginste and Prof. Daniël De Zutter for the fruitful discussion and collaboration on the topic of nonlinear PC-based simulations during their visiting research periods at Ghent University.

REFERENCES