Hierarchical Approach in Modeling and Simulation of Power Electronics for Education

P.J. van Duijsen*, D.C. Zuidervliet†
THUAS, Delft, The Netherlands, *www.dc-lab.org*
* p.j.vanduijsen@hhs.nl †d.c.zuidervliet@hhs.nl

***Abstract*—For teaching power electronics circuits, circuit simulation is well adapted, to show the working principles of the various types of power electronics converters. However, depending on the complexity of the models used, the teaching purpose might be troubled by the implementation details, required for the specific simulation program. Instead of focusing on the power electronics working principles, limitations and/or parametrization of models, becomes a disturbing factor. Depending on the goal of the simulation results, modeling should be fitted exactly to the knowledge level and availability of parameters. In this paper, a hierarchical modeling method is presented, which allows the simulation, of basic principles, up to detailed circuits. Linear models and open-loop operation, are for the understanding of the basic principles. Generic models are added to simulate closed-loop behavior. Detailed component models, are finally required, for the simulation of closed-loop feedback, EMI and thermal behavior. Finally, parasitic components are added, for detailed electromagnetic interference and component stress simulation. The basic working principles are modeled using simple models, and then, depending on the required simulation results, the models are expanded into more detailed models.**

***Index Terms*—Modeling, power electronics, switched mode power supply, simulation, generic model, current mode control**

I. Introduction

THe role of simulation in the education of power electronic, and especially switched mode power supplies is nowadays not questionable any more. However, in the past, this used to be different, and simulation was more or less accepted on the fence, regarding modeling effort and the accuracy of simulation results.

With the availability of microcomputers in the early seventies, modeling and simulation for power electronics became within reach for many power electronics engineers. In integrated circuit design, simulation already was taking his place, simply because of the large size of the circuits and the inability to breadboard such large circuits. From those early developments in circuit simulation programs, Spice [1], developed at the Berkeley University, became the most used electronics simulation package. Because it was very flexible and the original source code was freely available, it was adopted by many research institutes and commercial developers. Commercial packages for microcomputers started to appear, and some of them even exists today. Since so many engineers were using Spice, it somehow became a verb [2]. Spice became the de-facto standard in circuit simulation and is also widely used for power electronics, although there were no models to tackle the switching behavior correctly, other than detailed simulation models.

The first approaches dedicated towards using simulation for power electronics, relied heavenly on the mathematical background of users. Knowledge on applying state space equations and numerical routines, were required from the users [3]. The application of state space equations, versus the Modified Nodal Analysis [4] for simulation, was extensively discussed during power electronics specialist conferences [5].

In the eighties and nineties, new simulation programs were developed, especially for switched mode power supplies [SMPS] [6], [7]. Multilevel modeling techniques were introduced, that allow a more dedicated modeling of power electronics, control and electrical machines [8], and let to the development of the program Caspoc [9]. An overview of simulation programs by the end of the nineties, is given in [10].

By the turn of the century, simulation becomes accepted for studying power electronics behavior [11]. Nowadays, simulation is so common, that it is simply integrated into undergraduate and graduate curriculum [12], [13].

Simulation is used in the popular textbooks on power electronics. Especially Spice is included in [14]–[18], while Caspoc is used in [19], [20].

The need for the hierarchical approach is first outlined in section II, and in section III, the three modeling approaches are introduced. The advantages of a high simulation speed, to first get an idea of the power electronics circuit, is described in section IV. The generic approach, which includes much more functional control and semiconductor models with parasitic components, is discussed in section V. The approach where the semiconductor models are modeled in detail, is discussed in section VI

II. Hierarchical approach

Teaching power electronics requires a broad knowledge from the students. Not only electronics, but also topics like magnetic, thermal, control and electromagnetic interference are important. The undergraduate laboratory courses, are mostly focusing on a single specific topic. Because the operation of power electronics requires knowledge from all of the above mentioned topics, also modeling and simulation of all those topics could overwhelm the students.

A. Approach

The approach shown in this paper is intended to start at the second year undergraduate level, where students do have a background in basic electronics and electrical circuits, but are not yet familiar with detailed electronics circuit analysis and behavior. However during the three year studies, they are developing their knowledge and skills on the above mentioned topics. During this three year period, the students are thus confronted with a first understanding of the basics, up to a profound and detailed analysis in their last year. It would be possible to have different software for the different knowledge levels of the students, but giving them a hierarchical software approach, where they can start with simple basic models up to detailed models, allows them to share user experience and build-up useability knowledge, over the years. The continuous use of the same modeling methods with increasing complexity, is inline with the development of their theoretical knowledge of the subject under study.

B. Multilevel

The multilevel approach [6], [8], [9] was a first attempt to the hierarchical approach, but the focus was on solving the engineering simulation issues, not the educational aspects. The hierarchical approach for the theory of power electronics, is also applied in the early textbooks [14], [17] and [18]. In the explanation of the theory, simple ideal switch models instead of detailed Mosfet models were used, to demonstrate the operation and behavior of the power electronics. The derivation of the mathematical relations is first based on the basic approach, the switch model, but finally extended to include the typical details of the Mosfets. A similar approach is used for simulation, where the students start with using a simple simulation model, but gradually improve the models with more details. The approach for the explanation of the theory in [14], [17], [18], is followed in [15] and [16] The hierarchical approach for modeling and simulation is practiced using the multilevel techniques [6], [8] and extended more generally to electric systems in [11].

C. Software

As an example of mismatch in use of software for education, are the many issues with convergence problems of the numerical integration methods used in the spice-based simulators [2], [8], [10]. If students are confronted with these kind of problems, not related to their actual learning process, time and effort is spend on solving those problems. Time and effort that could have been spend on their actual learning process of understanding and practicing the subject of power electronics. Therefore the entry level should be free of these above mentioned software problems, and dedicated to the teaching subject.

D. Level

The hierarchical approach as described in this paper, is thus applicable to the set of power electronics courses for undergraduate and graduate study. The examples in the next sections, detail how this hierarchical approach is used to structure the broad range of topics relevant to education of power electronics.

E. Implementation

To implement this hierarchical approach for modeling and simulation, the development of undergraduate and graduate courses, should address the importance of combining the theory, simulation and laboratory experiments, within the same semester, or half semester [12], [13].

The hierarchical approach was applied to the undergraduate third semester power electronics courses in Delft, Netherlands [12], [13], [22], where the entry level is only a basic understanding of electric circuits and electronics. The undergraduate students in the third semester do not yet have an understanding of numerical integration methods. The underlying numerical integration methods therefore have to be hidden to the students, and only models with basic parameters should be applied. Students thus practice the influence of those basic parameter(s), and are not affected by problems of the underlying software.

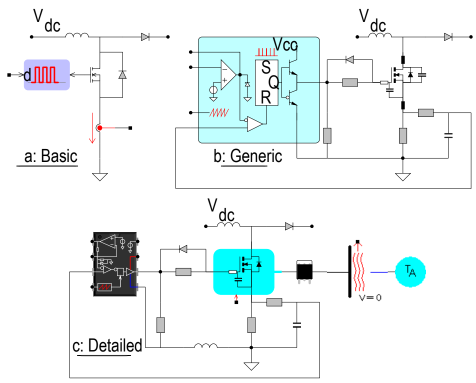

Fig. 1. Modeling approaches for SMPS. a:Basic model with constant duty-cycle and ideal current measurement, b:Generic peak-current-mode control model and inclusion of parasitic components around the semiconductor, c:Detailed semiconductor and control IC model [21].

III. Virtual Workbench

A virtual workbench is basically modeling and simulation. However the question is, if the model should be as simple and general as possible with less parameters, or if it should look as close as possible equal to the final device. The hierarchical approach is split over three types of models:

- Basic ideal model
- Generic functionality model
- Detailed component model

The followed approach, is to start with a simple basic ideal model, with a limited set of parameters. Gradually, the level of detail should be improved, using generic models, to finally have a model, based on real component specification.

It should however be emphasized that detailed control design, power loss estimation or EMI can be evaluated from each of the three model approaches. Figure 1 shows the differences between the various approaches.

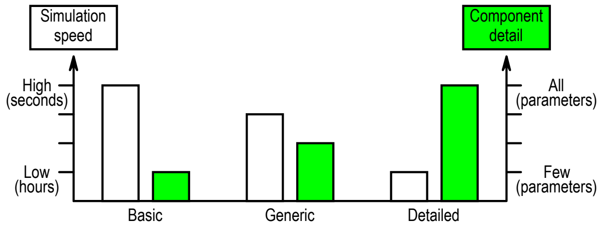

Fig. 2. Simulation speed decreases, while component detail increases, when going from a Basic towards a Detailed model.

In figure 1-a, a basic model is shown, where only an ideal switch model represents the semiconductor. It is either on or off, but has a series resistor, to model the conductance losses. If the current through the semiconductor is required in a peak-current-mode control [16], it can be measured using a simple ideal current sensor. Especially in the basic ideal model, we can extensively use the advantage of the multilevel modeling approach, in that we model the control circuitry using a block-diagram, where only signals are modeled [9]. In figure 1-a, the gate of the Mosfet circuit model, is directly controlled from a block-diagram signal [6].

In figure 1-b, the generic model is shown. Although this is still based on ideal models for semiconductors, the controllers and especially the analog controllers are modeled using a generic model. Such a generic model includes typical functionality of SMPS control IC's, such as build-in oscillator, current limitation and an operational amplifier, with correctly defined bandwidth and gain. Typical control IC's for SMPS for voltage mode and peak-current-mode, being single or dual ended, share a large common set, of functionality and behavior. The semiconductor models for the Mosfet, diode or IGBT, are in principle the basic ideal models, but now they are extended with the most important parasitic components that influence their behavior and the waveforms during switching. For example, the lead inductances are mainly responsible for transient voltage peaks [14]. The junction capacitors C_{GS}, C_{GD}(not drawn in the symbol) and C_{DS} define the charge required during switching as well as the switching losses [14].

Figure 1-c, shows the detailed component model. Here the typical component circuit models are employed, which contain the electrical specifications for that typical IC. Compared to the generic models, the oscillator is modeled inside the detailed model, in the same way it is operating in the real IC. It means that you can do the same analog circuit tricks, like using this signal for control purposes, such as Constant-On-Time control, or add external triggering or synchronization. Also the fan-out of the gate-driver is modeled, as well as the external charge pump circuit components, that have influence on the transient behavior. The semiconductor switch, is modeled using a non-linear relation between the gate-source voltage and drain-source current, to model the non-linear transconductance of the semiconductor.

Surprisingly, you can model most of the typical engineering questions and design issues with each of the three modeling approaches. Table I shows the three approaches and their strength per application.

TABLE I
ENGINEERING QUESTIONS AND DESIGN ISSUES DEPENDENCY ON THE MODELING APPROACH

	open-loop	closed-loop	Power Loss	EMI
Basic	++	+	-	-
Generic	+	++	0	0
Detailed	+	0	++	+

In table II, the modeling approaches are shown for typical engineering questions and design issues. Although in principle each modeling approach can be used for each topic, there are differences in the results and what is sought for. For example, the waveforms for open-loop or closed-loop control can be simulated with any of the approaches, but the basic simple model will have the shortest simulation time. For EMI, the generic model will already show the most important high frequency transients, where the basic model can only show the low frequency harmonics. The detailed model will not reveal more information compared to the generic model, when it comes to the transients, but will require a much longer simulation time. The power loss is best evaluated using the detailed model, as it includes the detailed non-linear behavior, during switching of the semiconductors. Depending on what is expected to be investigated using modeling and simulation, a choice has to be made, among the three modeling approaches.

Figure 2, shows the differences in component detail and simulation speed for the three approaches. Clearly visible is the high simulation speed for the basic model, compared to the detailed model. However, the detailed model will show more details in the waveforms, but requires more simulation time.

IV. Basic modeling approach

The basic model for the boost converter simulation, is shown in figure 3. The purpose of this simulation, is to reveal the start up behavior and the maximum inductor current, if the boost converter is started with a constant dutycycle. Also the inductor current ripple and the influence of the equivalent series resistor R_{esr} of the capacitor C_1, can be studied using this simulation. The maximum current during start up is around three times the maximum inductor current for nominal operation. The duty cycle is set to the ideal value of $d = 0.6$

$$\frac{V_{out}}{V_{in}} = \frac{1}{1-d} \tag{1}$$

If an output voltage of 48 volt is expected, the duty cycle should be set a little bit higher, to compensate for the losses due to the voltage drops, in the boost converter. A value of $d = 0.6$ is used, and ideally we expect:

$$\frac{V_{out}}{V_{in}} = \frac{1}{1-0.6} = \frac{1}{0.4} = 2.5 \tag{2}$$

$$V_{out}^{ideal} = 2.5 \cdot 20 = 50\text{volt} \tag{3}$$

Since the output voltage in the simulation is less than 48 volts, it shows that using the basic simulation, the voltage drop over the series resistance and output diode, are already modeled.

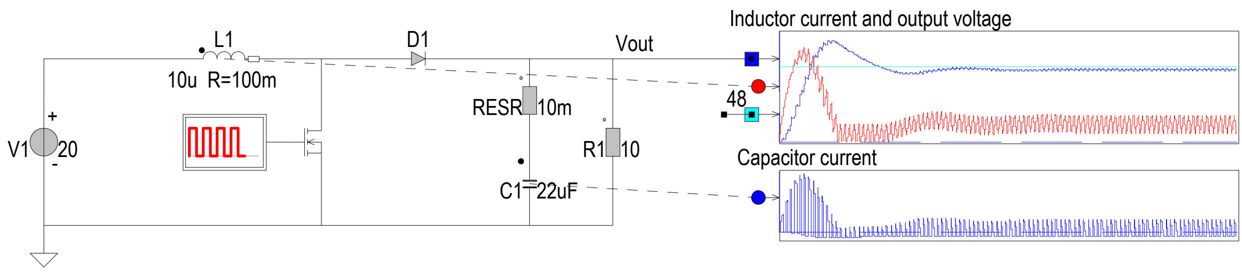

Fig. 3. Basic model for the open-loop simulation(Total simulation time $T = 1ms$, step-size $dt = 100ns$) of the Boost converter [21].

TABLE II
SIMULATION GOALS

	Basic	Generic	Detailed
Control	• Steady state Behavior, • Start up,Stability	• Control limits, • Stability	• Component non-linearity
Losses	• Conduction, • Inductor winding loss, • Capacitor ESR loss	• Conduction, • Inductor winding loss, • Capacitor ESR loss	• Conduction, Switching, • Inductor winding loss, • Core loss approximation, • Capacitor ESR loss
EMI	• Harmonics from the switching waveforms	• Harmonics from the switching waveforms and parasitic components	• EMI from the parasitic components combined with dynamics from semiconductors

In the basic model approach, these conduction losses are already modeled, by simply inserting series resistance in the circuit. The inductor has a series resistance of $100m\Omega$, the Mosfet has series resistance $R_{DS} = 100m\Omega$ modeled, as well as the forward voltage drop of the diode, being equal to around 0.7 volt. The conduction losses in the output capacitor are modeled by the series resistance $R_{esr} = 10m\Omega$.

Since the Mosfet model is ideal, the simulation speed is high, but the only parameter used for modeling the Mosfet, is the on-state resistance R_{DS}. But although we use simple models, we can already study the harmonic content in the input current, being equal to the inductor current. Also the ripple in the output capacitor, and thus in the output voltage, is simulated, showing the influence of the series ESR of the capacitor. Adding capacitors with different ESR in parallel will directly show the influence on the output voltage ripple. Although a control could be added in this simulation, the generic model approach is better suited for this as is outlined in section V.

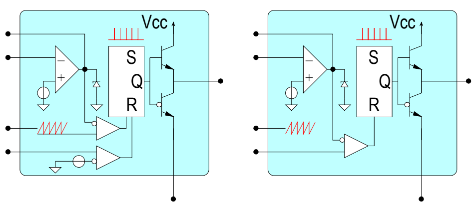

Fig. 4. Generic models for voltage mode(left) and current mode(right) controllers [21].

V. Generic Modeling Approach

Closing the loop can be done using analog or digital control. In this simulation example, we choose a peak-current-mode control IC. There are various types from different manufacturers, but internally they show the same functioning, see figure 4. Important when closing the analog control-loop, is the compensation network and bandwidth of the Opamp from the control IC. The current sense filtering and slope compensation, are added to the simulation.

Since the series resistance of the inductor and capacitor are modeled, the output voltage ripple as well as the conduction losses are included in the generic modeling approach. The poles and zeros of the control to output small signal transfer function are modeled in the generic model, and therefore, effects of the loop compensation circuit can be studied. Measuring the inductor current for the peak-current-mode control, is done using a simple sense resistor, and an additional low pass filter. These simple components in combination with the current sense input pin of the generic IC model, will give adequate information on the shape of the waveform of the measured current.

The non-linear behavior of the Mosfet has little influence on the stability and transient response of the closed-loop control, and is therefore omitted in the generic model. However, parasitic wire inductance and junction capacitance can be added, to model some of the transients during switching. Since the generic Mosfet model already includes the gate to source capacitance C_{GS}, the influence of the gating resistor can be

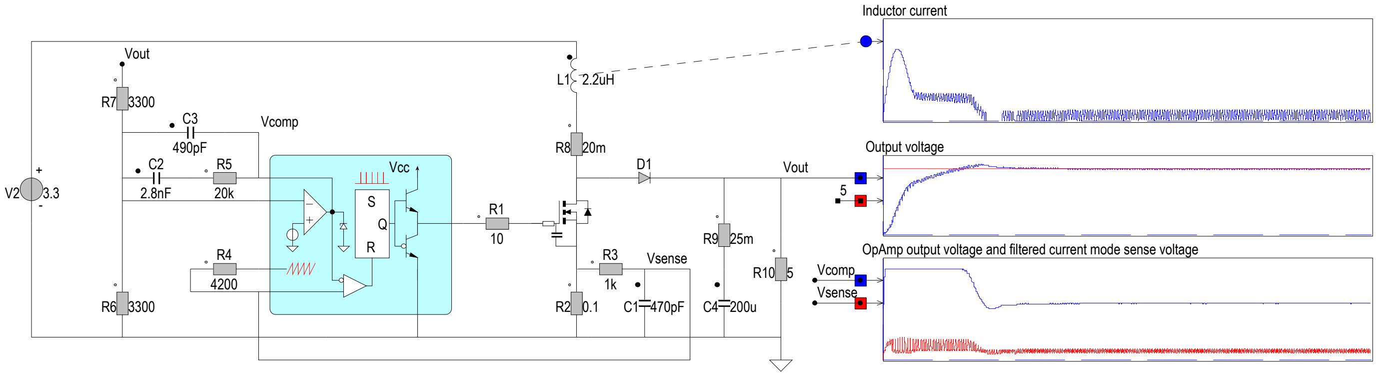

Fig. 5. Generic model for the peak-current-mode control and semiconductor in the simulation(Total simulation time $T = 1ms$, Step-size $dt = 100ns$) of the Boost converter [21].

studied to some extend. For a more detailed simulation of the influence of the gating resistance on the transient switching performance of the Mosfet, the detailed modeling approach as outlined in section VI is required. Since there are more parameters to be included in the simulation compared to the basic simulation, the model is more accurate when it comes to transient behavior, but also requires more simulation time. This is visible in figure 2, where the number of parameters of the model is increased compared to the ideal model. Because of the more refined control model, the simulation speed for the generic model, is decreased compared to the basic model.

Figure 5 shows a boost converter with peak-current-mode control. The closed-loop control is modeled using a generic peak-current-mode model.

A single ended peak-current-mode control IC, is directly driving a Mosfet via a gating resistor. The Mosfet model includes the gate-source capacitance C_{GS} and internal gate resistor $R_{Gate}^{internal}$, as well as the threshold voltage level V_{TH}. The current is measured through the sense resistor R_2. The low-pass filter, build from R_3 and C_1, filters the transient response and removes the sharp spikes, that could lead to wrong current measurements. The losses in the current sense resistor, $I_{DS}^2 \cdot R_2$ are modeled, and the delay of the low-pass filter are thus included in the simulation. To compensate for sub-harmonic oscillation, slope compensation [16] is added in the simulation by R_4. The oscillator pin from the generic peak-current-mode control IC provides a sawtooth voltage, as commonly seen in current mode IC circuits.

The closed-loop feedback circuitry, is a type II, modeled by the components R_5, C_2 and C_3 [15]–[17]. Optimizing the control and changing the pole locations, directly shows the influence on the start-up behavior of the circuit.

The generic model approach in the simulation in figure 5, is especially suited for testing the closed-loop control. It includes all components that have to be designed, and has just enough parameters, like bandwidth and gain in the generic control IC model. It is therefore ideally suited for an undergraduate course, on SMPS control.

VI. Detailed Modeling Approach

To study the influence of the parasitic inductance in the circuit, the rise and fall times for the current in the Mosfet have to be simulated. For this, the detailed Mosfet is used. Here the non-linear behavior of turn-on and turn-off, of the Mosfet is included. It depends on the non-linear relations between V_{GS}, V_{DS} and I_{DS} and the temperature of the junction. Small inductors, modeling short PCB traces of around $10mm$, in the size of typically $10nH$ to $20nH$ per lead [14], are added in the simulation. Over these parasitic inductors, over-voltages during switching are simulated.

The peak-current-mode control IC is included in the detailed simulation, which includes the details of the internal oscillator circuit and its dependency on the external components around the oscillator pin. Slope compensation and the influence of the slope compensation resistor R_4 on the oscillator voltage V_{rc}, are included in this detailed simulation.

The transient behavior as revealed by the simulation of the detailed model, would be equal to a simulation using the generic modeling approach. However, in the detailed simulation, the voltage peaks during switching are visible in the simulation results.

The thermal behavior is included, by adding a thermal model for the Mosfet package. The temperature on the junction is simulated, and influences the on-state resistance R_{DS} of the Mosfet. The detailed model is typically applied in undergraduate and graduate design courses, where the emphasis is on evaluating the influence of the various design details in the experimental results, based on simulation results.

Conclusions

Applying simulation in undergraduate and graduate courses, requires an hierarchical approach, in order to correctly match the complexity of the simulation model, to the knowledge level of the student. Starting with basic models in second year undergraduate courses, and via generic models for second and third year undergraduate, detailed models are applied in last year undergraduate and second year graduate courses.

Instead of focusing on a single model or simulation method, the idea is to use a different type of model, depending on

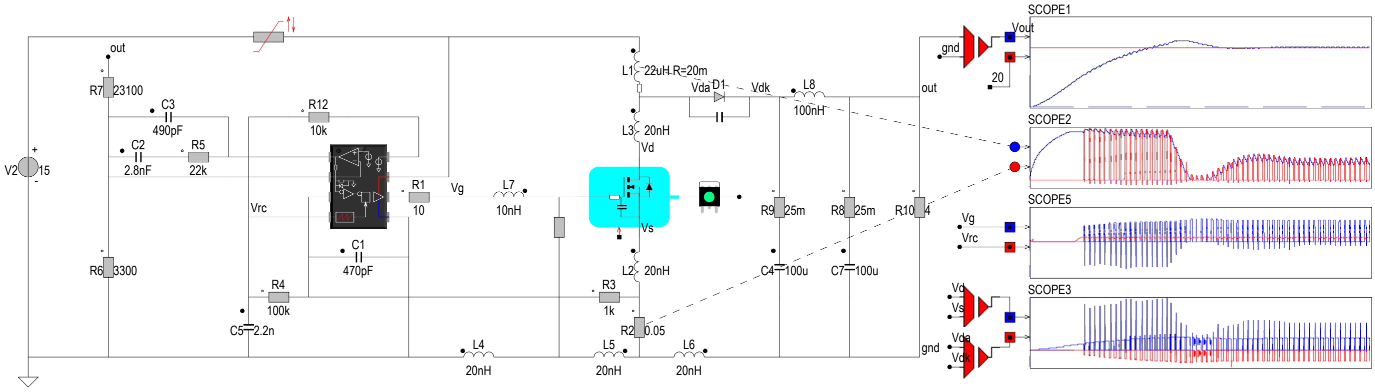

Fig. 6. Detailed model for the peak-current-mode control IC and Mosfet in the simulation(Total simulation time $T = 1ms$, Step-size $dt = 10ns$) of the Boost converter [21].

the needs for simulation results. A basic model would show just enough information regarding the voltage and current waveforms. A generic modeling approach is to be applied when the control has to be designed, while a detailed model is used when thermal issues and EMI have to be simulated.

Depending on what simulation result is required, a modeling approach has to be selected. The basic model will reveal the currents and voltage waveforms, including the influence of series resistance of the passive and active components. Only conduction losses can be simulated using the basic model. The generic model is used when simulating closed-loop behavior. Feedback compensation circuitry can be studied using the generic model. If the influence of parasitic components has to be studied in detail, a detailed semiconductor model is required, where the switching delay is model.

The main advantage for education of the hierarchical approach, is that only those effects are modeled and simulated that are of interest. Students only have to supply a minimum set of parameters, if not all simulation details are required, and will be able to correlate the simulation results due to parameters variations.

References

[1] L. W. Nagel and D. O. Pederson, "Simulation Program with Integrated Circuit Emphasis (SPICE)," presented at 16th Midwest Symp on Circuit Theory, Ontario, Canada, April 12, 1973. [Online]. Available: http://www.eecs.berkeley.edu/Pubs/TechRpts/1973/22871.html

[2] C. C. McAndrew, "How Come SPICE Is a Verb?: The Natural Course of a Diverse Engineering Tool," in IEEE Solid-State Circuits Magazine, vol. 11, no. 1, pp. 14-18, Winter 2019, doi: 10.1109/MSSC.2018.2882279.

[3] H. A. Owen, A. Capel and J. G. Ferrante, "Simulation and analysis methods for sampled power electronic systems," 1976 IEEE Power Electronics Specialists Conference, Cleveland, OH, USA, 1976, pp. 45-55, doi: 10.1109/PESC.1976.7072897.

[4] Chung-Wen Ho, A. Ruehli and P. Brennan, "The modified nodal approach to network analysis," in IEEE Transactions on Circuits and Systems, vol. 22, no. 6, pp. 504-509, June 1975, doi: 10.1109/TCS.1975.1084079.

[5] G. W. Wester, "Power electronics modeling, design and simulation session summary," 1975 IEEE Power Electronics Specialists Conference, Culver City, CA, USA, 1975, pp. 62-62, doi: 10.1109/PESC.1975.7085568.

[6] P. J. van Duijsen, "Multilevel modeling and simulation of power electronic systems," 1993 Fifth European Conference on Power Electronics and Applications, Brighton, UK, 1993, pp. 347-352 vol.4.

[7] P. Bauer and P. J. van Duijsen, "Large signal and small signal modeling techniques for AC-AC power converters," Conference Record of the Power Conversion Conference - Yokohama 1993, Yokohama, Japan, 1993, pp. 520-525, doi: 10.1109/PCCON.1993.264201.

[8] G. A. Franz, "Multilevel simulation tools for power converters," Fifth Annual Proceedings on Applied Power Electronics Conference and Exposition, Los Angeles, CA, USA, 1990, pp. 629-633, doi: 10.1109/APEC.1990.66362.

[9] P.J. van Duijsen, "Multilevel modeling and simulation of power electronic converters and drive systems.", Proceedings of the PCIM 1994, Nuremberg, Geermany, 1994

[10] O. Apeldoorn, "Simulation in power electronics," Proceedings of IEEE International Symposium on Industrial Electronics, Warsaw, Poland, 1996, pp. 590-595 vol.2, doi: 10.1109/ISIE.1996.551009.

[11] P. Bauer and P. J. Van Duijsen, "Challenges and Advances in Simulation," 2005 IEEE 36th Power Electronics Specialists Conference, Dresden, Germany, 2005, pp. 1030-1036, doi: 10.1109/PESC.2005.1581755.

[12] P. J. van Duijsen and D. C. Zuidervliet, "Structuring a Switched Mode Power Supply Course, Part I: Lectures," 2022 45th Jubilee International Convention on Information, Communication and Electronic Technology (MIPRO), Opatija, Croatia, 2022, pp. 1319-1324, doi: 10.23919/MIPRO55190.2022.9803707.

[13] P. J. Van Duijsen and D. C. Zuidervliet, "Structuring a Switched Mode Power Supply Course, Part II: Laboratory," 2022 45th Jubilee International Convention on Information, Communication and Electronic Technology (MIPRO), Opatija, Croatia, 2022, pp. 1325-1330, doi: 10.23919/MIPRO55190.2022.9803446.

[14] N. Mohan, T.M. Undeland, W.P. Robbins, Power Electronics: Converters, Applications, and Design 3rd Edition, Wiley 2002, ISBN:978-0471226932

[15] S. Ang, A. Oliva, Power-Switching Converters, 3rd ed. CRC Press 2011, ISBN:978-1439815335

[16] M.K. Kazimierczuk, Pulse-Width Modulated DC-DC Power Converters 2nd Edition, Wiley 2015, ISBN:978-1119009542

[17] J.G. Kassakian, M.R. Verghese, G.C. Schlecht, Principles of Power Electronics, Addison-Wesley 1991, ISBN:978-0201096897

[18] R.W. Erickson, D. Maksimovic, Fundamentals of Power Electronics 3rd ed., Springer 2020, ISBN:978-3030438791

[19] J. Pollefliet, Electronic Power Control: 1 Power Electronics, 8th Edition, Lannoo Publishers, 2019, ISBN:978-9038225258

[20] J. Pollefliet, Electronic Power Control: 2 Electronic Motor Control, 8th Edition, Lannoo Publishers, 2019, ISBN:978-9038225265

[21] Caspoc Simulation and Animation, Simulation Research, [Online] Available: https://www.caspoc.com,

[22] DC laboratory, The Hague University, TIS, Delft. [Online] Available: http://dc-lab.org