PC IM, 13-23, 1999

Simulation of Power Conversion Systems: From the State of the Art to Future Trends

Sam Ben-Yaakov

Power Electronics Laboratory
Department of Electrical and Computer Engineering
Ben-Gurion University of the Negev
P. O. Box 653, Beer-Sheva 84105, ISRAEL

Tel: +972-7-646-1561; Fax: +972-7-647-2949;
Email: sby@bguee.ee.bgu.ac.il; Web Site: http://www.ee.bgu.ac.il/~pel

Abstract - This overview presents the capabilities of current simulation tools when applied to power conversion systems. Two basic approaches are discussed and explained while pointing out to their usefulness and shortcomings: cycle-by-cycle simulation, and behavioral average simulation. It is shown that these techniques, and in particular the latter, can help the designer in optimizing PWM and resonant converters systems and handle intricate problems such as feedback design and dynamic stability analysis. The review also points to some new approaches that are just being developed such as envelope average simulation. Finally, some observation on the future of power conversion system simulation is offered and argued.

1. INTRODUCTION

Since the introduction of SPICE (Simulation Program with Integrated Circuit Emphasis) by Berkeley university researchers in the early seventies [1], analog circuit simulation made great strides toward becoming a one of the important tools in analog circuit design. In the area of Power Electronics, penetration of simulation is especially slow. This has many objective reasons but also some underlying psychological barriers. The objective situation is that Power Electronics circuits and systems are hard to simulate. In fact, the simulation domain is not different from the physical environment. When you have a simple system it can be simulated easily. Power electronics systems are not only complex but are highly dependent on subtle effects: interconnection, ground loops, capacitive and magnetic couplings, parasitic leakages and inductances and many other problems that turned the hair of many designers white. Notwithstanding the difficulties, simulation tools are slow to penetrate in the Power Electronics area partially because many esigners look at these tools in suspicious. However, as we approach the millennium, application of simulation in Power Electronics is no longer a curiosity. The tools and methodologies are maturing and become efficient and hence can shorten the time to market. More and more power electronics designers discover the marvels of simulation and many companies initiate educational programs to help the fast assimilation of these tools in their R&D activity.

The objective of this paper is to provide an overview of the state of the art of simulation in Power Conversion technology, stressing in particular the main immediate contributions that simulation can make to product development. The attached reference list includes most relevant papers by the author in the area of simulation. An extended bibliography, as well as additional valuable relevant material, can found at the Web sites of Dr. V. Bello, the early pioneer of switching regulators simulation by average SPICE models (http://SpiceSim.com; http://Members.AOL.com/DrVGB).

2. THE TOOLS

Most Power Electronics simulation are presently based on general purpose SPICE based simulators. The core of most commercial packages is still the SPICE algorithms [1] to which many improvement have been made in the last decade. These include: new device models, new numerical algorithms, better and user friendly interfaces for circuit entry (e.g. schematic capture) and graphical examination of the results. Power Electronic systems can be simulated by general purpose packages directly and/or by applying descriptive modules (to be discussed in detail below). Other, more specialized simulation packages are starting to appear on the market (as of early 1999). These however are still in the infancy and will not be discussed in this review.

All SPICE based simulators include the major three analyses [2, 3]: steady state or bias point evaluation for one or several excitations (DC), small signal analysis in the frequency domain which include as a first stage linearization around the operating point (AC), and time domain analysis (TRAN). Each model of a given simulation package will be compatible with the three types of analysis. Further, the simulator will have, built in, a linearization procedure which is transparent to the user, to derive the model signal equivalent circuit of the system under test. Hence, once a circuit is fed to a simulator, the user can pursue any or all of the three basic analyses (DC, AC, TRAN). In addition all simulators include many analysis options which are built around the major three analyses. These include: sensitivity analysis, effect of component variation based on some statistical distribution (e.g. Monte Carlo) and many other options [2, 3]. Clearly, a detailed discussion of all the simulation options is beyond the scope of this paper.

Although the main focus of this review is on SPICE oriented simulation, we also cover some aspects of MATLAB (The MathWorks, Inc) simulation of Power Conversion systems. We refrain, as much as possible, from mentioning commercial products in order to keep impartial. At present (early 1999) though the main general packages that are being used Power Electronics professionals (judged by popularity in journal and conference publications) are IsSPICE (Intusoft, USA) and PSPICE (MicroSim, USA). This by no means implies necessarily that these simulators are better than other commercial products. It is important to emphasize that in most cases the type of simulator used is not an important factor in achieving good simulation results. However, although branching from the same root (Berkeley's SPICE), today's analog circuit simulators are not compatible anymore. Hence, an entry prepared for one simulator will not run on another. Further, device library of one vendor may not run on another simulator. This unfortunate situation is one more reason for the slow development of analog circuit in general and Power Electronics simulation in particular.

3. TYPES OF POWER ELECTRONIC SIMULATION

The main analysis and simulation options, available today, are summarized in the block diagram of Fig. 1 [4]. The right arm (cycle-by-cycle) simulation is the conventional analysis of the circuit in its original form that can be carried out to obtain the detailed time domain response. This can be done by SPICE or any other general purpose electronic circuit oriented software package. The average modeling approach shown on the left arm, can be proceeded by any one of three possible options that can be combined for a comprehensive treatment of a given problem [4].

3.1 CYCLE-BY-CYCLE SIMULATION

This analysis is the simplest and seemingly the straightforward approach [5]. In this case, a complete description of the circuit is given, and a time domain numerical calculation of the (non-linear) system equation is carried out by the simulator. Here, again, one has several options. To explain these we consider here a Boost Power stage (Fig. 2a). The straightforward approach is to simulate the actual circuit as given. This will produce the time domain behavior of the system. For the major parameters of the circuit (such as the inductor current), the characteristics of the transistors and diode are not that important. In fact, to observe the main features one can replace the transistor by a switch (Fig. 2b) and get just about the same results. In most cases, the nature of the primary processes (such as the inductor current) are normally of little use for the designer who is well aware of these waveforms. More interesting are the salient effects such as the effects of the forward and reverse recovery of the diode, expected spikes without and with a given snubber, conduction and switching losses etc. As would be obvious to workers in the field, some of these can not be probed by the simple representation of the basic circuit (Fig. 2a). For example, leakage inductance can not be estimated before a Printed Circuit Board (PCB) layout is actually worked out. Further, for accurate simulation of some parameters there is a need for an accurate model of semiconductor (both discrete and VLSI) devices. Accurate analysis will require an advanced model of the transistor that includes the non linearity of its parasitic capacitances, a good model of the diode that shows the reverse recovery process, a good model of the driver to be used to emulate the dynamic behavior of the source during turn on and off, and good representation of the parasitic and capacitive leakages. Hence even if the device models are perfect, replacing the breadboard experiment by simulation is certainly not a simple task. Device models are indeed improving constantly, showing a better and better match to the real physical devices. However, given the dependence of power electronics systems on parasitic effects and nonlinear effects which are complex and highly depend on operating conditions (e.g. current, voltage, frequency, temperature, layout etc.) simulation can not substitute at present the breadboard testing. Adding to the above the fact that time domain simulation is very lengthy, it is clear that application of simulation in Power Electronics is far from being straightforward or simple.

As an example to a useful localized application of a cycle-by-cycle we consider here the case of a resonant power stage: Current Sourcing Push Pull Resonant Inverter (CS-PPRI) presented earlier [6] and found applications in MHz Ballast design [7]. The objective of the simulation presented here is to achieve an understanding of the processes in the inverter, to map the expected waveforms and to study the dependence of the output current on the load resistance. Atypical input to a simulator is shown in Fig. 3. The diagram is compatible with PSPICE (MicroSim, USA) but other simulators could be used. Note, however that input to each simulator will be different and the files created are incompatible. The circuit diagram of the inverter is fed as-is except for the switches which are described by an on-off device Sbreak (PSPICE, MicroSim Co.). Typical waveforms for a switching frequency of 200kHz and 50% duty cycle are shown in Fig. 4a. A more sophisticated study of the circuit can be achieved by using some of the option built in all circuit simulators. For example, by applying the 'parametric analysis' feature of PSPICE one can obtain the dependence of the output current as a function of the load resistance (Fig. 4b). Other options can provide some other useful information.

The example given above clearly points to the usefulness of simulation. The cycle-by-cycle simulation results do not provide the full picture and certainly leave many issues unanswered e.g. efficiency, drive requirements etc. Nonetheless, the information obtained is non-trivial and certainly helpful.

3.2 AVERAGE SIMULATION

As pointed out above, one of the main shortcomings of cycle-by-cycle simulation is its occupation with the fine details of the waveforms within each switching period. This not only consume long computer time but detracts in fact the attention from the global behavior. For this system aspect, the designer would be more interested in the average values of the voltages and currents rather than in the ripple effect. The idea of average modeling is not new. It dates back to the pioneering work of Middlebrook and Cuk [8] that applied state space averaging to extract the linearized small signal response of switch mode systems. The averaging concept is very powerful, allowing the designer to deal with the fundamental performance issues: output voltages and currents, transient and small signal responses, dynamic stability and interactions between systems. Consequently this simulation approach offers the designer a useful tool that provides vital design information in the virtual environment without having to build the system. Considering the practical importance of average simulation and the unfamiliarity of many designers with the concept, some background information is in order. In this review paper we focus on the methods developed by the research group of the author. The method is based on the equivalent circuit approach which replaces the switching parts of the system by continuous average circuits. This makes the models automatically SPICE compatible. Other approaches are possible, however, and the reader will find references to relevant papers in the attached bibliography. The main feature of the methodology presented here is the systematic treatment of all PWM cases: voltage mode, peak current mode and average current modes in both continuous and discontinuous inductor current cases.

The equivalent circuit approach for modeling switch mode systems presented here, hinges on replacing the switching part of the converter by a low frequency, or "average" Switched Inductor Model (SIM) and emulating the function of the Duty Cycle modulator. That is, the power conversion system is envisioned as composing of three parts (Fig. 5): the SIM, the Duty Cycle Generator (DCG) and the rest of the circuit (passive and active elements) that is then left as is, being already compatible with SPICE based simulation programs. The method makes use of behavioral dependent sources now available on practically all simulators. The behavioral dependent source could be a current or voltage source whose output is a function on other variables (currents or voltages) of the circuit.

The Switched Inductor Model (SIM). Close examination of the power stage of common PWM topologies reveals that they all include an inductor which is switched at one end between two points [9]. The switching action is normally carried out by a transistor and a steering diode. The net behavior, however, is that of a switch which toggles the inductor between the two end points (Fig. 6). The Switched Inductor Model (SIM) depicted in Fig. 7 replaces this switching part by an equivalent circuit, using dependent sources, to emulate the average behavior of the three terminals. Hence, the objective of the *Average Equivalent Circuit* approach would be to replace this module by an equivalent circuit, such that the *average* voltages seen across the inductor and the *average* currents flowing through terminals (a), (b) and (c) (Fig. 7), will remain the same as in the physical system. The expressions for the dependent sources, for the general case of continuous and discontinuous conduction modes (CCM and DCM) are given in [4, 10, 11].

As already shown, the SIM approach can be extended to peak and average current mode [12, 13], to quasi resonant converters [14,15] and to PWM based Magamp stabilizers [16]. For the sake of brevity only general aspects are covered here.

Duty-Cycle Generator (DCG). Applying the behavioral concept, the next stage is to emulate the Duty-Cycle Generator (DCG). This could be for voltage mode, peak current mode, average current mode, in CCM or DCM. Since the relationships involved are algebraic, the block that emulate the DCG can be based on a behavioral dependent source. Details are found in [4, 10-13, 17-22]

Compatibility with general purpose circuit simulator is obtained by replacing the inductor and switches by the SIM equivalent circuit and by defining the Duty Cycle Generators and coding the duty cycle into voltage [9, 22]. The rest of the circuit is left as it is. The example of Fig. 8a describes the SPICE compatible equivalent circuit for a buck converter, operating in CCM and DCM modes[4]. Fig. 8b demonstrates implementation in PSPICE Versions 8 or 7. A demonstration of the average model behavior relative to the actual operation of the switched circuit is shown in Fig. 9. The results of the average model simulation follow, accurately and smoothly, the average values of the rippled waveforms obtained by a cycle-by-cycle simulation. The speed up ratio of the average model simulation was found to be more than of 100 folds. The major benefit of the averaging technique is in the ability to linearize the model, which is done automatically by SPICE (in AC analysis), and to get the small signal transfer functions for frequencies lower than half of the switching frequency.

As evident from the above the main advantages of the topology independent models based on the SIM concept is the fact that they can be defined as subcircuits and directly applied to run DC, small signal (AC) and large signal (TRAN) analysis in open and closed loop configurations. A major benefit is the ability to examine the loop-gain of the system [23,24] and hence help design the feedback loop. The SIM approach can be extended to peak and average current mode [12,13], to quasi resonant converters [14,15], to PWM based Magamp stabilizers [16], to SEPIC converters [25,26] and in fact to any PWM topology.

3.3 ACCURACY OF DEVICES AND SYSTEM MODELS

When applying cycle-by-cycle or average simulation one needs to present active and passive devices by proper SPICE compatible models. The issue is less of a problem in average simulation which is normally carried out under the assumption of "ideal" components. Nonetheless, even in average simulation, some non ideality aspects can be easily incorporated such as: voltage drops on diodes and transistors and resistances of inductors and capacitors [22]. In the case of cycle-by-cycle simulation the situation is much more complex. If 'ideal' elements are used then the simulation provides only the pure theoretical solution (which by itself could be useful). If more intimate details are of interest (e.g. waveforms of MOSFET's gate voltage and current), the situation is much more complex. In this case one needs not only accurate device models but also a good representation of the interconnection (parasitic inductances, capacitances, resistances and coupling effects). It is the opinion of the author that at this stage of the game such very detailed simulation may not very practical in the general case. In fact, it may prove a colossal waste of time adding very little new knowledge. Yet, cycle-by-cycle simulation could be useful to study specific effects (e.g. effect of leakage inductance in gate-drive transformer) to gain understanding of the problem.

Magnetics pose a special problem when it comes to precise cycle-by-cycle simulation. In most cases, the complete transformer's model is unavailable. Consequently, there is no possibility at this time to completely replace experimental measurements by simulation when it comes to systems that include magnetics. Nonetheless, as designers did in the past, approximate analysis could provide the first iteration stage. The advantage of simulation is that the procedure is fast and less prone to human errors.

Cycle-by-cycle simulation by 'ideal' element does not require by itself additional verification since it is just a mathematical computation of network's performance. What would require an experimental verification is a claim (if made explicitly or implicitly) that the non-ideality of the elements will not change the basic performance. For example, suppose that a lossless snubber or a dual switch soft switcher is proposed and demonstrated by simulation [27-29]. Here non-ideal effects could be crucial (reverse recovery of the diodes) and hence the simulation will have to be verified by detailed experiments. On the other hand, if we consider the simulation example of Figs. 4a-b. The information obtained is very good and could be a basis for the design of the resonant elements. This is due to the fact that the simulation is used to probe into the main features of the inverter rather than the very fine details (e.g. the waveform of the transistors' current).

In average simulation we have again the situation that uncertainty is related to the model used in building the average equivalent circuit. Here one can use cycle-by-cycle simulation to verify the average simulation results. Computer time in cycle-by-cycle simulation is long but a good option when it comes to validate average simulation. An example of the degree of matching that is obtained is shown in Fig. 9. Similar matching is found in ac analysis (Fig. 10) used to verify an average model of SEPIC converter [25,26]. Although cycle-by-cycle verification of average simulation is sufficient, it is reassuring that experimental measurement also verify the validity and accuracy of average simulation. An example of such a comparison is given in Fig. 11 [30].

4. EXTENSION TO RESONANT CONVERTERS

The basic idea of behavioral average simulation can be extended to resonant converters by replacing, again, the switching part by continuous equivalent circuits. In this case, the modeling methodology hinges on the ideas put forward by Steigerwald [31]. Following his reasoning, the basic operation of a resonant converter, such as a series-parallel converter (Fig. 12), can be represented by a damped resonant network (Fig. 13). In this representation the virtual AC resistor (R_{ac}) expresses the effect of the dissipative nature of the load (R_{out} , Figs. 12, 13) on the resonant circuit. The value of (R_{ac}), under steady state conditions, can be obtained by equating the AC power dissipated by it to the DC power delivered to the load (R_{out}). Further details are found in [31-33].

5. LINKS TO MATLAB

Switch mode circuits are in fact non linear control systems. Hence, it would be beneficial to examine their behavior by general purpose software packages which were especially designed for simulating dynamic systems (such as MATLAB, The MathWorks, Inc.). The difference between this approach and that of the analog circuit simulation is two fold:

1. MATLAB and other similar simulators are system rather than circuit simulators. Hence, the system aspects of the problem can be illuminated and better understood when such a tool is applied for studying switch mode systems.
2. MATLAB type simulators are Discrete Domain simulators. They are extremely powerful tools when applied for studying and designing digital controllers, a task which is incompatible with analog simulators.

In the approach developed by the author's group, discrete simulators are used to examine the average behavior of switch mode systems rather than the cycle-by-cycle response. The advantage of the discrete simulators, such as MATLAB, is their compatibility with discrete control laws including Fuzzy Logic and Neural Networks. This simplifies the exploration of novel control techniques by being able to remain within the same numerical package.

We have used SIMULINK, the graphical tool of MATLAB to define a MATLAB compatible block diagram of the Switched Inductor Model (SIM) [4]. By adding the block that describes the function of the Duty-Cycle Generator [22], a complete SIMULINK representation of a given topology can be easily derived [4]. The SIMULINK compatible representation can be used to analyze and run simulations of a switch mode system under study and can be further explored by any of the available control toolboxes of MATLAB for various types of control design. As an example to MATLAB simulation [4], we present the benchmark current mode Buck converter published earlier [18] (Fig. 14). The converter was analyzed here in closed loop configuration for various values of current loop gains. The results of the small signal frequency domain analysis (Fig. 15) were found to be in a very good agreement with earlier results which applied other analysis and simulation techniques [12, 18, 22].

6. EDUCATION

Two educational aspects need to be considered in relation to simulation of power conversion system: educating the practicing engineer and application of simulation in engineering education. Concerning the first aspect, no question that the proliferation of simulation in the power electronics industry is due in part to the unfamiliarity of designers with simulation tools and techniques. Experience has shown that this can be easily corrected by having the designers attend continued education seminars and courses on the subject. Some companies resort to the mode on in-house seminars which are very effective.

The other aspect of education, that of applying simulation in power electronics courses was found to be extremely useful [4]. This is in part due to the fact that theoretical power electronics courses fail to bring across practical knowledge that are normally equated with "experience". Simulation can bridge the gap in some respect making the courses closer to the real thing.

7. FUTURE TRENDS

The ideal situation would be able to simulate a complete power conversion system including the precise device behavior, layout effects, parasitic coupling, conducted and radiated EMI as well as efficiency and protection. Obviously, we are still far from this target and much work is needed to get closer to it. The main thrust of simulation related research will be put in the near future in improving the device (including magnetic) and average models. New and improved average simulation techniques that are transparent to the high frequency component will shorten simulation time especially when analyzing large systems. Already under investigation are new average simulation techniques that permit envelope simulation of system that are excited by modulated analog signals (amplitude modulation, frequency modulation and phase modulation). This and other simulation techniques will open the simulation approach to many more engineering applications.

It is the opinion of the writer that the main practical near future uses of simulation in power electronics will be in average simulation on one hand and cycle-by-cycle simulation of localized isolated problems on the other hand. What is lacking today in cycle-by-cycle simulation is temperature dependence of the parameters and linkages to the power dissipated by the devices that causes temperature rise. However, the biggest and most meaningful foreseen step will be the development of tools that will translate PCB layout to electrical network parameters including radiated EMI. Some aspects of this are already available in some simulators (HSPICE, MetaSoftware USA). This will take out eventually most of the 'art' and intuition out of layout design and verification.

8. CONCLUSIONS

Even though complete simulation of power conversion systems is still a fantasy (as of early 1999) , the tools available today are already useful to the researches and designer. Global aspects including feedback design, dynamic stability and system interaction [23, 24], can be easily and accurately studied by simulation saving thereby costly experimental time and shortening time to market. Furthermore, the development of SPICE compatibele models of variuos non-linear loads [34, 35] can facilitate the simulation of a complete system.

Considering the imperfection of device models, inability to model exactly parasitic effects and the heavy numerical computation involved, cycle-by-cycle simulation should be used sparingly. Nonetheless, this simulation approach could be used in lieu of cumbersome mathematical derivations.

Like the case in any other new method, educational investment is needed to familiarize workers in the field with the power of simulation. On the other hand, simulation can help the engineering education process by making the courses more appealing and hence closer to the physical world. Further, simulation can be used to solve complex control issues easily and without much 'sweat'.

It would appear that simulation of power electronics systems is here to stay. Those who dodge or delay its use in R&D miss not only the benefit of a powerful and useful tool but also, as people who use simulation will attest, miss the fun.

REFERENCES

1. L.W. Nagel, *SPICE 2: A computer program to simulate semiconductor circuits*, Memorandum No. ERL-M520, University of California, Berkeley, 1975.
2. L. Meares and C. Hymowitz, *Simulating with SPICE*, Intusoft, CA., 1988.
3. P. Tuinenga, *SPICE: A Guide to Circuit Simulation and Analysis Using PSPICE*, Prentice-Hall, NJ. 1988.
4. S. Ben-Yaakov, and D. Adar (Edry), "Average models as tools for studying the dynamics of switch mode DC-DC converters."*IEEE Power Electronics Specialists Conference, PESC 94*, 1218-1225, Taipei, 1994.
5. S. M. Sandler, *SMPS simulation with SPICE 3*, McGraw Hill, 1997.
6. M. Gulko, and S. Ben-Yaakov, "Current-sourcing parallel-resonance inverter (CS-PPRI): Theory and application as a discharge lamp driver." *IEEE Trans. Industrial Electronics,* 451, 285 - 291, 1994.
7. M. Gulko, and S. Ben-Yaakov, "A MHz electronic ballast for automotive-type HID lamps." *IEEE Power Electronics Specialists Conference, PESC-97*, 39-45, St. Louis, 1997.
8. R. Middlebrook and S. Cuk, "A General Unified Approach to Modeling Switching Converter Power Stages," IEEE PESC, 18-34, 1976 .
9. S. Ben-Yaakov,"SPICE simulation of PWM DC-DC convertor systems: voltage feedback, continuous inductor conduction mode," *IEE Electronics Letters,* Vol. 25, No. 16, 1061-1063, August 1989.
10. Y. Amran, F. Huliehel and S. Ben-Yaakov," A unified SPICE compatible average model of PWM converters, "*IEEE Trans. on Power Electronics*," vol. 6, 585-594, Oct. 1991.
11. D. Edry, O. Mor, M. Hadar and S. Ben-Yaakov, "A Spice compatible model of Tapped-Inductor PWM converters," *IEEE APEC Conference*, APEC-Orlando, Florida, Feb. 13-17, pp. 1021-1027, 1994.
12. D. Kimhi and S. Ben-Yaakov,"A SPICE model for current mode PWM converters operating under continuous inductor current conditions,"*IEEE Trans. on Power Electronics*, vol. 6, 281-286, Apr. 1991.
13. S. Ben-Yaakov and Z. Gaaton, "A unified model of current feedback in switch mode converters, "*International Symposium on Circuits and Systems (ISCAS 92)*, 1992, Vol. 4, pp. 1891-1894, San Diego,.
14. S. Ben-Yaakov, D. Edry, Y. Amran and O. Shimony," SPICE simulation of quasi-resonant zero-current-switching DC-DC convertors," *IEE*

Electronics Letters, Vol. 26, No. 13, 847-849, June 1990.
15. S. Ben-Yaakov, "A unified SPICE compatible model of PWM and Quasi Resonant Converters," *International Symposium on Circuits and Systems (ISCAS 91)*, 1991, 1069-1072, Singapore.
16. D. Edry and S. Ben-Yaakov, "A SPICE compatible model of Magamp post regulator," *IEEE APEC rec.* , 793-800, 1992.
17. S. Ben-Yaakov, "Average simulation of PWM converters by direct implementation of behavioral relationships," *IEEE APEC rec.*, 510-516, 1993.
18. R. D. Middlebrook, "Modeling current programmed Buck and Boost converters," *IEEE Trans. on Power Electronics*, Vol. 4, 36-52, January 1989.
19. F. Dong Tan and R. D. Middlebrook, "Unified modeling and measurement of current-programmed converters," *IEEE PESC rec.*, 380-387, 1993.
20. R. B. Ridley, "A new continuous-time model for current-mode control," *PCIM rec.* 16-20, 1989.
21. G. C. Verghese, C. A. Bruzos and K. N. Mahabir, "Averaged and sampled-data models for current mode control: A reexamination," *IEEE PESC rec.*, 484-491, 1989.
22. S. Ben-Yaakov, "Average simulation of PWM converters by direct implementation of behavioral relationships,"*International J. of Electronics*, 77, 731-746, 1994.
23. H. Tsafrin and S. Ben-Yaakov, "The dynamic response of PWM DC-DC converters with input filters." *IEEE Applied Power Electronics Conference, APEC-92*, 764-771, Boston, 1992.
24. I. Zafrany and S. Ben-Yaakov, Average modeling and simulation of current shared DC-DC converters, *IEEE Power Electronics Specialists Conference, PESC-98*, 640-646, 1998.
25. D. Adar, G. Rahav, and S. Ben-Yaakov, "Behavioral average model of SEPIC converters with coupled inductors," *IEE Electronics Letters*, Vol. 32, No. 17, 1525-1526, 1996.
26. S. Ben-Yaakov, D. Adar, and G. Rahav, "A SPICE compatible model of SEPIC converters." *IEEE Power Electronics Specialists Conference, PESC-96*, 1668-1674, Boveno 1996.
27. H. Levy, I. Zafrany, G. Ivensky and S. Ben-Yaakov, "Analysis and evaluation of a lossless turn on snubber." *IEEE Applied Power Electronics conference (APEC'97)*, 634-640, 1997.
28. S. Ben-Yaakov, G. Ivensky. O. Levitin and A. Treiner, "Optimization of the Auxiliary Switch Components in a Flying Capacitor ZVS PWM Converters." *International J. of Electronics*, 81, 699-712, 1996.
29. G. Ivensky, D. Sidi and S. Ben-Yaakov, "A soft switcher optimized for IGBTs in PWM topologies." *International J. of Electronics*, 83, 703-716, 1997.
30. C. Basso, "A SPICE model for high-voltage integrated switcher", Motorola Application Note # AN1683/D, 1999.
31. R. L. Steigerwald, "A Comparison of Half-Bridge Resonant Converter Topologies," *IEEE Transaction on Power Electronics*, Vol. 3, No. 2, 174-82, April 1988.
32. S. Ben-Yaakov and G. Rahav, "Average modeling and simulation of series -parallel resonant converter by PSPICE compatible behavioural dependent sources," *IEE Electronics Letters*, Vol. 32, No. 4, 288-290, 1996.
33. S. Ben-Yaakov and G. Rahav, "Average modeling and simulation of series-parallel resonant converter by SPICE compatible behavioral dependent sources." *IEEE Applied Power Electronics conference (APEC'96)*, 116-120, 1996.
34. S. Ben-Yaakov, "Modeling the High Frequency Behavior of a Fluorescent Lamp: A Comment on "A PSPICE Circuit Model for Low Pressure Gaseous Discharge Lamps Operating at High Frequency" by T.-F. Wu, J.-C. Hung and T.-H. Yu [1]," *IEEE Transactions on Industrial Electronics*, 45, No. 6, 947-949, 1998.
35. S. Ben-Yaakov, M. Shvartsas and S. Glozman, "Statics and Dynamics of Fluorescent Lamps Operating at High Frequency: Modeling and Simulation," *Applied Power Electronics Conference, APEC-99*, (In print)

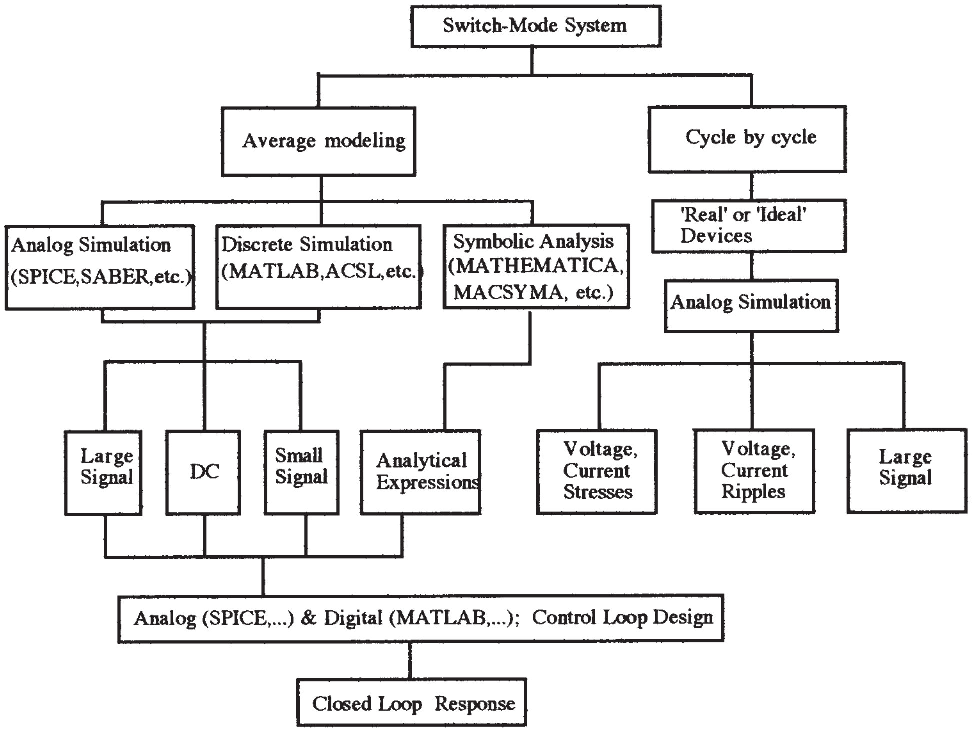
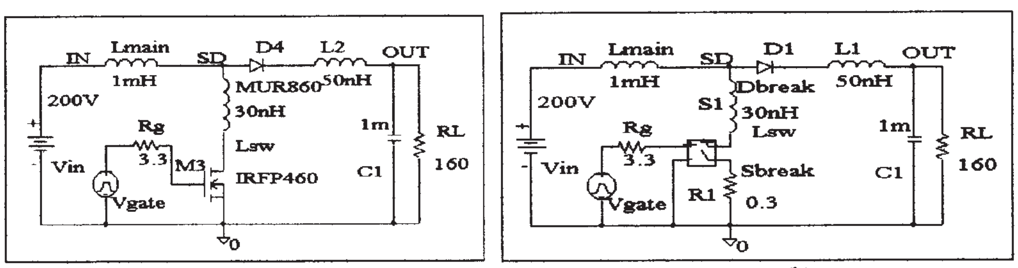


Fig. 1. Overview of simulation options of power conversion systems.



(a) (b)

Fig. 2. Simulation of a Boost Power stage: (a) applying 'real' devices; (b) replacing the transistor by a switch (Sbreak).

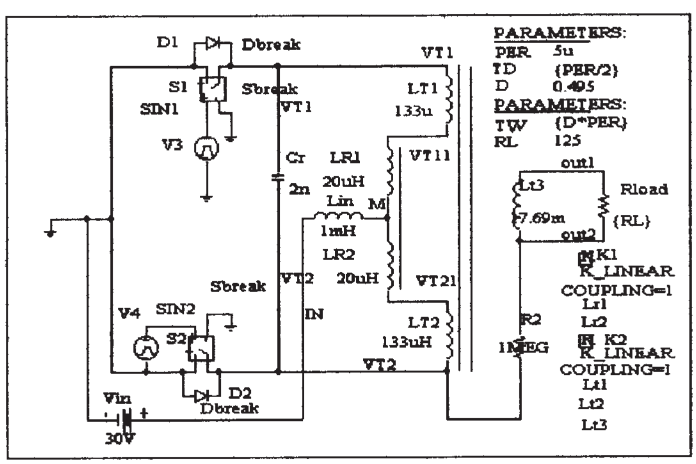
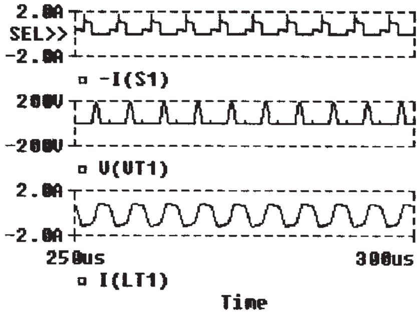
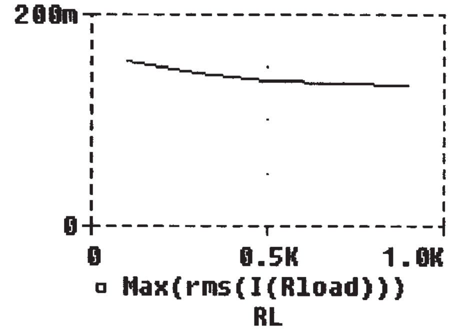


Fig. 3. Schematics of the CS-PPRI power stage.



(a)



(b)

Fig. 4. Waveforms of Fig. 3: (a) switch current (I(S1), switch voltage ((V(T1)) and transformer current (I(LT1)) ; (b) load rms current as a function of load resistance (I(Rload)).

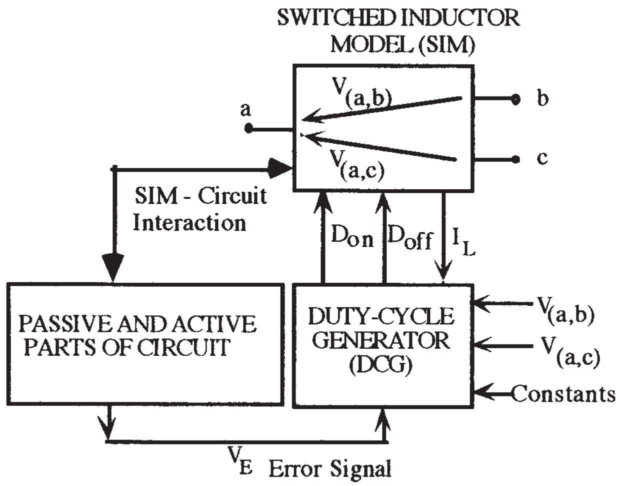


Fig. 5. Behavioral Average-Model representation of a power conversion system.

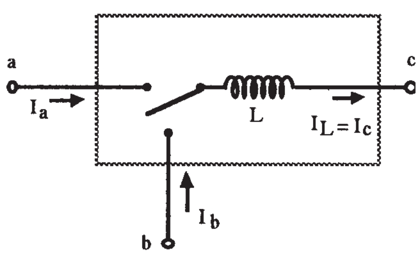


Fig. 6. The Switched Inductor (SI).

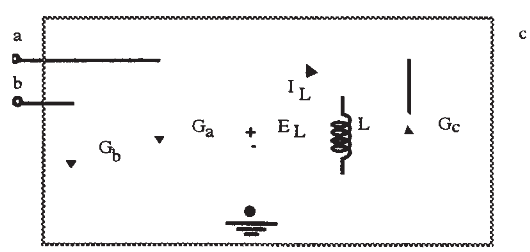
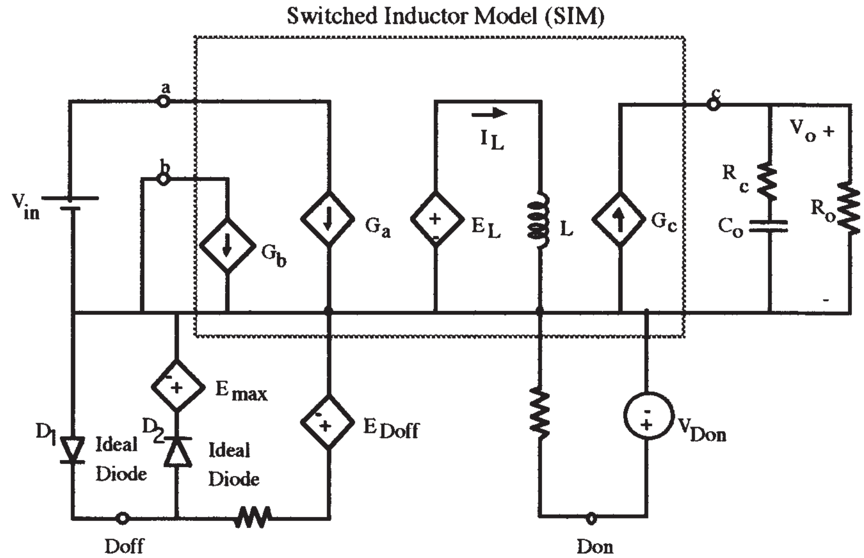
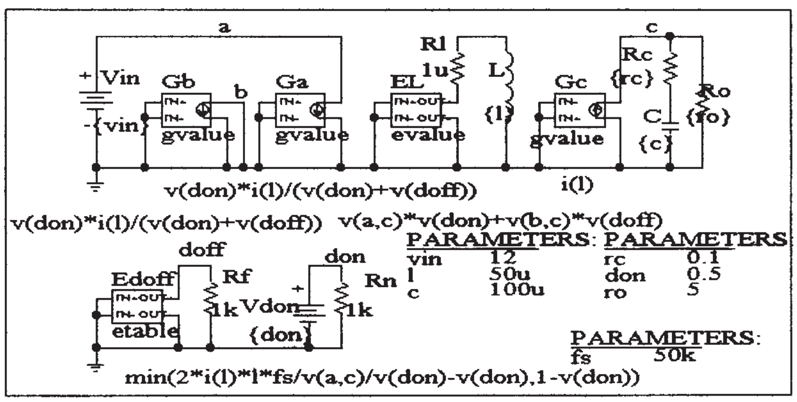


Fig. 7. The Switched Inductor Model (SIM).



(a)



(b)

Fig. 8. Behavioral representation of a buck converter (DCM &CCM). (a) generic representation; (b) SCHEMATICS input (PSPICE Versions 7 & 8).

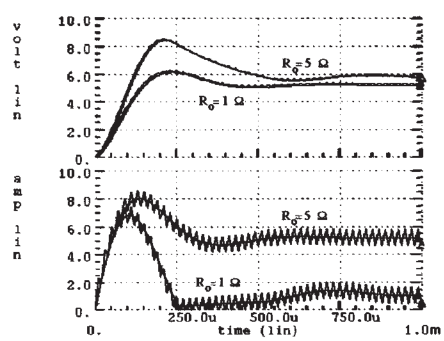


Fig. 9. SPICE startup response of opened loop buck converter obtained by average model simulation (smoothed line) and cycle-by-cycle simulation (rippled line) for two values of output resistor. See [4] for further details.

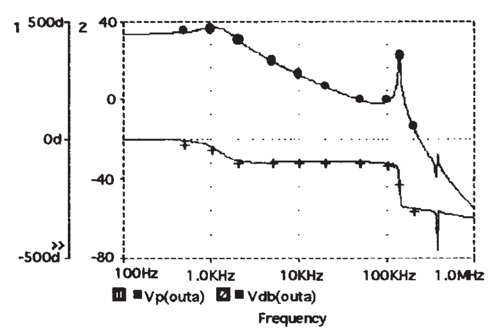
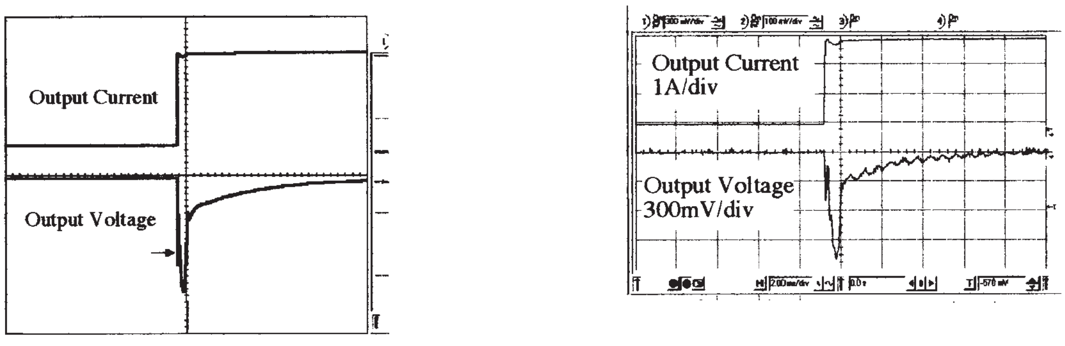


Fig. 10. AC small signal control-to-output $\left(\frac{Vo}{Vdon}(f)\right)$ response of SEPIC converter with coupled inductors. Cycle-by-cycle simulations (o for Magnitude[db], + for Phase[deg]) compared to results of average model simulation (continuous lines). See [25,26] for further details.



(a) (b)

Fig. 11. Output voltage as a function of step in load current in flyback converter: (a) average simulation; (b) experimental results. See [30] for details.

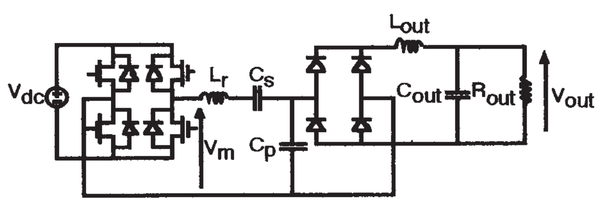


Fig. 12. Basis configuration of the series-parallel resonant converter topology.

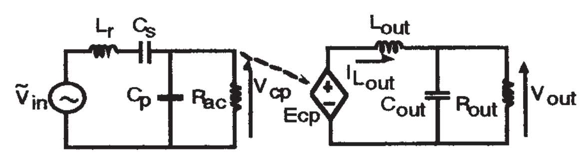


Fig. 13. First-harmonic approximation of the series-parallel resonant converter (Fig. 12). See [31-33] for further details.

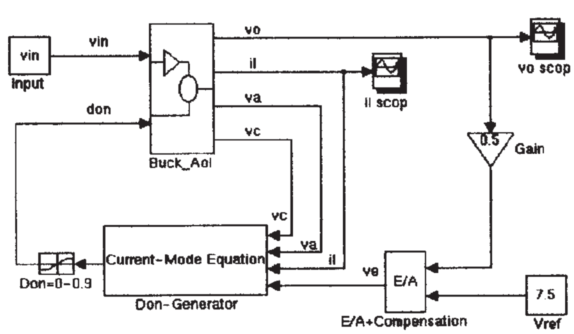


Fig. 14. SIMULINK compatible average model for Current-Mode Buck converter (Closed loop). For details see [4].

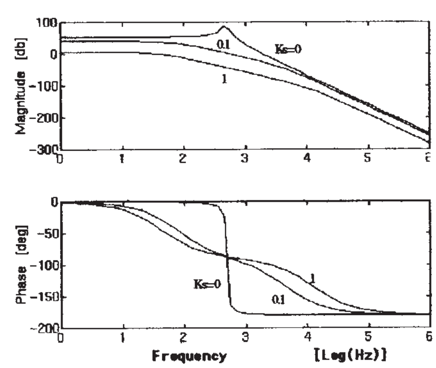


Fig. 15. Magnitude (upper trace) and phase (lower trace) of the small signal V_e/V_o transfer function of the current mode converter shown in Fig. 14 for various values of current gain (K_s).