Trends in Modeling and Simulation for Power Electronics Convertors

Ekaterina N. Dimitrova[1], Vencislav C. Valchev, [2]
Nikolay R. Nikolov[2], Dimitar M. Kovachev[2]

Abstract – **The simulation of Power Electronics Systems (PES) enables to predict its behavior before any hardware to be built. It allows checking various options, unusual operating or failure conditions and protection operating. These checks are difficult to realize in practice. The simulations allow also increasing the quality of students teaching. The current and future trends in the simulation of PES are discussed here. A comparison of the three most useful simulators for PES – MATLAB/SIMULINK, SABER and PSPICE is given. Power semiconductor models and magnetic models for PSPICE and SABER are compared and evaluated.**

Keywords: **PSPICE, SABER, MATLAB/SIMULINK, Power semiconductor models, magnetic models**

I. Introduction

The general trend in the simulations in power electronics is the remarkable increase in its application. The State-of-the-art power electronics laboratory set up along with supporting simulation laboratory is an important part of modern education process. Some curses in power electronics education are simulation supported [2].

Usually, general purpose electronics software is used for power electronics, rather than software specifically designed that application. In this case, the proper choice of the relevant simulation tool is important in power electronics design.

In this article, current and future trends in PES simulation, simulation software tools and device modeling are described.

II. Current and future trends in the simulation of PES

Background to SPICE, SABER, and MATLAB

With one notable exception, almost all commercially analogue simulation tools for electronics are based on the SPICE [3] program. Its algorithms are robust, powerful and thus, SPICE became an industry standard tool and the most commonly used simulator for power converter circuits. The main disadvantage of SPICE is inflexibility of the embedded models, which are difficult to adapt to the particular tasks in power device modeling.

SABER [4] is a relatively new simulator in which the models are separated from the simulator. Thus, one can readily create a library of accurate models, specifically designed for power devices. For many engineers, SABER remains unfamiliar and expensive compared to the available versions of SPICE.

SIMULINK is a simulation tool based on the popular MATLAB package. The equations of the system are essentially modeled using a wide range of graphical building blocks including control system notation, s-plane, state-space representation, etc. Users can add their own C programs too.

Mixed-Mode Simulation

One of the most significant developments in simulation technology, which affects the world of PES, is the advent of True Concurrent Mixed Analogue and Digital Simulation (TCMADS). Many switch-mode power supplies are based on integrated circuits that contain digital logic as part of the switching control circuitry. As the "pure analogue simulation" time increases exponentially with the size of circuit model for such circuits, it is sensible to look for techniques that reduce the model size while keeping the accuracy of the model. Opposite to the commonly known approaches based on model simplifications, TCMADS runs an analogue and digital simulators as separate processes in a multi-tasking environment, passing information between them at run time.

State Averaging Techniques (SAT)

SAT is based on characterization of the switching and the control circuitries as behavioral models rather than as combination of discrete components. Such technique provides an accurate model of the behavior of the power supply in response to variations in load without having to simulate each switching cycle individually and thus, it eliminates the great deal of data due to very small time step of simulation, which may vary down to picoseconds. This method can reduce the simulation time hours down to a matter of seconds.

Device models

In addition to the techniques described above, the modeling of the circuit at the component level is critical for the success of the simulation. One of the greatest problems here is the availability of device models. While the manufacturer's data provides a "starting point" for modeling the devices, it falls short in at least two aspects.

Firstly, the parameters provided by the manufacturer do not map easily into the internal SPICE parameters. A lot of models for typical small signal devices are available directly from manufacturers. However, for power devices and less commonly used components, these models are sadly lacking.

Secondly, the manufacturers tend to provide the kind of parameters that designers use in an average application. Their models often fall short, when an accurate simulation is required, especially in the cases when the devices work on the edge of its specification. Furthermore, often the only

[1] Dept. of Electronics, Technical University of Varna 9010, Bulgaria, E-mail: katy@ieee.bg
[2] Dept. of Electronics, TU of Varna 9010, Bulgaria

parameters that are provided are nominal values with no tolerance specifications.

The solution of the described problems lies in the increased use of simulation by Original Equipment Manufacturers (OEMs). This “positive feedback” results in providing the more necessary data in relevant formats. Nevertheless, if the information required is not available from the manufacturer, then the only solution is to measure the device parameters.

High –Voltage, High Current, Layout and EMC

A number of specific problems occur in high power applications modeling, such as welding. They are caused by the extreme values of process parameters (100s of volts, 100s of amps, switching times of 50-100ns, etc) combined with the large number of very small parasitic capacitance effects and transmission line effects. These make the modeling of the equivalent electrical circuit very dependent on the mechanical dimensions of components, mountings and interconnect. Shortcomings in device models also cause problems in this area. At this sort of level, the only real solution is to provide a simulator that solves electromagnetic field effects and converts them into an equivalent circuit. With the advent of EEC wide regulations on electromagnetic emissions we can expect to see much work being done in this area to solve a pressing problem.

III. A COMPARISON OF THREE POPULAR PACKAGES

Although there are areas of overlap between the simulations, each has some particular qualities. Sometimes the tools are used in a complementary manner. For example, MATLAB can be used to calculate parameters for a SABER model or even, vice versa, SABER can be used to derive parameters for a SIMULINK model.

MATLAB/SIMULINK

The simulation engine is based on MATLAB’ s powerful matrix processing core and several fixed and variable time step solving algorithms are available. The full power of MATLAB is available for graphing and post processing. SIMULINK is mainly useful for application level modeling since it contains a wide variety of control system models.

SABER

SABER, from Analogy Inc., was developed as an engineering simulation tool. The library of models covers several technologies, not just electrical/electronic. All the models can be mixed in the same simulation. Users can create their own models (if the required function is not in the library) using Analogy’s proprietary Hardware Definition Language, MAST. SABER is also compatible with SPICE models.

The simulation engine has separate simulators for the analogue and digital domains and a patented algorithm ensures synchronization of two simulations when required. This is especially important for modeling wide range of switching circuits. The analogue simulator engine has a variable time step algorithm with various simulation controls available to the user. Graphical display of the simulation is good, with an intuitive user interface. A variety of post processing tools are available including waveform measurements and a waveform calculator. Some work has been done using MASAT/SABER to develop a range of physics based power device models [1].

PSPICE

PSPICE is one of the most popular of the commercial SPICE-based simulation packages. Models are created by schematic capture or by text editing of a netlist. Large numbers of models are available since SPICE models are interchangeable between all SPICE based simulation tools. Unlike SABER, PSPICE is only aimed at electronic engineering applications, although it is possible to develop analogues models using electrical elements (R, C, and L) for non-electrical problems e.g. Heat dissipation. Graphical display of the simulation output is good, with a reasonable user interface. A variety of post processing tools are available including waveform measurements and FFT.

Table I compares basic features of the packages.

IV. POWER SEMICONDUCTOR MODELS

The specially developed power semiconductor device models for SPICE are compared and evaluated in this section.Table II presents the generic and special SPICE models for power devices. Note that no models exist for several of the devices, and that most of the models have major limitations. The comments about DIODE and BJT are illustrated on Fig.1 and Fig.2 respectively.

Table III compares three simulations: GENERIC SPICE, a typical ENHANCED SPICE, and SABER for their relative capability in modeling phenomena relevant to power semiconductor devices

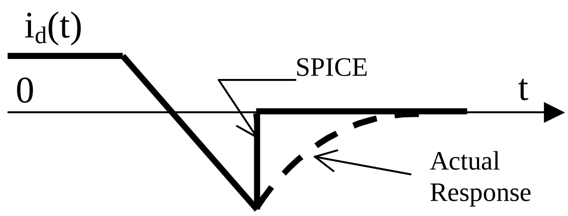

Fig.1. Diode Turn-OFF Waveform

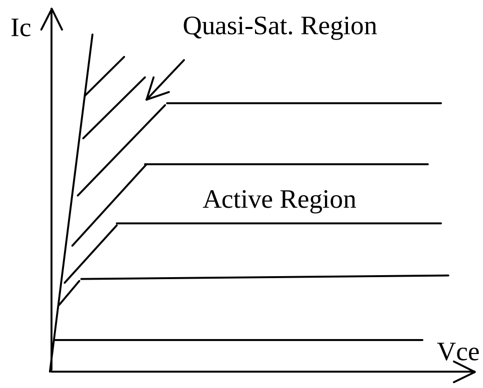

Fig.2. BJT Collector Characteristics Showing Quasi-Saturation Region

Table I. Comparisons between MATLAB, SABER and PSPICE

PROPERTY	MATLAB/SIMULINK	SABER	PSPICE
VERSION	5.2/2.2	4.3	9.2
Supplier	The Mathworks Inc	Analogy Inc.	Orcad
Platforms	PC (Windows 98/NT or Linux), Unix Systems, Apple	PC (Windows 98/NT), HP and Solaris Unix systems	PC (Windows 98/NT)
Circuit Entry	Schematic or text	Schematics or text	Schematics or text
Simulation Engine	Based on Matlab' s matrix solving algorithms; Various solving algorithms available (both fixed and variable step)	Variable time step. Separate analogue and digital simulation engines with synchronized interface.	Spice 2
Available Simulations	DC, time domain	DC, time domain, frequency domain, parameter sweep (nested), Monte Carlo	DC, time domain, frequency domain, parameter sweep, Monte Carlo
Display of Results	Uses Matlab's Graphing Functions which include 3D. Wide range of functions available but not user friendly.	Only 2D graphs available. User interface very good. Frequency domain results can be shown as Bode or Nyquist plots	Only 2D graphs available. User interface fair.
Availability of Models	Increasing. Several "block sets" representing components are now available, including the Power System Block set aimed for modeling of power generation and distribution. Users can interface own C or FORTRAN routines.	The generic template library gives a good coverage. Good coverage of electronic components used in the automotive industry. The user group has a library of templates donated by Saber users. Users can interface their own C or FORTRANE routines	Very large availability of models due to the wide Spice user base.
Post-Processing	All Matlab functions available since results stored as matrices: FFT, Filtering, waveform extraction and calculations. Data can be saved in ASCII format.	Waveform calculator allows manipulation of data: arithmetic, parameter extraction, FFT. Waveform measurements available. More functions available in AIM macro language including matrix arithmetic. Data can be saved in ASCII format.	Calculations can be performed on waveforms e.g. calculate power from V and I. FFT available. Data can be saved in ASCII format
Documentation of results	Graphs can be annotated with user information. Graphs can be saved for later recall. Wide variety of graphical formats and printer support available (PS, BMP, TIFF and JPEG)	Graphs can be annotated with user information and measurements. Graphs can be saved for later recall. Limited number of graphical formats and printer support: basically Postscript, PCL and MIF.	Graphs can be annotated with user information. Graphs can be copied to the Windows clipboard and then passed into word processor, spreadsheet etc. Graphs cannot be saved as files.
Batch Processing	Simulink can be controlled from a Matlab M-script.	The simulation engine can be rum in true batch mode on Unix systems. Simulator can be controlled using a command line script (which is separate from the netlist) or an Aim script.	All parts of the tool (schematic, simulator and results analysis) can be run separately. The simulator is controlled from commands embedded in the netlist. These can be inserted via the schematic GUI or by editing the netlist.
Add-ons	Additional Matlab toolboxes are available covering wide range of applications. DSpace real-time simulator	Sensitivity and stress analysis. Linear system (pole/zero) analysis. Magnetic system design tool. Links to other simulation tools.	PCB layout tool
Customization	Matlab contains a range of GUI. Creation commands which allow users to create their own applications/ interfaces	The AIM macro language is a superset of TCL/Tk which includes a full range of GUI creation commands. AIM is a powerful tool which includes comm.-ands for waveform manipulation, graph creation and controlling the simulator. However, it is not well documented.	Users can create their own measurement functions.

TABLE II. POWER SEMICONDUCTOR DEVICE MODELS FOR SPICE

COMPONENT	GENERIC MODEL	POWER MODEL	COMMENTS
DIODE	Charge Control	Yeung &Shackleton; Xu & Schrőder	Lacks “soft” recovery and forward recovery. Requires complex subcircuit
BJT	Ebers-Moll and/or Gummel- Poon	Bowers et al.; Getreu; Yeung et al.	Lacks quasi-saturation and dynamic saturation
MOSFET	Frohman-Grove	Neinhaus,Bowers Lauritzen& Shi Wheatley, Ronan,Dolny Fay& Sutor Hancock Yee & lauritzen Simas, Piedade,Freire Xu & Schrőder Cordonnier	Basic Model, Const. Cgd. Unusable for BVdss>80V. Inaccurate on Turn-OFF. Const. Cgd, internal info req. Cgd:switched. Accurat e, but complex Cgd:2C values plus switch. Cgd:3C values plus switch. Insufficient information to evaluate. Cgd: 1C value plus diode C.
JFET or SIT	Shichman & Hodges	None	The power JFET is called the Statistic Induction Transistor SIT.
SCR, GTO	None	Xu & Ki Avant&Lee Wong &Lin McMurray	All models have same dynamic deficiencies as DIODE and BJT models. Includes cond. Modulation.
IGBT	None	McDonnald&Fossom	Uses SPLICE (special SPICE)
BJT Darlington	2 BJT models	None	Model for integrated Darlington is needed

TABLE III. SIMULATOR COMPARISON FOR DEVICE MODELING

MODELING CAPABILITY	TYPICAL SIMULATORS		
	GENERIC SPICE	ENHANCED SPICE	SABER
1. Provides Two Levels of Accuracy			
Basic	YES	YES	YES
High Accuracy	?	YES	YES
2. High Voltage Device Phenomena			
Diode Recovery and Turn-ON	?	YES	YES
Quasi –Sat., Dynamic Sat. in BJTs	NO	?	YES
Wide Cgd Capacitance Swing in MOSFETs	?	YES	YES
Tail Currents in GTOs, IGBTs	?	YES	YES
3. Temperature Effects			
T as Independent Device Parameter	NO	NO	YES
T as function of Device Dissipation	NO	?	YES
T as Func. Of Heat Sink Dynamic Thermal Impedance	NO	?	YES
4. Power Ics and Control ICs	Subcircuit only	Behav. model	Behav. model

Of the inductor and transformer models developed for SPICE and SABER, only the Pei & Lauritzen and Meares models listed in Table IV will work with ordinary versions of SPICE. They both simulate saturation well, but not the variation of hysteresis or losses with frequency. These models are based on empirical curve fitting. Teegardin has recently implemented a physical model, the Jiles-Atherton model on SABER. However, the Jiles-Atherton equations are probably too complex for simulation on SPICE

TABLE IV. MAGNETIC DEVICE MODELS FOR SPICE AND SABER

AUTHOR	FEATURES	COMMENTS
Pei & Lauritzen	Saturation and hysteresis simulated by non-linear controlled sources on SPICE	Good for constant freq. excitation
Mears	Saturation and hysteresis simulated by diodes and voltage sources on SPICE	Complex sub circuit;
Tabrizi[]	Closed model, Insufficient information to evaluate	Works on Analog Workbench SPICE only

Models available in a public domain: http://www.orcadpcb.com/PSPICE/models.

V. CONCLUSIONS

Although the analog circuit simulators currently available such as SPICE or SABER contain elementary semiconductor and magnetic device models, these models fail to simulate important power (high voltage) device phenomena. Thus, many power device types such as thyristors or IGBTs are omitted from these model libraries. New device types and combination devices such as Darlingtons also need to be included.

The new simulator SABER offers a number of special features for modeling of power devices from their fundamental equations. However, some enhanced versions of SPICE also allow direct insertion of the special device equations for modeling.

However, building good models is not simple. The efforts to create models for power devices have just begun.

In the paper we give the current and future trends in the development and improvements of some most widely spread simulators. A comparison of the three most useful simulators for PES – MATLAB/SIMULINK, SABER and PSPICE is given. The specific features and advantages of each of the discussed simulators are given. Thus, the designer can benefit reading the results of that comparison and to make the right choice of the most appropriate simulator.

REFERENCES

[1] Lauritzen P. O, Subramamanin Y, Bi Y, Green L. “An Efficient Way to Implement Electrical and Thermal Device Models in MAST”, North American ASSURE Meeting, March 1997

[2] Batarseh, I.; Kemnitz, D.A**,** “ Undergraduate education in power electronics”-Southcon/94. Conference Record , 29-31 March 1994

[3] Paul W. Tuinenga- “SPICE- AGuide to Circuit Simulation & Analysis using PSPICE.

[4] SABER, Analogy,Inc., P.O. Box 1669, Beaverton OR 97075