Invited paper

New approach to power semiconductor devices modeling

Andrzej Napieralski and Małgorzata Napieralska

Abstract — The main problems occurring during high power device modeling are discussed in this paper. Unipolar and bipolar device properties are compared and the problems concerning high time-constant values related to the diffusion phenomena in the large base are explained. Traditional and novel concepts of power device simulation are presented. In order to make accurate and modern semiconductor device models widely accessible, a website has been designed and made available to Internet users[1], allowing them to perform simulations of electronic circuits containing high power semiconductor devices. In this software, a new distributed model of power diode has been included. Together with the existing VDMOS macromodel library, the presented approach can facilitate the design process of power circuits. In the future, distributed models of IGBT, BJT and thyristor will be added.

Keywords — power device modeling, SPICE, circuit simulation, VDMOS, PIN diode, IGBT, web-based simulation.

1. Introduction

Many CAD programs have been developed in microelectronics and successfully applied to circuit analysis. The question arises, why there are no CAD programs for power-circuit analysis? In this paper, we will discuss the need for a development of a new kind of power semiconductor device models. Additionally, we will present a new concept of a free, Internet-based software development, dedicated to power semiconductor circuit analysis. Until now, high prices and hardware requirements limited the access to professional CAD tools for the majority of educational institutions, students and small enterprises. Free versions of commercial software have limited functionality and do not contain the modern numerical algorithms and the libraries of device models.

This is especially true in the case of power electronics, as it requires advanced device models to obtain simulation results of good accuracy. Such models are not available now, even in the commercial versions of the CAD software.

The increasing popularity of the Internet can help to solve this problem. Wilamowski, Malinowski and Regnier [33] were probably the first to see the possibility of performing circuit simulations over the Internet by means of dedicated Internet applications. Simulation software may run on remote servers and results may be sent to the user in the form of numerical data or graphics. Wilamowski *et al.* [34] have put special emphasis on the pay-per-use access to simulation software and platform-independent user interface provided by a web page. In this paper, a development of a free simulation environment is discussed that includes new numerical algorithms and modern device models. The Internet is not only considered as a data transmission system but first of all as a medium for international cooperation of scientific centres, enterprises and individual users that would enable the development of better and freely accessible simulation tools and models [27–30, 34].

[1] http://www.dmcs.p.lodz.pl/dmcs-spice

2. Power device modeling

The main parameters of power semiconductor devices are:

- voltage-blocking capability,
- current capability,
- switching performance,
- safe-operation limits.

The basic parameters of modern power devices are presented in Table 1. As can be seen from this table, the switching performance of a VDMOS (a unipolar device) is very good, but its current capability and maximum blocking voltage are much smaller than in the case of bipolar devices (GTO [14, 31] and GCT). The IGBT transistor is a kind of fusion of the best properties of unipolar and bipolar devices. Voltage and current rating is much better than in the case of VDMOS and switching properties are much better than in the case of GTO and GCT.

Table 1
The basic parameters of the most commonly applied modern power devices

Parameters	VDMOS	IGBT	GTO	GCT
V_M [V]	1000	4500	6000	6000
I_T [A]	350	900	6000	6000
t_{ON} [μs]	0.1	0.2	10	–
Storage time [μs]	0	6	30	5
f_T [kHz]	2000	100	1	1
V_{ON} [V]	Very high	Low	Very low	Very low

Figure 1 presents the product of device V-I ratings for power semiconductor devices. The solid line represents the present state and the dotted line the future trends in power semiconductor device development.

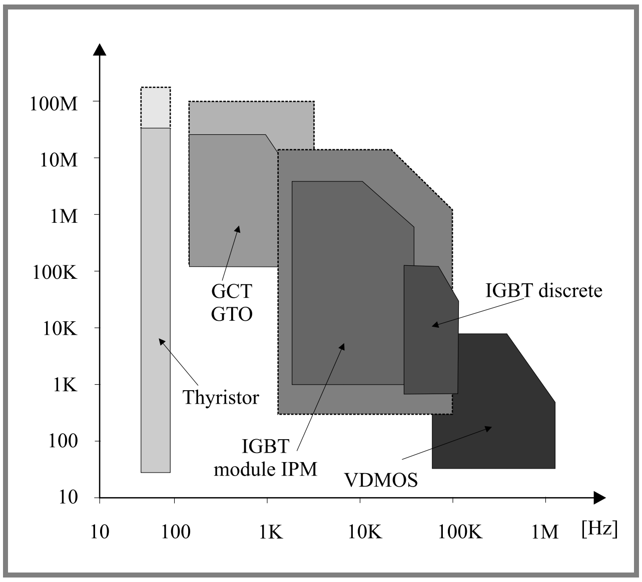

Fig. 1. Product of device V-I ratings [VA] as a function of operating frequency.

In all types of power devices, the blocking capability is a function of the large base width and its doping concentration. Expression (1) gives the value of the maximum blocking voltage (breakdown voltage) as a function of the large base doping concentration N_D:

$$V_B = 60\left(\frac{10^{16}}{N_D}\right)^{\frac{3}{4}}, \tag{1}$$

where: N_D – doping concentration [cm^{-3}],
V_B – blocking voltage [V].

Expression (2) gives the value of the large-base width as a function of blocking voltage V_B and doping concentration N_D without taking the punch trough effect into account:

$$W = \sqrt{\frac{2\varepsilon(\Phi + |V|)}{qN_D}} \cong \sqrt{\frac{2\varepsilon|V|}{qN_D}}, \tag{2}$$

where: W – large base width.

The conclusion from these two expressions is that for a high value of blocking voltage, the semiconductor devices must have a low doping and a large width of the base.

The necessary base width values have been calculated from expressions (1) and (2) for three different values of large base doping with the corresponding maximum blocking voltage V_B. The results are presented in Table 2.

The most important conclusion from this table is that for the high power semiconductor devices the voltage must be supported in a large base. Therefore, the charge transport through the base cannot be instantaneous and the voltage drop over such a structure cannot be neglected. Let us consider now two types of power devices – a unipolar one and a bipolar one. In the case of the unipolar device, the current capability, or in other words, the value of R_{ON} resistance, can be found from a very simple consideration depicted in Fig. 2.

Table 2
The maximum blocking voltage
and necessary base width for three different values
of large-base doping concentration

Device	N_D	V_B [V]	W [μm]
1	$1.25\cdot10^{14}$ cm^{-3}	1600	125
2	$1\cdot10^{14}$ cm^{-3}	1895	157
3	$1\cdot10^{13}$ cm^{-3}	10700	1175

According to Table 2, for a given value of the large base doping concentration, the corresponding maximum blocking voltage V_B and the necessary base width can be calculated. In the case presented in Fig. 2, the value of N_D is equal to $1.25\cdot10^{14}$ cm^{-3}. The corresponding maximum blocking voltage is 1600 V and the base width is equal to 125 μm. Current density in the case of the unipolar device can be expressed by:

$$J_n = q\mu_n nE = q\mu_n n\frac{V}{W}, \tag{3}$$

where: E is the electric field, n is the carrier concentration (equal to N_D in this case), μ_n is the electron mobility, q is the elementary charge.

Taking into account Fig. 2, the value of R_{ON} for the unipolar device having the surface equal to 1 cm^2 can be found from the following expression:

$$R_{\mathrm{cm}^2} = \frac{V}{J_n\mathrm{cm}^2} = \frac{W}{q\mu_n N_D\mathrm{cm}^2}, \tag{4}$$

In the case of the device from Fig. 2, this value will be equal to 0,462 Ω. In the case of high current this value of on-state resistance is definitely too high. Even for a relatively low current of 10 A, the voltage drop will be 46 V. We can thus conclude that unipolar devices cannot be applied in the case of high voltages. This conclusion is in accordance with the real device parameters presented in Table 1.

In the case of a bipolar device, the situation is completely different. Two mechanisms are responsible for the current density and it can be expressed by a set of the following equations:

$$J_n = q\mu_n nE + qD_n\frac{\partial n}{\partial x}, \tag{5a}$$

$$J_p = q\mu_p pE - qD_p\frac{\partial p}{\partial x}, \tag{5b}$$

$$J = J_n + J_p + \varepsilon\frac{\partial E}{\partial t}, \tag{5c}$$

where: D_n and D_p are electron and hole diffusion constants, respectively, and ε is the permittivity.

As can be seen from Eqs. (5a–5c), the current capability in the case of bipolar devices is higher than in the previous case. Due to the diffusion, the maximum current density is much higher and consequently the corresponding voltage drop much lower than in the case of unipolar devices. Additionally, in the bipolar structure some effects resulting from the changes in the effective base doping can appear, for example: the base widening effect, and thyristor effect rendering in the additional current capability.

Now, we will analyze how the mechanism of charge transport through the large base influences the switching performance of the device.

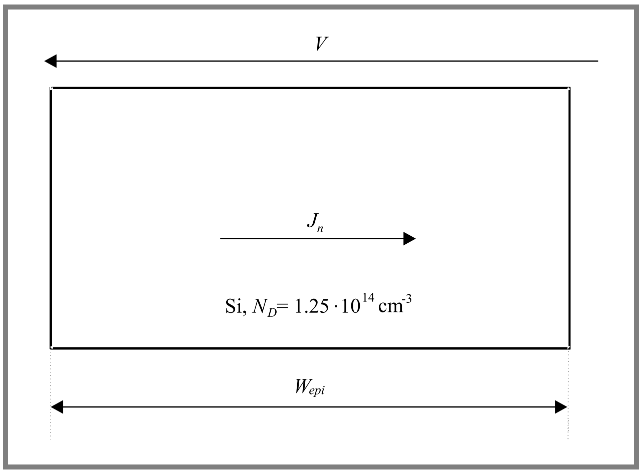

Fig. 2. Illustration of R_{ON} calculation in the case of unipolar devices.

According to Fig. 2 the charge transport time through the large base in the case of the unipolar device can be expressed by:

$$t = \frac{W_{epi}}{v_{nsat}}, \tag{6}$$

where: v_{nsat} is the electron saturation velocity:

$$v_{nsat} = 10^7 \, \frac{\text{cm}}{\text{s}} \left[\frac{T}{300\,\text{K}}\right]^{-0.87}. \tag{7}$$

In the case of power devices operating in conditions of high electric field, one can assume that all the charge carriers are able to attain their maximum speed. For the temperature equal to 300 K, the transit time through the large base can be easily calculated. In Table 3 transit times for three values of base width are presented.

As can be seen from Table 3, in the case of unipolar devices even for a very large base the transit time is very short and, in the majority of cases, the internal time constant of the device can be neglected, compared with the time constant of the external circuit.

The situation is different in the case of bipolar devices. As a first order simplification, we can assume that the transit time of minority carriers through a large base can be expressed by [25]:

$$\tau_D = \frac{W_{epi}^2}{2D} = \frac{W_{epi}^2}{4D_n}, \tag{8}$$

where: $D = 2D_n$ at a high injection level.

Table 3
Electron transit time through the large base of a unipolar device for three different base widths

Device	W [μm]	Transit time [ns]
1	125	1.25
2	157	1.57
3	1175	11.75

In Table 4 the transit time for three values of base width is presented.

Table 4
Electron transit time through the large base of a bipolar device for three different base widths

Device	W [μm]	Transit time [μs]
1	125	1.12
2	157	1.76
3	1175	98.62

In this case the transit time through the large base is very long and the internal time constant of bipolar devices can be much higher than those of the external circuit. In such a case, a simple lumped model cannot be used any longer. It is necessary to take into account the carrier transport inside the bipolar devices and the distributed model should be applied. Until now such a model did not exist in the circuit CAD simulation programs.

3. Contemporary methods of power device simulation

Because of the problems mentioned above, correct power device simulation is not possible in standard simulation programs. The first question is why power-device models are necessary? Their main areas of application are as follows:

- power integration – simulation of power functionality in integrated circuits [24],
- correct simulation of power dissipation inside power semiconductor devices (with special emphasis on the losses during the switching period),
- computation of the temperature distribution inside the semiconductor structure,

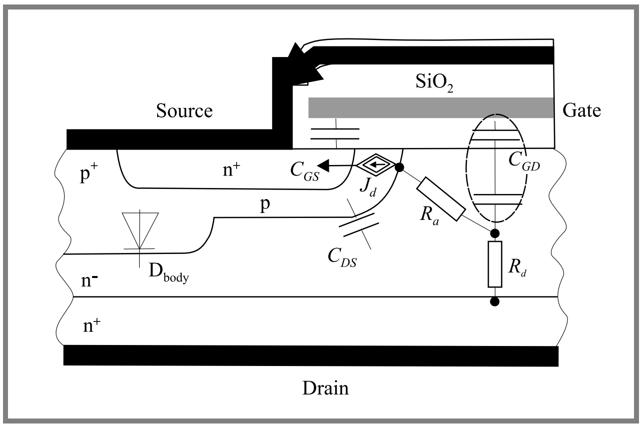

Fig. 3. The internal structure of VDMOS power transistor.

– correct design of the cooling environment,

– prediction of the second thermal breakdown.

In the case of unipolar devices, such as VDMOS (Fig. 3), minority carriers are responsible for current conduction in the normal mode of operation. Therefore, it is possible to build a relatively simple macromodel [17–23, 25] taking into account all the internal elements of the presented structure.

A macromodel of this type, developed for SPICE-like simulators, is presented in Fig. 4 [17–23]. In the case when the body diode D_{GD} is not conducting (VDMOS is not reverse biased), this model is quite accurate and can be applied for a simulation of all types of circuits. The situation is different when the body diode D_{GD} starts to conduct current and diffusion phenomena have to be considered.

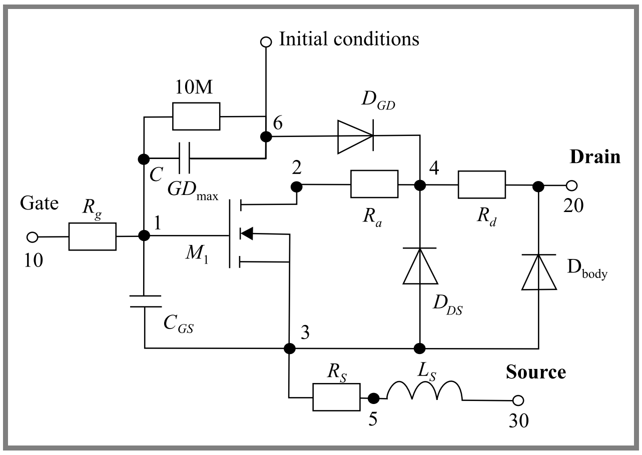

Fig. 4. VDMOS SPICE-like macromodel [2, 23].

In Fig. 5 a half-bridge circuit configuration is presented. The corresponding results of the simulation and measurement are shown in Fig. 6. The voltage and current waveforms are in good agreement with the experiment, and even the power dissipation inside the structure can be correctly predicted.

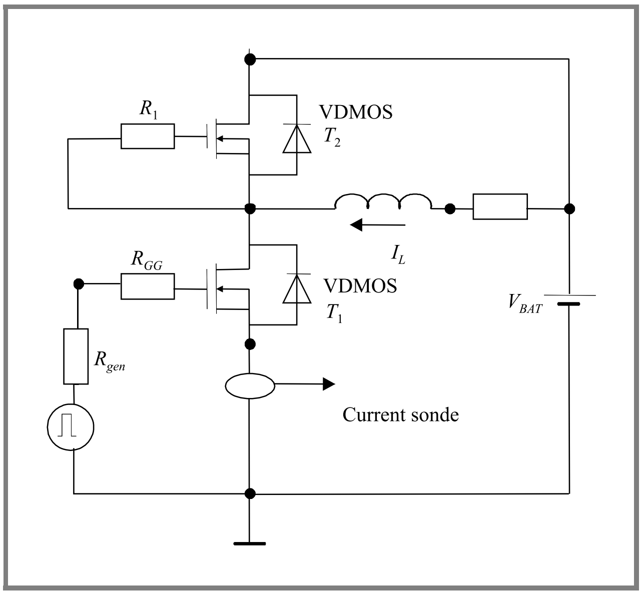

Fig. 5. Half-bridge circuit configuration [23].

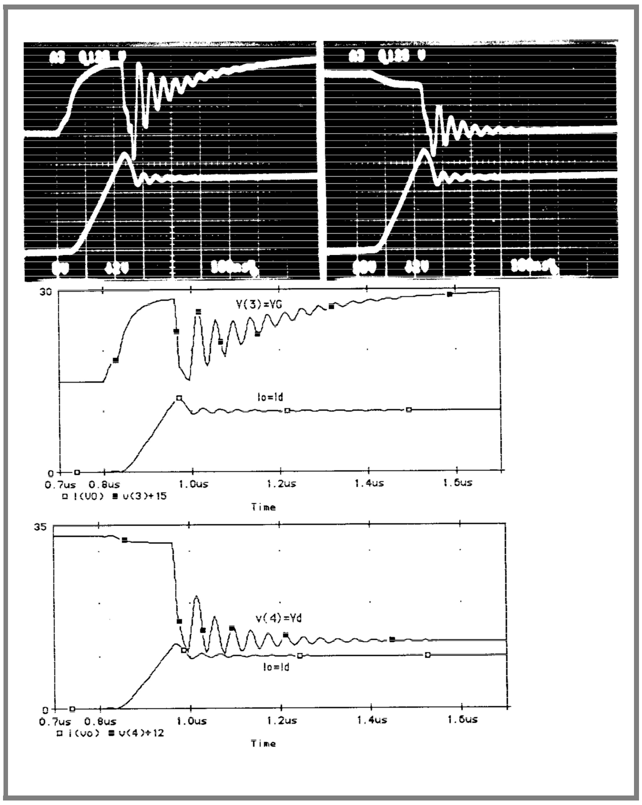

Fig. 6. The simulation and measurement results of a half-bridge with VDMOS transistors [23].

The above considerations lead us to the conclusion that in the case of unipolar devices we have:

– no problem with the diffusion phenomena (minority carriers only),

– very good accuracy obtained by means of lumped models and fast simulation,

– clear physical/geometrical interpretation of model parameters,

– distributed models are necessary only to model the operation of the body diode D_{GD}.

Such a simple approach cannot be applied in the case of a bipolar structure, such as PIN diode, thyristor or IGBT. In Fig. 7, the cross-section of an IGBT elementary cell is presented. All the important phenomena occur in the large base (in this case in the n^- region). In this punch-through IGBT transistor, the large base is relatively short, but even in such a case, one cannot neglect the minority carrier transport phenomena.

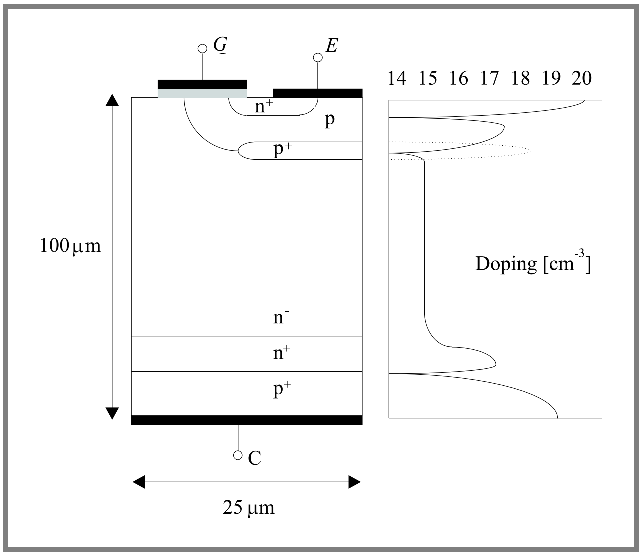

Fig. 7. The internal structure of IGBT power transistor (cross-section of an elementary cell) [5–8].

The most commonly adopted approach to solve this problem is the application of a finite element, finite difference or finite box method.

In Fig. 8 the discretisation of one IGBT cell for 2D simulation is presented. The application of such models enables not only correct current and voltage response simulation but even evaluation of the internal power losses, energy dissipation, temperature distribution (with an additional thermal submodel [6, 10–13]) and current density inside the structure.

As an example of this type of simulation, collector-emitter voltage and collector current as well as dissipated power and energy losses during IGBT switching are presented in Figs. 9 and 10.

The application of an additional thermal model enables the computation of the temperature rise inside the semiconductor structure. This additional thermal model has to be a three-dimensional one. In Fig. 11, the time waveforms of

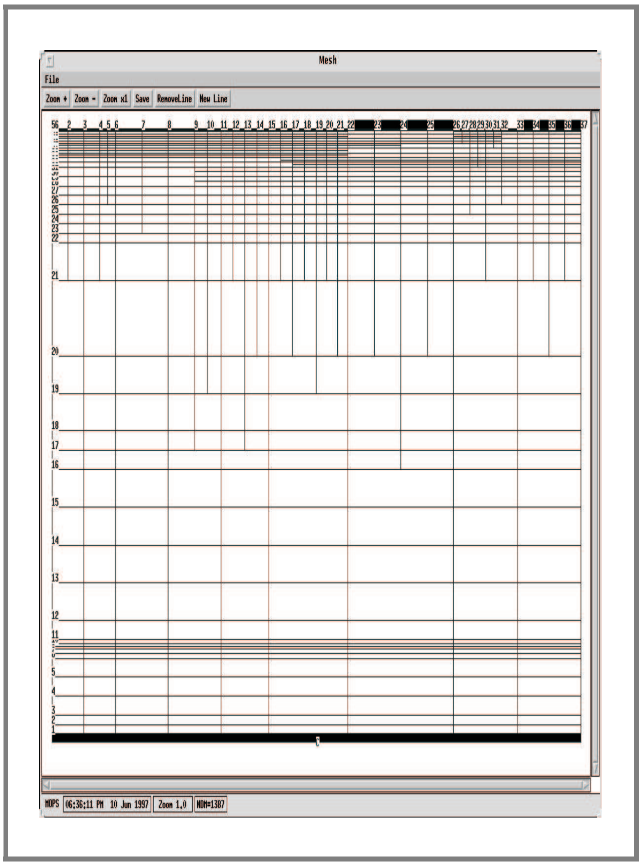

Fig. 8. Discretisation mesh used during 2D analysis of an IGBT structure [6].

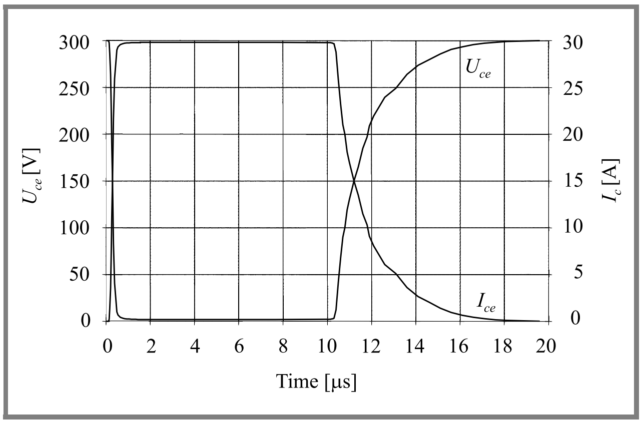

Fig. 9. V_{CE} and I_{CE} during IGBT switching [6].

total power dissipation and the maximum temperature rise inside the studied IGBT structure, are presented.

The main drawback of such an approach is the very long time of simulation and difficulty in simulating more than one or two power devices in the same circuit. The power circuits designers need relatively simple models which can be applied in standard SPICE-like circuit simulation programs with a very short time of simulation and the possibility of high power circuit simulation. In the next section a new approach to this problem will be presented.

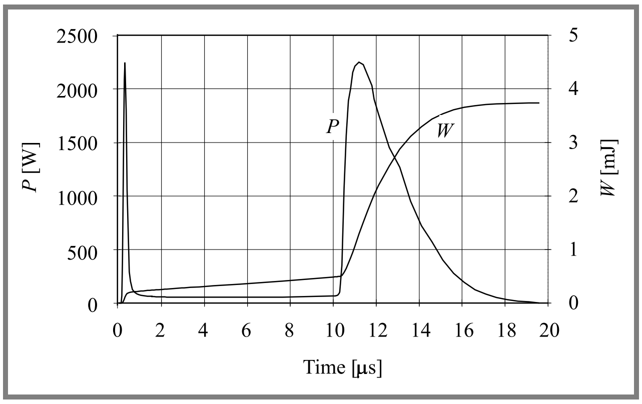

Fig. 10. Power and energy losses during IGBT switching [6].

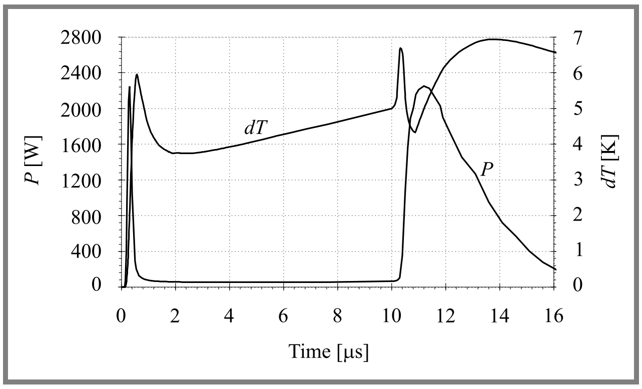

Fig. 11. Time waveforms of total power dissipation and maximum temperature rise inside the IGBT structure [6].

4. Developing the new type of bipolar power device model

Lumped semiconductor devices models are applied in all versions of SPICE programs. Such models can be used in the case of low-power devices, where the base is relatively short and device internal time constants are much lower than those of the external circuit. For the reasons that will be explained later, our model will be implemented in SPICE3F5 simulator.

In order to enable simulation of circuits containing power semiconductor devices, we will try to develop separate device models, taking into account the distributed nature of the phenomena occurring in bipolar power devices and determining their dynamic response. In this section, a model of PIN diode will be presented [27]. Its development is the key point for future design of this type of models for BJT, IGBT or thyristor.

In order to simplify the modeling procedure, we will apply the so-called "modular" approach. In the semiconductor structure considered here, we will distinguish several regions of different physical and/or electrical nature. Then, a simplified sub-model will be assigned to each of them. This approach allows for decreasing the simulation time considerably without any important loss of accuracy [4, 15].

As explained above, in the case of power semiconductor devices the most important is the large base region, which assures high voltage blocking capability during the off-state and where the excess carriers are stored during the on-state (Fig. 12). The simplest device in which one can consider all important physical phenomena is the PIN diode. The well known, one-dimensional Benda-Spenke model [1] has been adopted for this purpose. The behaviour of stored charge carriers can be described by means of the ambipolar diffusion equation:

$$\frac{\partial^2 p(x,t)}{\partial x^2} - \frac{1}{D}\left(\frac{p(x,t)}{\tau} + \frac{\partial p(x,t)}{\partial t}\right) = 0 \tag{9}$$

where: p is the carrier concentration, D is the ambipolar diffusion constant and τ is the common electron and hole lifetime.

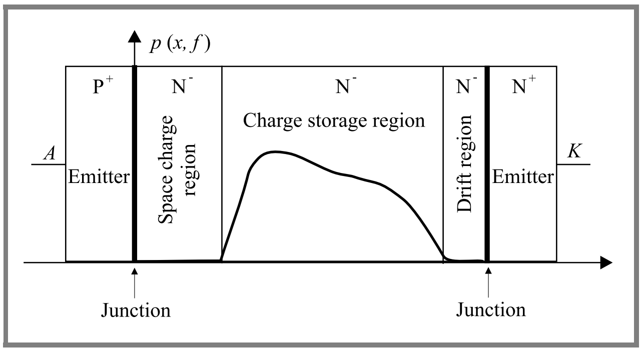

Fig. 12. Modular modeling concept and its application to PIN diode.

Equation (9) cannot be solved analytically, therefore many different approaches using numerical solutions have been presented in the literature. The proposed model is based on the algorithmic approach, i.e. the solution is obtained with a numerical algorithm. A new efficient centered weighted essentially non-oscillatory (CWENO) scheme [16] is used to ensure high computational stability and good accuracy. After the solution of Eq. (9) is obtained, the negative voltage drop in the space charge region can be calculated applying Poisson's equation [29]:

$$\frac{dE(x)}{dx} = -\frac{\rho(x)}{\varepsilon}, \tag{10}$$

where: E is the electric field, ρ is the charge density and ε is the permittivity.

In order to solve this nonlinear equation, the Newton-Raphson method has been applied [29].

5. Simulation environment

In the proposed approach, the simulation software runs on a network server and the user interface is provided by a web page ensuring data entry point and result presentation. One should be carefully consider operations to be performed on the client side and on the server side [30, 33]. Current server performance and network bandwidths enable all the operations to be performed on the server side. The user receives simulation results in the form he requested and all he needs to use to develop a simulation environment is a web browser. This makes the proposed solution as portable and platform-independent as possible, which can ease the cooperation between different users. However, in some situations it may be more suitable to do some data processing on the client side. Thus, we are considering a development of some client-side software that could be used when desired.

The designed simulation environment comprises four main modules as illustrated in Fig. 13. Computational resources are provided by means of Apache server running under Linux operating system. Nevertheless, the code is portable to Windows and Unix operating systems. Circuit analysis is performed with a batch-executed simulator that may include additionally implemented device models. Simulation results are processed with GNUPlot, providing graphical data representation. Finally, graphical user interface functions (circuit description and simulation parameters input point as well as results visualization) have been implemented in PHP code that dynamically generates HTML pages rendered by the user's web browser (Fig. 14).

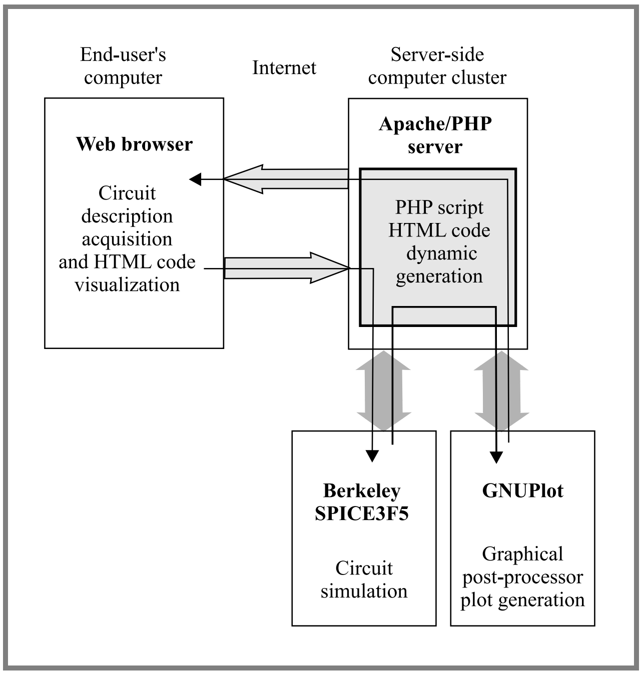

Fig. 13. Structure and data flow of the developed simulation environment.

Thanks to the proposed solution, simulation and data processing can be performed on dedicated servers, therefore not engaging the computers of end-users. Another advantage is that no additional software has to be installed on the end-user side. In order to ensure free access to the environment, it has been based on open source and GNU-licenced software.

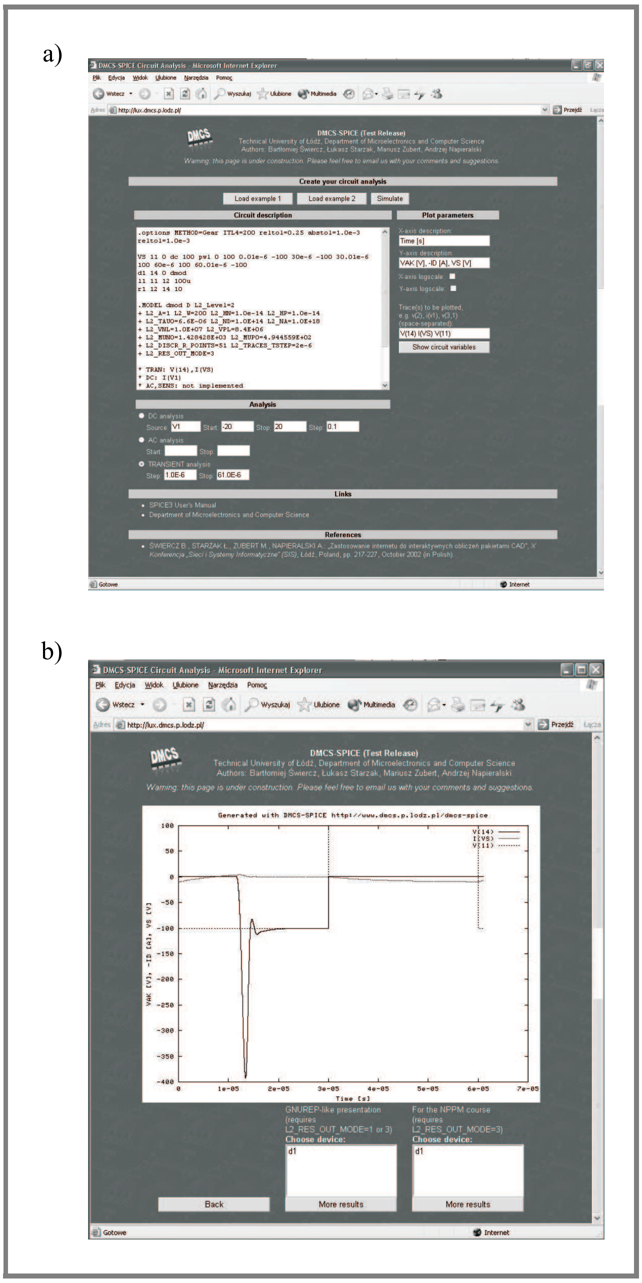

Fig. 14. Designed website – circuit data input (a) and plotted simulation results (b).

The circuit simulation core has been based on Simulation Program with Integrated Circuit Emphasis (SPICE) because of high popularity and strong position of SPICE-like simulators. This choice ensures wide accessibility of the environment.

Because of its open source character, Berkeley SPICE3F5 [26] has been used. This enables constant improvement of the designed environment by implementation of new device models and more efficient numerical algorithms. Also, the simulation core could be customized to meet the requirements of the project.

During the development of the presented environment, it turned out that linearization algorithms implemented in

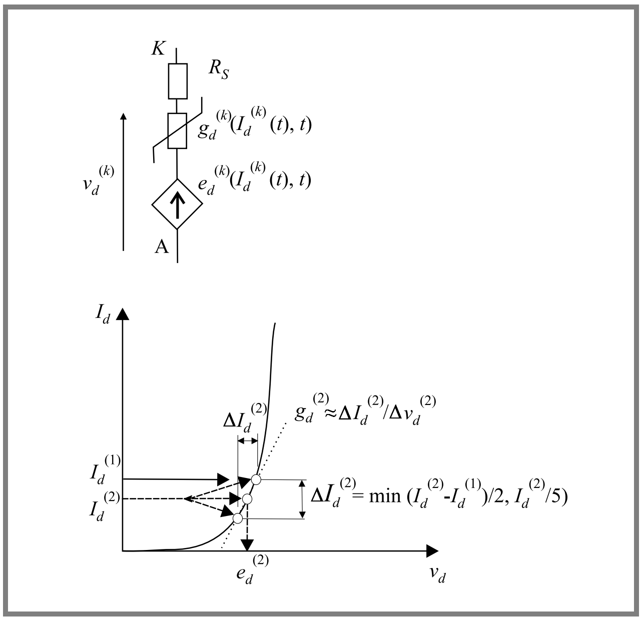

Fig. 15. The implemented linearization algorithm – power diode example.

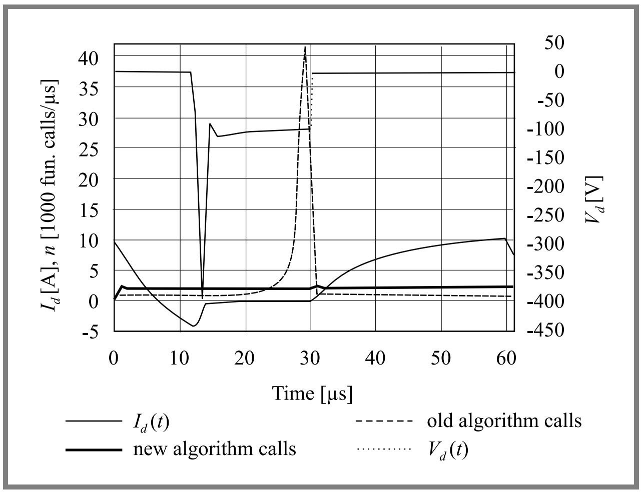

Fig. 16. Number of diode voltage evaluations using the original SPICE3F5 and the new linearization algorithm (calls per microsecond of simulation time). The calculations were made for diode reverse and forward recovery (see device current I_d and voltage V_d waveforms).

the original SPICE3F5 were unable to assure numerical convergence during the simulation of circuits containing highly nonlinear elements, such as the power diode model presented. Thus, a better-suited algorithm has been proposed and included in the developed software. During each iteration, the conductance value is calculated based on two points from its closest neighbourhood, as presented in Fig. 15. If convergence problems occur, a simplified algorithm is used. The new algorithm has permitted the number of calls of the function implementing the diode characteristics to be decreased, especially when fast dynamic processes take place in the device structure (Fig. 16).

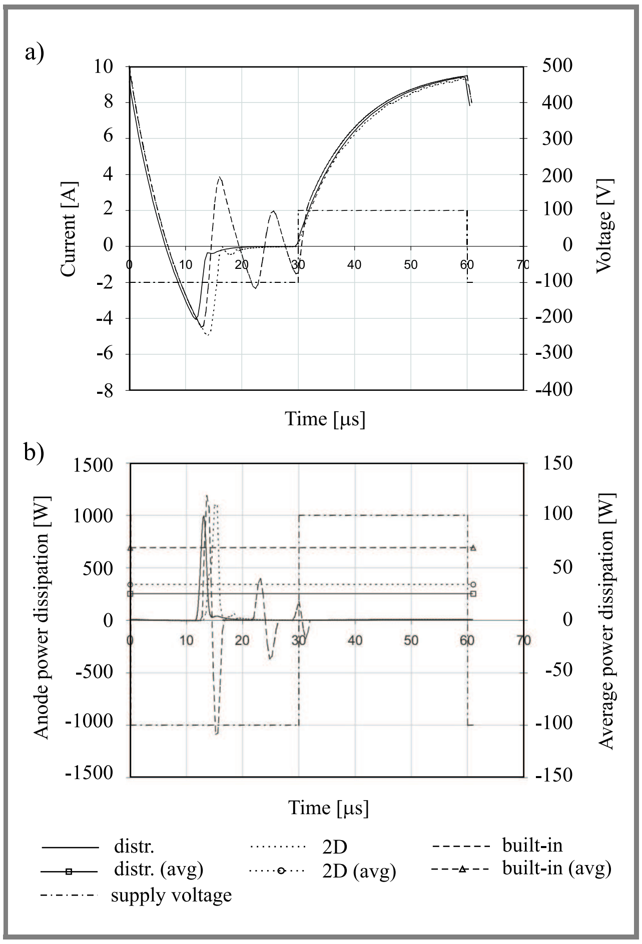

Fig. 17. Simulation results – PIN diode reverse and forward recovery with inductive load: (a) current waveforms; (b) power dissipation waveforms and average values: 2D – bidimensional model; built-in – built-in PSPICE power diode model with fitted parameters; distr. – distributed model included in the presented package.

Simulation results obtained with the model and the developed simulation environment described above are presented in Fig. 17. They show a good agreement with 2D simulations performed with the simulator MOPS [5, 8, 9, 32] based on the finite-box method, and demonstrate the erroneous results obtained with the built-in SPICE diode model with fitted parameters.

6. Conclusion

In this paper, the most important phenomena in power semiconductor devices have been discussed. As a conclusion concerning power bipolar devices we found that:

- there is a problem with the diffusion (minority carriers),
- the accuracy of computation is very poor when lumped models are used, therefore new compact models are very welcome [35],

– for correct simulation, distributed models are still necessary.

In order to make the simulation of power devices possible, a new type of a PIN diode model has been proposed in this paper. The results obtained seem to be very promising as far as the accuracy and simulation time are concerned.

In the last part of this paper, a free environment allowing the electronic circuit design by means of Internet has been presented. Thanks to the implemented PIN diode model, users have been given the possibility to simulate a realistic behaviour of circuits containing power diodes and VDMOS transistors. One should note that a professional design process might require the designer to consider electro-thermal couplings in power devices [3, 35]. However, it seems that the developed tool offers new possibilities as compared to the commercial CAD packages currently available.

The client-server system architecture and the Berkeley SPICE basis enable the application of the environment in the education of electronics and facilitate the future development of such tools in cooperation with other scientific, educational or industrial centres, as well as with individual users worldwide. Future extensions of the environment are being considered by including models of other power semiconductor devices.

Acknowledgements

I would like to express my gratitude to Dr. Marek Turowski, Dr. Mariusz Grecki, Dr. Mariusz Zubert, Dr. Grzegorz Jabłoński, Dr. Michał Furmańczyk, Dr. Marcin Janicki, Mr. Łukasz Starzak and Mr. Bartłomiej Świercz, for all scientific discussions during our common work on the fascinating topic concerning power semiconductor device modeling.

References

[1] H. Benda, A. Hoffmann, and E. Spenke, "Switching processes in alloyed PIN rectifiers", *Solid State Electron.*, vol. 8, pp. 887–906, 1965.

[2] K. Djellabi, M. Napieralska, H. Tranduc, P. Rossel, and K. Kassmi, "Modéles du transistor MOS de puissance", *Rev. Gén. Eléectr.*, no. 6, pp. 8–16, 1992.

[3] M. Furmańczyk, "Elektrotermiczna symulacja układów VLSI ze szczególnym uwzględnieniem integracji w środowisku projektowania". Politechnika Łódzka, Łódź, 1999 (Ph.D. thesis in Polish).

[4] H. Goebel, "A unified method for modeling semiconductor power devices", *IEEE Trans. Pow. Electron.*, vol. 9, no. 5, pp. 497–505, 1994.

[5] M. Grecki, "Dwuwymiarowe modelowanie bipolarnych i unipolarnych struktur półprzewodnikowych". Politechnika Łódzka, Łódź, 1995 (Ph.D. thesis in Polish).

[6] M. Grecki, G. Jabłoński, and A. Napieralski, "Transient temperature evaluation during switching process in IGBT transistor", in *Proc. 4th Int. Sem. Pow. Semicond. ISPS*, Prague, Czech Republic, 1998, pp. 119–124.

[7] M. Grecki, M. Turowski, and A. Napieralski, "2D analysis of more then one semiconductor devices together with external circuit", in *Proc. Fifth IEE Int. Conf. Pow. Electron. Var.-Speed Driv. PEVD*, London, England, 1994, pp. 441–446.

[8] M. Grecki, G. Jabłoński, and A. Napieralski, "MOPS – parallel environment for simulation of electronic circuits using physical models of semiconductor devices", in *Proc. 4th Eur. PVM/MPI Users' Group Meet.*, Kraków, Poland, 1997, pp. 478–485.

[9] G. Jabłoński: "Komputerowa analiza układów elektronicznych z zastosowaniem wielowymiarowych modeli fizycznych przyrządów półprzewodnikowych mocy". Politechnika Łódzka, Łódź, 1999 (Ph.D. thesis in Polish).

[10] M. Janicki, "Thermal modelling of semiconductor structures with special consideration of inverse problem methods for parameter estimation". Ph.D. thesis, Technical University of Łódź, Łódź, Poland, 1999.

[11] M. Janicki, M. Zubert, and A. Napieralski, "Application of inverse problem algorithms for integrated circuit temperature estimation", *Microelectron. J.*, vol. 30, pp. 1099–1107, 1999.

[12] M. Janicki, G. De Mey, and A. Napieralski, "Transient thermal analysis of multilayered structures using Green's functions", *Microelectron. Reliab.*, vol. 42, no. 7, pp. 1059–1064, 2002.

[13] M. Janicki, G. De Mey, and A. Napieralski, "Application of Green's functions for analysis of transient thermal states in electronic circuits", *Microelectron. J.*, vol. 33, no. 9, pp. 733–738, 2002.

[14] Ph. Leturcq, J. Gaubert, A. Napieralski, and Z. Khatir, "Simulation sur microcalculateur de l'ouverture des thyristors GTO", *Rev. Gén. Eléctr.*, no. 1, pp. 40–48, 1990.

[15] Ph. Leturcq, M. O. Berraies, J.-P. Laur, and P. Austin, "Full dynamic power bipolar device models for circuit simulation", in *Proc. Record. Pow. Electron. Special. Conf. PESC*, Fukuoka, Japan, 1998, pp. 1695–1703.

[16] D. Levy, G. Puppo, and G. Russo, "Central WENO schemes for hyperbolic systems of conservation laws", *Math. Model. Numer. Anal.*, vol. 33, no. 3, pp. 547–571, 1999.

[17] R. Maimouni, M. Belabadia, P. Rossel, H. Tranduc, D. Allain, C. E. Cordonnier, M. Napieralska, and J. Costa Freire, "Modèle SPICE du transistor V.DMOS pour circuits de commutations", in *Proc. L'Electronique de Puissance du Futur*, Bordeaux, France, 1988.

[18] R. Maimouni, D. Allain, M. Napieralska, H. Tranduc, P. Rossel, and C. E. Cordonnier, "Model library for power MOSFETs in switching circuits and converters", in *Proc. Mediterr. Electrotech. Conf. ELECON*, Lisbon, Portugal, 1989.

[19] R. Maimouni, P. Rossel, D. Allain, M. Napieralska, H. Tranduc, and C. E. Cordonnier, "Modele "universel" du transistor MOS de puissance pour le logiciel SPICE", in *J. d'Etud. SEE "Simulation en Electronique de Puissance du Composant au Systeme"*, Marseille, France, 1989.

[20] R. Maimouni, P. Rossel, D. Allain, M. Napieralska, H. Tranduc, and C. E. Cordonnier, "Modèle universel du transistor MOS de puissance pour le logiciel SPICE", *Rev. Gén. Eléctr.*, no. 1, pp. 49–59, 1990.

[21] M. Napieralska, H. Tranduc, P. Rossel, and C. E. Cordonnier, "Bibliotheque de transistors MOS de puissance dans le couplage des logiciels "HyperCard" et "Spice", in *Proc. 3ème Coll. Natl. SEE "L'Electronique de Puissance du Futur" EPF*, Toulouse, France, 1990.

[22] M. Napieralska, H. Tranduc, C. E. Cordonnier, J. P. Berry, and P. Rossel, "Power MOSFET's library builder for switching circuits simulation and design", in *Proc. Symp. Mater. Dev. Pow. Electron. MADEP*, Florence, Italy, 1991.

[23] M. Napieralska, "Modélisation du transistor V.DMOS pour simulation de circuits en électronique de puissance". Ph.D. thesis, INSA, Toulouse, France, 1991.

[24] A. Napieralski, M. Napieralska, and M. Grecki, "Computer laboratory for CAD of power integrated circuits", in *Computer-Aided Design and Computer-Aided Engineering in Electronic Engineering Education*, A. Filipkowski, Ed. Warszawa: WUT, 1996, pp. 135–184.

[25] A. Napieralski and M. Napieralska, *Polowe półprzewodnikowe przyrządy dużej mocy*. Warszawa: WNT, 1995 (in Polish).

[26] T. L. Quarles, A. R. Newton, D. O. Pederson, and A. Sangiovanni-Vincentelli, "SPICE3 version 3F5 user's manual", Department of EECS, University of California, Berkeley, USA, 1994.

[27] L. Starzak, M. Zubert, A. Napieralski, P. Austin, G. Bonnet, M. Marmouget, J.-L. Sanchez, and T. Bordignon, "Physical power diode model and its implementation in SABER environment", in *Proc. 8th Int. Conf. Mix. Des. Integr. Circ. Syst. MIXDES*, Zakopane, Poland, 2001, pp. 213–220.

[28] L. Starzak, A. Napieralski, and J. J. Charlot, "VHDL-AMS: a competitor for SPICE in modeling of semiconductor devices", in *Proc. Int. Conf. "Modern Probl. Radio Eng., Telecommun. Comput. Sci." TCSET*, Lviv-Slavske, Ukraine, 2002, pp. 353–356.

[29] L. Starzak, M. Zubert, and A. Napieralski, "The new approach to the power semiconductor devices modelling", in *Proc. Int. Conf. Model. Simul. Microsyst. MSM*, San Juan, Puerto Rico, 2002, pp. 640–644.

[30] B. Swiercz, L. Starzak, M. Zubert, and A. Napieralski, "An interactive website as a tool for CAD of power circuits", in *Tech. Proc. 2003 Nanotech. Conf. Trade Show (NanoTech)*, San Francisco, USA, 2003, pp. 346–349.

[31] M. Turowski and A. Napieralski, "Two-dimensional analysis of GTO switching under the influence of external circuit", *IEE Proc.-G, Circ., Dev. Syst.*, vol. 141, no. 6, pp. 483–488, 1994.

[32] M. Turowski, "Dwuwymiarowe modelowanie półprzewodnikowych przyrządów dużej mocy". Politechnika Łódzka, Łódź, 1992 (Ph.D. thesis in Polish).

[33] B. Wilamowski, A. Malinowski, and J. Regnier, "Internet as a new graphical user interface for the SPICE circuit simulator", *IEEE Trans. Ind. Electron.*, vol. 48, no. 6, pp. 1266–1268, 2001.

[34] B. Wilamowski, A. Malinowski, and J. Regnier, "SPICE based circuit analysis using web pages", in *Proc. 2000 ASEE Ann. Conf.*, Saint Louis, USA, 2000.

[35] M. Zubert, "Wielowymiarowe modelowanie sprzężonych zjawisk fizycznych w nowoczesnych strukturach półprzewodnikowych, z zastosowaniem języka VHDL-AMS i modeli o stałych rozłożonych, ze szczególnym uwzględnieniem modelowania elektrotermicznego układów VLSI oraz Smart Power". Politechnika Łódzka, Łódź, 1999 (Ph.D. thesis in Polish).

Małgorzata Napieralska received the M.Sc. degree in electronics in 1982 from Technical University of Łódź, from 1987 to 1991 she was a Ph.D. student in Laboratoire d'Automatique et d'Analyse des Systèmes du CNRS, Toulouse, France, researching in the field of modeling of power devices. In 1991 she received the Ph.D. in microelectronics from Institut Nationale des Sciences Appliquées (France). Currently she is with the Department of Microelectronics and Computer Science, Technical University of Łódź working on modeling and macromodeling of semiconductor devices, integrated circuits and micro-electro-mechanical structures and the CAD of electronic circuits. She is author or co-author of over 60 publications in scientific journals and conference proceedings, 2 books and over 20 80 scientific reports.

e-mail: MNapier@dmcs.p.lodz.pl
Department of Microelectronics and Computer Science
Technical University of Łódź
av. Politechniki 11
93-590 Łódź, Poland

Andrzej Napieralski received the M.Sc. and Ph.D. degrees from TUL in 1973 and 1977 respectively, and D.Sc. (habilitation) degree in electronics from Warsaw University of Technology (Poland) in January 1989, and in microelectronics from Université de Paul Sabatié (France) in May 1989. He received the title of Professor 1995, and become the Tenured Professor in 1999. In 2002 he has been elected as the Vice-Rector for Promotion and International Co-operation. He is author or co-author of 376 publications in scientific journals and conference proceedings, 24 books and textbooks, 15 chapters in books, 2 patents and over 90 scientific reports and expertises. Since July 1985 until September 1991, he has been a member of the research team on power devices at the CNRS LAAS, Toulouse, and Visiting Professor at the INSA of Toulouse. Since October 1991 he has been Professor at TUL. From 1992 until 1995 he was the Vice-Director of Institute of Electronics, and since 1996 he has been Director of DMCS. In 2000 he was appointment as the Chair of IEEE Poland ED Chapter and in 2001 the Editor of IEEE EDS Newsletter for Scandinavia and Central Europe.

e-mail: Napier@dmcs.p.lodz.pl
Department of Microelectronics and Computer Science
Technical University of Łódź
av. Politechniki 11
93-590 Łódź, Poland