An Interactive Website as a Tool for CAD of Power Circuits

Bartlomiej Swiercz, Lukasz Starzak, Mariusz Zubert, Andrzej Napieralski

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

ABSTRACT

The rapid development of microelectronics results in more complex semiconductor device structures and makes it necessary to use modern computer software. However, high prices and hardware requirements considerably limit the access to these CAD tools by educational institutions, students or small enterprises, and free versions of commercial software usually lack for well-implemented numerical algorithms and device models.

In order to overcome these limitations and to make accurate and modern semiconductor device models widely accessible, a website has been designed and made available to Internet users (http://www.dmcs.p.lodz.pl/dmcs-spice), allowing to perform simulations of electronic circuits.

A new distributed model of power diode is included in this package to show that the presented approach can facilitate the design process of power circuits, as accurate power device models can be made available for designers worldwide.

Keywords: SPICE, circuit simulation, power device modeling, Internet.

1 INTRODUCTION

The dynamic development of microelectronics results in more complex semiconductor device structures and makes it necessary to use modern simulation software. However, high prices and hardware requirements limit the access to professional CAD tools by some educational institutions, students or small enterprises. Free versions of commercial software are unable to solve this problem because of their limited functionality and lack of well-implemented numerical solution algorithms and device models.

This is especially true when power electronics is concerned, as it requires advanced device models to obtain simulation results of good accuracy. Such models are not available with free simulation environments.

However, the increasing popularity of the Internet can help to solve this problem. Wilamowski, Malinowski and Regnier [1] were probably the first to see the possibility of performing circuit simulations over 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 numeric data or graphics. With the constant increase of network bandwidths, time needed to receive data will become less important.

However, Wilamowski et al. [2] have put special emphasis on pay-per-use access to simulation software and platform-independent user interface provided by a web page. In this paper it is proposed to take advantage of free simulation software to develop a free simulation environment with new numerical algorithms implemented and modern device models included. The Internet is not only considered as a data transmission system but first of all as a medium for international cooperation of scientific centers, enterprises and individual users that would enable to develop better and freely accessible simulation tools and models.

2 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. It should be thoroughly considered what operations to perform on the client side and on the server side [1]. Current server performances and network bandwidths enable to perform all operations on the server side. The user receives simulation results under the form he requested and all he needs to use the developed simulation environment is a web browser. This makes the proposed solution maximally portable and platform-independent, 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 it is considered to develop some client-side software that could be used when desired.

The designed simulation environment comprises four main modules as illustrated in Figure 1. 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 user's web browser (see Figure 2).

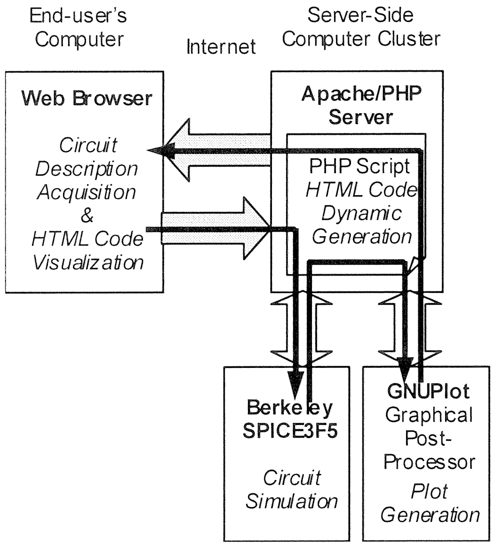
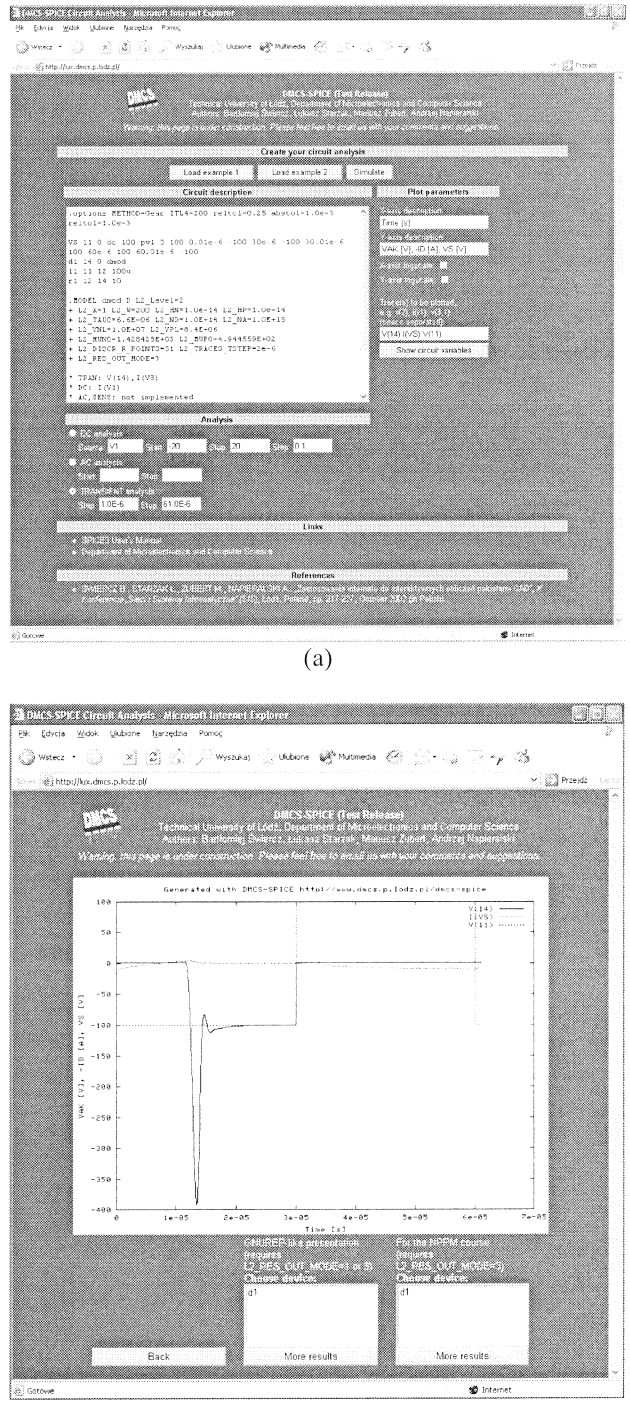


Figure 1: The developed simulation environment structure and data flow.

Thanks to the proposed solution, simulation and data processing can be performed on dedicated servers, thus not engaging the end-users' computers. 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.

The circuit simulation core has been based on SPICE[1] because of high popularity and strong position of SPICE-like simulators. This choice ensures wide accessibility of the environment, as an average electronic engineer or student has basic knowledge of SPICE circuit description format. Because of its open source character, Berkeley SPICE3f5 [3] 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 development of the presented environment, it turned out that linearization algorithms implemented in the original SPICE3f5 are unable to assure numerical convergence during simulation of circuits containing highly nonlinear elements, such as the power diode model presented in the next section. 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 neighborhood, as presented in Figure 3. If convergence problems occur, a simplified algorithm is used. The new algorithm has permitted to decrease the number of calls of the function implementing the diode characteristics, especially when fast dynamic processes take place in the device structure (see Figure 4).



(a)

(b)

Figure 2: Designed website—circuit data input (a) and plotted simulation results (b).

[1] SPICE – Simulation Program with Integrated Circuits Emphasis.

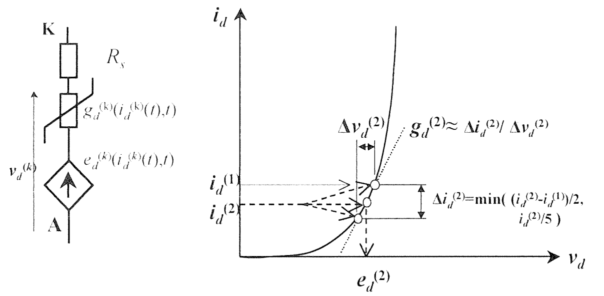


Figure 3: The implemented linearization algorithm—power diode example.

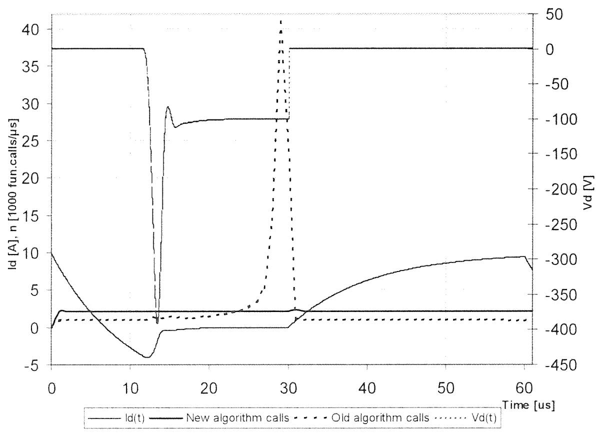


Figure 4: Number of diode voltage evaluations using the original SPICE3f5 and the new linearization algorithm (calls per microsecond of simulation time). The calculations were made for diode reverse and forward recovery (see device current *Id* and voltage *Vd* waveforms).

3 POWER CIRCUIT SIMULATION

Only low-power device models are embedded in SPICE3f5. However, these are often not suitable for power device simulation. This is because specific structures of the latter cause their behavior to differ significantly from behavior of the first. High accuracy is often needed in power circuit simulation, e.g. for overvoltage, power dissipation or turn-off time evaluation. In such cases, separate models must be developed, taking into account the distributed nature of phenomena occurring in power devices and determining their dynamic response. In this section we present a model of PIN diode [4]. Its development enables future design of models of such devices as IGBT or thyristor.

The adopted approach to power device modeling is called 'modular'. It consists in discerning in the considered semiconductor structure several regions of different physical and/or electrical nature. Then a simplified sub-model is assigned to each of them, only the most relevant phenomena being taken into account in each region. As several works have shown, this approach allows for decreasing the simulation time considerably without any important loss of accuracy [5, 6].

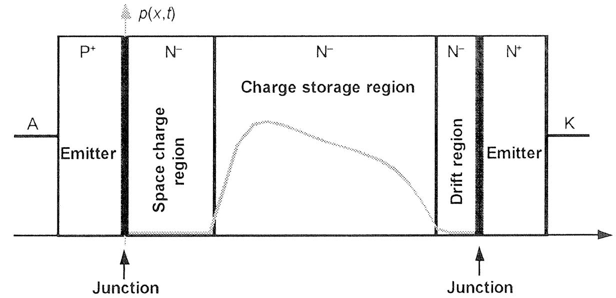


Figure 5: Modular modeling concept and its application to PIN diode.

Like most power semiconductor devices, the PIN diode contains wide and lightly doped base layer, which provides for high voltage blocking capability during the off-state and where excess carriers are stored during the on-state (see Figure 5). The width of this central layer makes it necessary to take the distributed nature of phenomena into account. The one-dimensional Benda-Spenke model [7] has been adopted for this purpose. The behavior of stored charge carriers is described by means of the ambipolar diffusion equation,

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

where: p is the carrier concentration, D is the ambipolar diffusion constant and τ is the common electron and hole lifetime. As no analytical solution to the equation (1) has been found, many different approaches to numerical solution have been presented. The developed model is based on algorithmic approach, i.e. the solution is obtained with a numerical algorithm. A new efficient CWENO[2] scheme [8] is used to ensure high computational stability and good accuracy.

After a solution of (1) is obtained, the negative voltage drop in the space charge region can be calculated after applying the Poisson's equation,

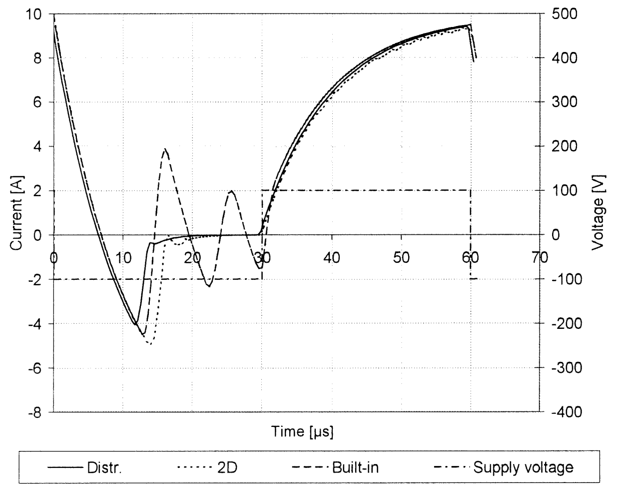
$$\frac{dE(x)}{dx} = \frac{\rho(x)}{\varepsilon}, \tag{2}$$

where: E is the electric field, ρ is the charge density and ε is the permittivity. To solve this non-linear equation, the Newton-Raphson method is used.

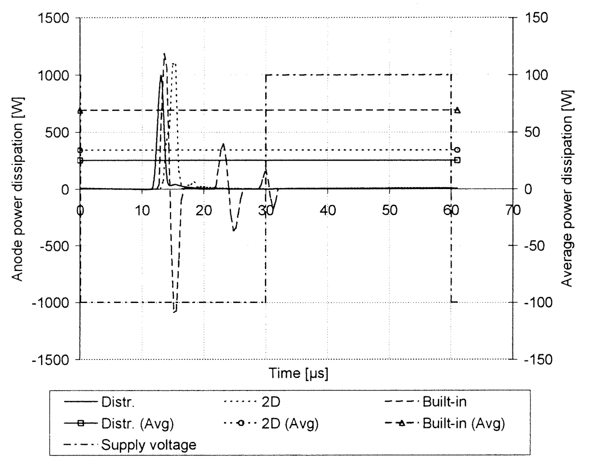
Simulation results obtained with the presented model and the developed simulation environment are presented in

[2] CWENO – Centered Weighted Essentially Non-Oscillatory.

Figure 6. They show a good agreement with 2D simulations performed with the simulator MOPS [9] and demonstrate the erroneous results got with the built-in SPICE diode model with fitted parameters.



(a)



(b)

Figure 6: 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.)

4 CONCLUSION

In the paper, a free environment allowing for 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 behavior of circuits containing power diodes. One should note that a professional design process might require the designer to take into account the electro-thermal couplings in power devices. However, it seems that the developed tool offers new possibilities as compared to currently available commercial CAD packages.

The client-server system architecture and the Berkeley SPICE basis enables the application of the environment to education of electronics and facilitates the future development of such tools in cooperation with other scientific, educational or industrial centers as well as with individual users worldwide. In the future, it is considered to extend the environment by including models of other power semiconductor devices, to implement behavioral description languages as well as multi-domain distributed models.

REFERENCES

[1] B. Wilamowski, A. Malinowski and J. Regnier, "Internet as a new graphical user interface for the SPICE circuit simulator," IEEE Trans. Industrial Electronics, Vol. 48, No. 6, pp. 1266 –1268, Dec. 2001.

[2] B. Wilamowski, A. Malinowski and J. Regnier, "SPICE based circuit analysis using web pages," Proc. 2000 ASEE Annual Conference, Saint Louis, MO, USA, June 2000.

[3] 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.

[4] L. Starzak, M. Zubert, A. Napieralski, M. Marmouget, J.-L. Sanchez et al., "Physical power diode model and its implementation in SABER environment," Proc. International Conference on Mixed Design of Integrated Circuits and Systems (MIXDES), Zakopane, Poland, pp. 213-220, June 2001.

[5] Ph. Leturcq, M. O. Berraies, J.-P. Laur and P. Austin, "Full dynamic power bipolar device models for circuit simulation," Rec. Power Electronics Specialists Conference (PESC), Fukuoka, Japan, pp. 1695-1703, 1998.

[6] H. Goebel, "A unified method for modeling semiconductor power devices," IEEE Trans. Power Electronics, Vol. 9, No. 5, pp. 497-505, 1994.

[7] H. Benda, A. Hoffmann and E. Spenke, "Switching processes in alloyed PIN rectifiers," Solid State Electronics, Vol. 8, pp. 887-906, 1965.

[8] D. Levy, G. Puppo and G. Russo, "Central WENO schemes for hyperbolic systems of conservation laws," Mathematical Modelling and Numerical Analysis, Vol. 33, No. 3, pp. 547-571, 1999.

[9] M. Grecki, G. Jablonski and A. Napieralski, "MOPS – Parallel environment for simulation of electronic circuits using physical models of semiconductor devices," 4th European PVM/MPI Users' Group Meeting, Krakow, Poland, pp. 478-485, 1997.