Electro-Thermal Circuit Simulation Using Simulator Coupling

Stefan Wünsche, Christoph Clauß, Peter Schwarz, Member, IEEE, and Frank Winkler

Abstract - The paper presents a methodology for simulating the static and dynamic performance of integrated circuits in the presence of electro-thermal interactions on the integrated circuit die. The technique is based on the coupling of a finite element method (FEM) program with a circuit simulator. In difference to other known simulator couplings a time step algorithm is used. Its implementation into simulation tools is described. The thermal modeling of the die/package structure and the extended modeling of the electronic circuit is discussed. Simulation results which indicate the capabilities of the methodology for electrothermal simulation are compared to experimental results.

Index Terms - Analog modeling with behavioral languages, circuit simulation, electro-thermal circuit simulation, finite element simulation, simulator coupling, thermal modeling.

I. INTRODUCTION

THE CURRENT trend in circuit design toward higher packaging density, higher speed and mixed-signal solutions results in the increasing of power densities in devices and on chip. Including the thermal behavior into the circuit design process of smart power integrated circuits, telecommunication circuits, and microprocessors play an essential role because of the different analog behavior at higher temperatures and temperature-caused higher failure rates.

Circuit simulators used in the design process (i.e., SABER, ELDO, Spectre) are capable of specifying the temperature of every element. However, element temperatures are defined as parameters and cannot be changed during the simulation run. This means, the circuit simulators do not allow the simulation of selfheating and thermal interactions in integrated circuits. The reason for this fact are the missing electro-thermal models of typical devices (i.e., BJT, MOS, ...).

Since the beginning of the 1990's, we have encountered intensified activities in the field of thermal modeling of integrated circuits and electro-thermal simulation. There are two principle approaches to simulate the electro-thermal behavior of integrated circuits - the direct method and the relaxation method (see Fig. 1).

The *direct method* is based on modeling the thermal and electronic behavior of the circuit for a single simulation tool [1],[2]. For instance, the thermal behavior can be described by an analog behavioral language [3], [4] and the electric circuit is given by a netlist. The simulation is processed by circuit and system simulators like SABER, ELDO or Spectre. The

Manuscript received October 31, 1996; revised May 15, 1997. This work was supported by the German government Project SFB 385/D4.

The authors are with the Department EAS Dresden, Fraunhofer Institute of Integrated Circuits, Dresden D-01069 Germany.

Publisher Item Identifier S 1063-8210(97)06351-8.

thermal modeling process is complicated due to the complex structure of die, header, and package.

The *relaxation method* is based on the coupling of a thermal and a circuit simulator [5]-[7]. The set of thermal and electric equations is solved separately. In general, the modeling process for the relaxation method is not so complicated, because the special simulators are designed for a particular thermal or electric problem. The know-how of the relaxation method is to realize data transfer, synchronization, convergence control, and the mathematical background.

The objective of the thermal modeling process is to create a model which represents the thermal behavior of the physical system. The temperature-dependent thermal conductivity of semiconductor material which leads to a nonlinear heat equation has to be considered in the modeling process. The thermal modeling process requires material and geometry parameters which can be extracted from a layout tool. Finally, the created thermal models should be verified using measurements. A large variety of approaches dedicated to thermal modeling can be found in literature. Finite difference [8],[9], finite element [10],[11], boundary element approaches [12], fourier series [13], analytical solutions [14] and thermal networks [15],[16] are employed. Besides these methods to solve the heat equation, methods to extract models from experimental results or thermal simulation are applied [17],[18]. Most of them are valid only in steady-state cases and for linear heat equations. Fig. 3 gives an overview of the thermal modeling approaches.

Apart from the thermal modeling process itself, both methods - the direct method and the relaxation method - require some modifications of the models in the circuit simulators. Standard device models which already include thermal dependencies of the electronic parameters have to be extended to electro-thermal models. Fig. 2 shows the extended electrothermal model of the BJT with an additional thermal pin T. The "through" value at this pin is the power dissipation of the device and the "across" value is the temperature of the device [19]. The power dissipation value is calculated from the electronic part of the model and the temperature at the thermal pin influences the electronic behavior of the device. Now the temperature is a signal which changes during the simulation time. In the electro-thermal model we have to calculate the active power which generates the heat.

The paper focuses on the relaxation method to realize a coupled electro-thermal simulation. The methodology of the simulator coupling is reported in section II. Section III discusses the implementation of the coupling between the FEM program ANSYS [20] and the circuit and system simulator



Direct Method

Fig. 1. Principle approaches to perform electro-thermal simulations.



Fig. 2. Electro-thermal BJT model.

SABER [21]. The realization of coupled simulation is shown at the example of a current mirror circuit as well as an op amp circuit in Section IV. Section V points out the advantages and disadvantages of the realized electro-thermal simulation and discusses further work.

II. METHODOLOGY OF SIMULATOR COUPLING

The simulators we want to couple should have some typical properties. A time incremental simulation principle is required. The coupling algorithm needs the repetition of the calculated time interval. Furthermore, the simulators should have properties to receive and to send calculated values to another simulator. The FEM program ANSYS and the circuit and system simulator SABER meet these requirements. They support time domain simulations and offer behavioral languages and interfaces to languages like C or FORTRAN. Therefore, the user can implement his own communication interfaces. The repetition of a calculated time step is available in both simulators, but in SABER, a MAST (behavioral description language) model cannot initiate a restart of the simulator.

In difference to other known simulator couplings the method we used is based on an automatic time step algorithm. Time step size is variable and depends on the convergence behavior of the system equations. If there are only small changes of the system values, large time steps are chosen. Large changes of these values lead to small time steps. Similar methods of time step control are used efficiently in circuit simulators.

The implemented algorithm is sketched out in Fig. 4. Assume that the results are processed until t_b . $t_x > t_b$ may be chosen. Six steps are required to calculate the next time interval as follows.

- 1) Extrapolation of the SABER results in interval $[t_b, t_x]$
- ANSYS calculation with the extrapolated SABER values (interval [t_b, t_x])

Relaxation Method



- 3) Processing of SABER using the results of the ANSYS simulation (interval $[t_b, t_x]$) until 4 is active.
- 4) During simulation the difference between the extrapolated and the calculated SABER value is measured. If it is larger than error criterion Δ_1 the end of the time interval is defined (interval $[t_b, t_e]$).
- 5) ANSYS uses the SABER results to calculate interval $[t_b, t_e]$ again, which leads to Δ_2 .
- 6) The calculation of the time interval $[t_b, t_e]$ is completed. The new time interval is chosen.

This algorithm of time step control was implemented because a MAST model can not initiate the repetition of a calculated time step in SABER. So only one repetition in ANSYS and one SABER calculation per interval $[t_b, t_e]$ is performed. The error criterion is defined by a relative and an absolute convergence parameter. The values of these parameters depend on the simulated system and have to be chosen by the user.

The heuristic procedure of stepsize control may be interpreted as relaxation method [22] performing only one iteration. Therefore, it is related to the simple noniterative timing analysis [23]. The arising disadvantages [24] will be compensated by the automatic $[t_b, t_e]$ time window control. The characteristics of the procedure following below have been determined by numeric test calculations:

- Convergence should not be expected when the iterations with constant stepsize do not converge;
- Knowing the consistent initial values is a fundamental precondition for good convergence behavior;
- The convergence behavior may be positively affected by the choice of the following timestep [t_b,t_x], which should not be much larger than the last one.

It is a main advantage of the algorithm that no iterations are necessary. A disadvantage is the lack in error control. Δ_2 is not always a sufficient error measure. Anyway, for well-posed problems and carefully chosen stepsizes t_x there may be obtained acceptable results, as shown in the enclosed examples. Another simulator coupling method using both time step control and iteration is under investigation.

III. IMPLEMENTATION OF THE SIMULATOR COUPLING

The implementation of the simulator coupling requires a control routine which realizes time step control and convergence test. This routine can be a stand-alone software or part of the coupled simulators. The second approach was chosen



Fig. 3. Thermal modeling approaches.



Fig. 4. Algorithm of automatic time step control.

because both simulators have the possibility to include routines, i.e control routines. In [25], an implementation is described in which ANSYS is the master and controls the coupling. The simulator coupling pointed out in this paper is based on SABER as the master.

Analog behavioral languages like MAST from SABER or HDL-A from ELDO [26] are suitable to implement complicated models into simulators. In general, there is also the possibility to include models written in C. The time step algorithm was implemented using the analog-digital interface of MAST. The interface supports the combination of time domain analog models with event-driven digital models. Since all system parts are normally analog components the analog modeling capabilities were used.

Only the time step control algorithm requires the eventdriven modeling. A MAST model was implemented which realizes the convergence control. This model compares the difference between extrapolated and calculated SABER values during the analog simulation. If the difference is larger than error criterion Δ_1 a digital event is created at current time t_e and it is used as the end of the current time interval. SABER sends the calculated data from $[t_b, t_e]$ to ANSYS. A routine written in APDL (ANSYS Parametric Design Language) controls the simulation and the data transfer in ANSYS. Fig. 5 shows a simplified structure of the MAST model.

For data communication between ANSYS and SABER file transfer as well as PVM (Parallel Virtual Machine) is used.





Fig. 5. Structure of the MAST model.

In the case of file transfer one simulator writes the results into a file and the other simulator reads these data. A more comfortable and faster solution is the use of PVM [27]. Parallel Virtual Machine is a software which supports the data exchange between different programs on several platforms. PVM provides both a C and FORTRAN library of functions to initialize connections between hosts, send and receive data,

*DECK	(USER04 USER PARALLEL FUNCTION USER04 (INTIN.DPIN.CH4IN.CH8IN)	SASI
	implicit none EXTERNAL WRINQR	
	INTEGER WRINQR	
	include '/home/sim/kosim/pvm3/include/fpvm3.h'	
	INTEGER USER04, INTIN(12), IOTT	
	DOUBLE PRECISION DPIN(12)	
	CHARACTER*4 CH4IN(12)	
C	CHARACTER [*] 8 CH8IN(12)	
C C	INTIN(2) switch EIL E-(=0)/PVM(=1)-coupling	
č	INTIN(3) number of values to send (ANSYS -> SABER)	
С	INTIN(4) number of values to receive (SABER -> ANSYS	S)
С	DPIN(i) data to send/receive	
С	Declaration of parameters, variables,	
С	FILE-coupling	
-	if(INTIN(2) .eq. 0) then	
С	write data to file	
С	create ready signal	
С	waiting for data from SABER	
С	read SABER data	
С	PVM-coupling	
	else if (INTIN(2) .eq. 1) then	
	if (begin) then	
С	create connection to pvm-daemon	
C	send data to SABER	
C	if (finish) then	
С	delete connection to pvm_daemon	
0	write received values to ANOVO	
U	RETURN	
	END	

Fig. 6. Structure of the ANSYS user command file.

etc. The data transfer procedure in SABER is written in C and is included into the MAST model. In ANSYS a user command written in FORTRAN was implemented. It is called by the ANSYS control routine (see Fig. 6)

IV. TEST SYSTEMS AND SIMULATION RESULTS

The simulator coupling was employed to analyze self-heating effects in a current mirror circuit as well as thermal interactions in an operational amplifier circuit.

A. Current mirror circuit

The self-heating of a current mirror circuit (see Fig. 8) described in [28] was analyzed using. Due to the power dissipation of the integrated bipolar junction transistors a thermal gradient on chip is produced. The temperature difference between the transistors Q_1 and Q_2 influences the electric behavior.

In SABER the BJT model was extended to an electro-thermal model. The analog behavioral language MAST was used to implement the model. The current mirror circuit was modeled using a SABER netlist, whereas the die/header structure was modeled in ANSYS. The three-dimensional (3-D) thermal FEM model was composed of 608 finite elements using a simplified model of the real system (see Fig. 7). During coupled simulation ANSYS sends the temperature T_Q of the devices and gets the power dissipation P_V of the transistors from SABER. In SABER the received temperatures are used as values of the temperature sources T_{Q1} and T_{Q2} . The power dissipations P_{V1} and P_{V2} calculated by SABER define timedependent loads in ANSYS (see Fig. 8). A transient simulation of the electro-thermal system was performed. The voltage of V_{cc} was switched on at time t = 0(see Fig. 8). The step response of the system can be observed. The increasing temperature as well as the collector currents of transistor Q_1 and Q_2 are depicted in Fig. 9. While the collector current of Q_1 depends on its temperature T_1 , the collector current of Q_2 is influenced only by the temperature difference between both transistors

It takes the coupled simulation four hours to simulate until end time of 2.5 ms is reached (Sun Sparc20). Most of the simulation time is used by ANSYS to calculate the complicated thermal model. There is a good agreement between simulated and measured data [28].

B. Op Amp Circuit

The simulated schematic of the 741 is depicted in Fig. 10 [29]. Fig. 11 shows the simplified layout of the circuit. The dissipation characteristic of the class AB-output stage with zero dissipation at $V_o = 0$ and dissipation peaks at $V_o = +V_{CC}/2$ and $-V_{EE}/2$ leads to small temperature gradients on chip [30]. Thus the critical transistor pairs Q_1 - Q_2 , Q_3 - Q_4 , and Q_5 - Q_6 of the input stage experience different temperatures. Assuming a 1k Ω resistor R_l loading at the output, a severe distortion of the transfer characteristic is produced by electro-thermal interactions. The distortion can be analyzed by processing electro-thermal simulation.

To simulate the electro-thermal behavior the die/header structure was modeled for thermal analysis with ANSYS. The three-dimensional thermal FEM model was composed of 5120 finite elements. The circuit simulation of the electronic part uses a SABER netlist and MAST models for the transistors, the coupling algorithm, and additional components. The implementation considers the power of the output stage and the thermal sensitivity of transistor pair Q_1 - Q_2 . Including other temperature sensitive transistor pairs like Q_3 - $Q_4, Q_5 - Q_6$ is easily to realize. During the coupled simulation ANSYS calculates the temperature T_{Q1} and T_{Q2} of the transistor devices and sends it to SABER. SABER simulates the circuit behavior using the received device temperatures. The power dissipations of Q_{14} and Q_{20} calculated by SABER define time-dependent loads in ANSYS.

The coupled simulation was employed to analyze the thermal influenced dc characteristic. A quasi static transient simulation of the electro-thermal system with increasing input voltage V_i was performed. The simulation results in Fig. 13 show the power of the output stage transistors. The temperature difference between transistor Q_1 and Q_2 is depicted in Fig. 14. The dc curve is compared to the dc curve without thermal interactions (Fig. 12). The simulation results agree with simulation and measured data published in [31]. It takes the coupled simulation eight hours to simulate the behavior of the electro-thermal system on a Sun Sparc20.

V. CONCLUSIONS

A method to simulate the electro-thermal behavior of integrated circuits was described. The simulation results show the capability of the time step control technique for electro-



Fig. 7. Thermal modeling process.



Fig. 8. Models and transferred values.



Fig. 9. Simulation results.



Fig. 10. Simplified circuit schematic of op amp 741.

thermal simulation. The method proposed offers the advantage to use an accurate thermal simulator which also handles the nonlinear heat equation of semiconductor material. Furthermore, the modeling process of the thermal and the electronic part is easy to be carried out. Standard simulation tools like



Fig. 11. Simplified layout of 741.



Fig. 12. DC transfer characteristic of the 741 op amp.



Fig. 13. Power dissipated in the output stage transistors.

the general purpose FEM program ANSYS and the circuit simulator SABER which are often integrated in the design process can be used to perform electro-thermal simulations. The method is applicable for other widely used simulation tools like ELDO and Spectre or special thermal simulators. The disadvantage of the relaxation method is the time-consuming simulation for large systems. Further work will be directed to the extraction of thermal networks and its implementation into circuit simulators with analog behavioral languages.

+0.435 +.0.484 +0.274 22+1404 +774.7 P +6



Fig. 14. Temperature difference between Q1 and Q2.

REFERENCES

- K. Fukahori, "Computer simulation of monolithic circuit performance in the presence of electro-thermal interactions," Ph.D. dissertation, Univ. California, Berkeley, 1977.
- [2] A. R. Hefner, and D. L. Blackburn, "Thermal component models for electro-thermal network simulation," in *Proc. 9th IEEE Semi-Therm Symp.*, 1993, pp. 88-98.
- [3] H. A. Mantooth, and A. R. Hefner, "Electro-thermal simulation of an IGBT PWM Inverter," IEEE 1993, pp. 75-84.
- [4] K. Hofmann, and M. Glesner, "Modellierung von Systemkomponenten in HDL-A," in Workshop Hardwarebeschreibungssprachen und Modellierungsparadigmen, Frankfurt A. M., Feb. 1995.
- [5] W. V. Petegem, B. Geeraerts, and W. Sansen, "Electrothermal simulation and design of integrated circuits," *IEEE J. Solid-State Circuits*, vol. 29, pp. 143-146, Feb. 1994.
- [6] W. H. Kao, and W. K. Chu, "ATLAS: An integrated thermal layout and simulation system for IC," in *Proc. European Design and Test Conf.*, Paris, France, 1994, pp. 43-48.
- [7] S.-S. Lee, "Simulation of electro-thermal interactions in integrated circuits," Ph.D. dissertation, Carnegie Mellon Univ., Pittsburgh, PA, June 1993.
- [8] K. Fukahori, and P. R. Gray, "Computer simulation of integrated circuits in the presence of electrothermal interactions," *IEEE J. Solid-State Circuits*, vol. SC-11, pp. 834-846, Dec. 1976.
- [9] G. Digele, S. Lindenkreuz, and E. Kasper, "Fully-coupled dynamic electro-thermal simulation," in *Proc. 2nd THERMINIC Workshop*, Budapest, Hungary, Sept. 25-27, 1996, pp. 73-77.
- [10] J.-L. Blanchard, and J.-M. Morelle, "Two software approaches for the thermal simulation of electronic equipment," ACM Trans., 1994.
- [11] W. K. Chu, and W. H. Kao, "A three-dimensional transient electrothermal simulation system for IC's," in *Proc. 1st THERMINIC Workshop*, Grenoble, France, Sept. 25-26, 1995, pp. 201-207.
- [12] M. Driscart, and G. De Mey, "A boundary element method approach to thermal modeling of microelectronic components and packages," in *Proc. Int. Boundary Element Symp.*, Nice, France, May 1990, pp. 394-404.
- [13] V. Szekely, A. Csendes, and M. Rencz, "μS-THERMANAL: An efficient thermal simulation tool for microsystem elements and MCM's," in *Proc. 2nd THERMINIC Workshop*, Budapest, Hungary, Sept. 25-27, 1996, pp. 13-20.
- [14] K. Poulton, K. L. Knudsen, J. J. Corcoran, "Thermal design and simulation of bipolar integrated circuits," *IEEE J. Solid-State Circuits*, vol. 27, pp. 1379-1387, Oct 1992.
- [15] K. Lu, P. Halloran, T. J. Brazil, "Simple method to simulate diode selfheating using SPICE," *Electron. Lett.*, vol. 28, No. 17, Aug. 1992.
- [16] R. S.Vogelsong, and Ch. Brzezinski, "Extending SPICE for electrothermal simulation," in *Proc. IEEE Custom Integrat. Circuit Conf.*, 1989.
- [17] N. Sabry, A. Bontemps, V. Aubert, and R. Vahrmann, "Realistic and efficient simulation of electro-thermal effects in VLSI circuits," in

Proc. 2nd THERMINIC Workshop, Budapest, Hungary, Sept. 25-27, 1996, pp. 95-103.

- [18] V. Szekely, A. Poppe, M. Rencz, A. Csendes, and A. Pahi, "Self-consistent electro-thermal simulation: Fundamentals and practice," in *Proc. 1st THERMINIC Workshop*, Grenoble, France, Sept. 25-26, 1995, pp. 188-194.
- [19] A. Vachoux, and K. Nolan, "Analog and mixed-level simulation with implications to VHDL," in *Fundamentals and Standards in Hardware Description Languages.*, J. Mermet, Ed. New York: Kluwer, 1993.
- [20] ANSYS User Guide, Swanson Analysis Systems, Inc. Sept. 15, 1994.
- [21] M. Vlach, "Modeling and simulation with SABER," in *Proc. 3rd Annu.* ASIC Seminar and Exhibition, Rochester, NY, 1990, T-11.1 - T-11.9.
- [22] J. White, and A. Sangiovanni-Vincentelli et al., "Waveform relaxation: Theory and practice." *Trans. Soc. Comput. Simul.*, vol. 2, pp. 95-133, 1985.
- [23] A. R. Newton, "Techniques for the simulation of large-scale integrated circuits." *IEEE Trans. Circuits* Syst., vol. CAS-26, pp. 741-749, 1979.
- [24] A. R. Newton, and A. Sangiovanni-Vincentelli, "Relaxation-based electrical simulation," *IEEE Trans. Electron. Devices*, vol. ED-30, pp. 1184-1207, Sept. 1983.
- [25] C. Clauß, R. Gruschwitz, P. Schwarz, and S. Wünsche, "Simulation mikrosystemtechnischer Aufgaben mit gekoppelten Simulatoren," in *Proc. 2 Chemnitzer Fachtagung*, Mikrosystemtechnik-Mikromechanik, TU Chemnitz-Zwickau, 16./17.10.1995, pp. 92-101.
- [26] The HDL-A Language Reference Manual, ANACAD Elec. Eng. Software, Apr. 1994.
- [27] A. Geist, and A. Beguelin, PVM: Parallel Virtual Machine: A Users Guide and Tutorial for Networked Parallel Computing, Cambridge, MA: The MIT Press, 1994.
- [28] P. C. Munro, and F.-Q. Ye, "Simulating the current mirror with a selfheating BJT model," *IEEE J. Solid-State Circuits*, vol. 26, pp. 1321-1324, Sept. 1991.
- [29] R. G. Gray, and R. G. Meyer, Analysis and Design of Analog Integrated Circuits. New York: Wiley, 1994.
- [30] J.-E. Solomon, "The monolithic op amp: a tutorial study," *IEEE J. Solid-State Circuits*, vol. SC-9, pp. 314-332, Dec. 1974.
- [31] S.-S. Lee, and D. J. Allstot, "Electrothermal simulation of integrated circuits," *IEEE J. Solid-State Circuits*, vol. 28, Dec. 1993.

StefanWünsche received the diploma degree in electrical engineering from the University of Technology of Chemnitz, Germany, in 1993.

Since 1993, he has been with the Fraunhofer Institute of Integrated Circuits, Dresden, Germany. His research interest includes modeling and simulation of microsystems. He is engaged in the field of electro-thermal simulation with the objective to include thermal interactions into VLSI design process.

Christoph Clauß received the diploma degree in mathematics from the Technical College of Karl-Marx-Stadt, GDR, in 1974. He received the Ph.D. degree in informatics from the Academy of Sciences of the GDR in 1990.

In 1992, he joined the Fraunhofer Institute of Integrated Circuits and worked on circuit simulation. His general interests are numerical mathematics, especially integration of (algebraic) differential equations.

Peter Schwarz (M⁶93) received the diploma degree, the Pd.D. degree, and the habilitation degree from the Dresden University of Technology, Germany.

He spent 14 years with a measurement device company, ten years with the Academy of Science, and joined the Fraunhofer Institute of Integrated Circuits in 1992. His main fields of interest are modeling, simulation, and optimization of circuits, mixed-signal simulation, and their application in electronics, microsystems technology, and telecommunication.

Frank Winkler received the diploma degree in computer science from the University of Technology of Chemnitz, Germany, in 1993.

He joined the Fraunhofer Institute of Integrated Circuits in 1993. He is working in the domains of graphical user interfaces, interprocess communication, and simulator coupling on the fields of algorithms and system dependent implementations.