Global Modeling of RF and Microwave Circuits

Michael B. Steer*, Carlos Christoffersen[?], Senthil Velu* and Nikhil Kriplani*

*Department of Electrical and Computer Engineering
North Carolina State University, Raleigh, North Carolina, USA,
email: m.b.steer@ieee.org, web page: http://www4.ncsu.edu/~mbs

[?]Department of Electrical Engineering
Lakehead University, Thunder Bay, Canada
email: cechrist@vision.lakeheadu.ca

Abstract. *Advances in computer hardware and algorithmic technology have brought us to the brink of being able to model large mixed-signal circuits incorporating comprehensive modeling of the full physics of devices in a circuit model. In this paper we describe an approach to delivering a revolutionary modeling tool that implements new modeling and simulation abstractions, fast linear and nonlinear solvers, full-wave EM modeling for on-chip parasitics and integrated RF/microwave circuit design modeling, digital and analogue behavioral modeling, and advanced electrothermal modeling.*

1 INTRODUCTION

Mixed signal circuits and systems, and especially RFICs, necessitate a re-evaluation of the underlying approaches used for modeling, simulation and design for chip- and package-level circuits and systems. It is believed that the basic approach to circuit simulation must change. Today the predominant circuit simulator is Spice (and its derivatives), which embodies the most common circuit simulation technology. The basic architecture was established decades ago with peculiar restrictions on model abstraction (e.g. nonlinearities being expressed as voltage or current controlled sources), the use of a global reference node, and the integral use of Newton's Method in element discretization. One result is that transistor modeling has become increasingly complicated in large part because physical behavior simply cannot be adequately abstracted to fit the Spice approach. The integral role of the nonlinear solver in an element model is required to accommodate local convergence control.

2 The Global Modeling Concept

From the above discussion it is clear that critical to global modeling is a transient, non-linear simulation environment capable of multi-physics abstraction. Spice represents an encapsulation of the computer aided circuit analysis theory of the late 1960s. We see several fundamental problems in the circuit abstraction of this thirty-year-old technology. This Section addresses these limitations, their consequences, and what is required to 'move us on to the next tier of circuit modeling.' We have been addressing this by developing a Global

Modeling Concept and developing a commercial-strength simulator called Transim.

2.1 Comprehensive and Fast Methods for Multi-Physics Modeling and Simulation

The aforementioned needs for electrical modeling with electromagnetic accuracy cannot be addressed successfully without attention paid to handling the complexity of state-of-the-art and future designs. What is needed is a class of Fast Extractors of Electrical Parasitics that provide the required electromagnetic accuracy with computational efficiency that matches or even surpasses that of the quasi-static extractors in use today. Furthermore, the output of these extraction tools needs to be in a format suitable for network-oriented system-level simulation both in the time and the frequency domain.

2.2 Simulation Approach

Concurrent electromagnetic, circuit and thermal analysis are supported. Fig. 1 illustrates the model integration concept [1]?[4]. In this circuit-oriented approach the high-level circuit abstraction is retained, and the results of EM analysis of the spatially distributed circuit are incorporated into the circuit framework. For example, the following analogy is used to represent thermal systems as circuits. The temperature of a surface where heat is being exchanged is a potential, equivalent to voltage in an electrical circuit, and the heat exchanged is a flux, equivalent to a current in an electrical circuit. Thus we have a general unifying concept of fluxes and potentials that leads to a universal error formulation: the sum of fluxes at a node must be zero and all potentials calculated for a particular node must be the same. This analogy is used to model the thermal subsystem as a circuit. The different physical thermal components are thus modeled independently and connected to form a thermal circuit.

A nonlinear device in Transim is described using a formulation based on state variables. This provides great flexibility for the design of new models. The state variables can be chosen to achieve robust numerical characteristics. Virtually any physical model can be implemented in the new modeling scheme. The number of nonlinear unknowns resulting from our formulation is generally much smaller than the number of unknowns in conventional circuit analysis. This results in an important reduction in the computation time. Another of the outstanding new features is the *generic element evaluation mechanism*. With this, the primitive model equations are ‘wrapped' in analysis-specific generic functions and so there is no need to write a separate routine for each analysis type. The derivatives of the model equations are obtained using *automatic differentiation* (AD). This technique allows the calculation of derivatives free of truncation errors at a small multiple of the run time required to evaluate the original function with little additional memory required. It is important to note that AD is not numerical differentiation and the same accuracy achieved by evaluating analytically developed derivatives is obtained. No other circuit simulator provides the same level of flexibility for the addition of new nonlinear device models and circuit analysis algorithms.

Transim also supports the local reference node concept. The conventional nodal specification enables circuit elements to be connected in any possible combination and only one reference node (commonly called the global reference node or simply ground) is used. With spatially distributed circuits it is possible to make non-physical connections such as connecting a non-spatially distributed element, say a resistor, across two physically separated (relative to the wavelength) parts of the circuit, *e.g.* the opposite ends of a transmission line. The local reference concept avoids this kind of problem and therefore is fundamental for the analysis of spatially distributed circuits as well as for simultaneous thermal-electrical simulations.

2.3 Design Flow Methodology and Model Reduction

As full wave effects become important for on-chip interconnect, managing the design at appropriate levels of extraction will become critical. On-chip interconnect structures are highly irregular and three-dimensional in nature. Even though the modeling and simulation tools proposed here will be able to handle order of magnitude more complex problems than previous tools can, whole-chip extraction and simulation at the required level of fidelity will continue to be impractical. We are developing appropriate levels of abstraction to enable overnight full-chip full-wave extraction. One method identifies return path structures that result in simplified modeling strategies. Another technique identifies full-wave interconnect macromodels than can then be used in Transim, instead of full fidelity models. Reduced order modeling of nonlinear circuitry is essential if whole chips and mixed signal systems are to be modeled. The closest existing industrially accepted model for nonlinear reduced order models is the IBIS model used in evaluating package-level and board-level signal integrity http://www.eigroup.org/ibis/ibis.htm. The IBIS model captures the characteristics of off-chip drivers and receivers. We believe that the only viable way to proceed is to extend this concept to on-chip modules so that, for example, there would be IBIS-compatible models for on-chip modules.

3. BSIM Model Simulation

Many device models have been successfully implemented in Transim. The speed at which models can be implemented is very fast and the amount of code that must be developed is typically 10 to 20 times less than with modeling approaches in conventional simulators. An example is the implementation of the BSIM4 model which requires 200 pages in the original Spice implementation but in only 17 pages in the Transim implementation. The Transim implementation was completed in 7 months.

4. CONCLUSION

Transim has a GNU public license that facilitates its linkage with the EDA environment of interested industry. Transim implements the ideas discussed above and constitutes a paradigm shift in the development of a comprehensive simulation environment for rigorous, multi-physics, global, non-linear simulation. In that respect, Transim is a rather unique simulation environment. Transim's approach to model abstraction provides the versatility needed to tackle the complexity and hybrid nature of integrated systems.

REFERENCES

[1] W. Batty, C. E. Christoffersen, A. J. Panks, S. David, C. M. Snowden and M. B. Steer, "Electrothermal CAD of power devices and circuits with fully physical time-dependent compact thermal modeling of complex non linear 3-d systems," *IEEE Trans. Components and Packaging Technology*, Dec. 2001, pp. 566?590.

[2] C. E. Christoffersen and M. B. Steer, ‘‘State-variable-based transient circuit simulation using wavelets," *IEEE Microwave and Guided Waves Letters*, Vol. 11. April 2001, pp. 161?163.

[3] C. E. Christoffersen, U. A. Mughal, and M. B. Steer, "Object oriented microwave circuit simulation," *Int. J. on RF and Microwave Computer Aided Eng.,* Vol. 10, Issue 3, 2000, pp. 164?182.

[4] C. E. Christoffersen and M. B. Steer, "Implementation of the local reference node concept for spatially distributed circuits," *Int. J. on RF and Microwave Computer Aided Engineering,* Vol. 9, No. 5, September, 1999, pp. 376?384.

[5] C. Christoffersen, M. Ozkar, M. B. Steer, M. G. Case and M. Rodwell, "State variable-based transient analysis using convolution," *IEEE Trans. Microwave Theory Techniques,* June 1999, pp. 882-889.

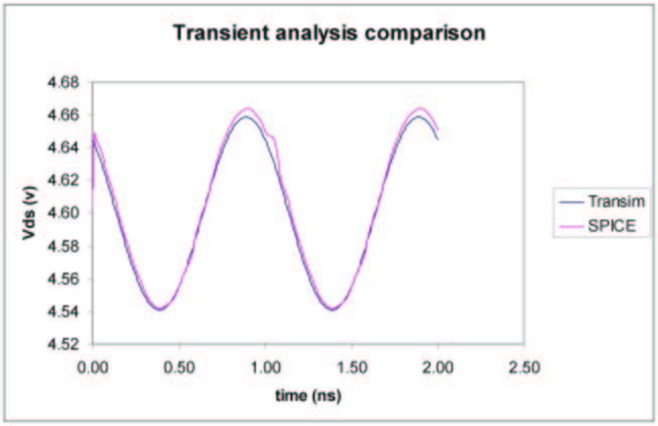

Figure 5: Comparison of transient Spice and TRANSIM simulations of a class A BSIM4 amplifier circuit.

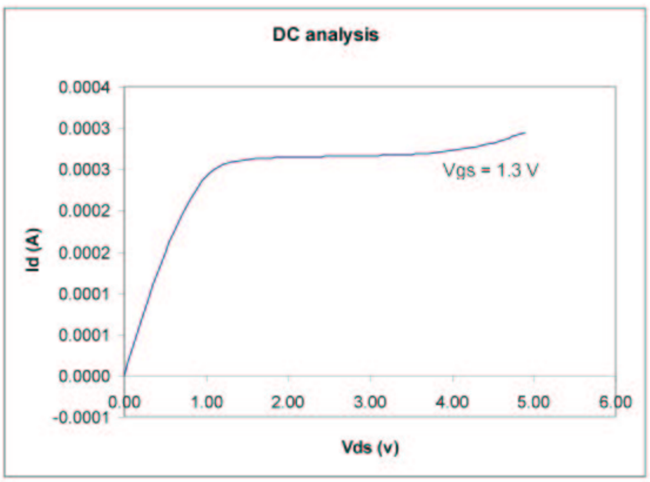

Figure 4. Output characteristic of BSIM4 transistor.

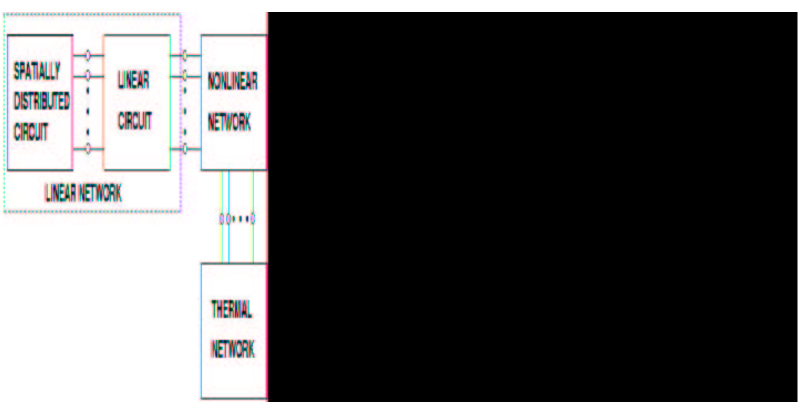

Figure 1: Partition of a system into spatially distributed and lumped linear circuit, nonlinear network, and thermal parts.