

# Global Modelling of RF and Microwave Systems Integrating Circuit, Thermal and EM Analyses

Michael Steer<sup>§</sup>, Paul Franzon<sup>§</sup> and Andreas Cangellaris<sup>¶</sup>

<sup>§</sup>Electronics Research Laboratory, Department of Electrical and Computer Engineering  
North Carolina State University [mbs@ncsu.edu](mailto:mbs@ncsu.edu), [paulf@ncsu.edu](mailto:paulf@ncsu.edu) <http://www.ece.ncsu.edu>

<sup>¶</sup>Department of Electrical and Computer Engineering, University of Illinois Urbana Champaign  
[cangel@decwa.ece.uiuc.edu](mailto:cangel@decwa.ece.uiuc.edu)

## Abstract

Mixed-signal simulation for communication systems must incorporate comprehensive modelling of the multi-physics phenomena that impact design functionality and performance. Future, multi-functional RF systems involve all electronic design technologies ranging from high-level logic synthesis, to custom digital, analogue, RF and microwave design. Furthermore, optoelectronic and micro-mechanical devices and functional blocks will be incorporated. In this paper we describe an approach to delivering a revolutionary modelling tool that implements new modelling and simulation abstractions, fast linear and nonlinear solvers, full-wave EM modelling for on-chip parasitics and integrated RF/microwave circuit design modelling, digital and analogue behavioural modelling, optoelectronic modelling and advanced electrothermal modelling.

## I Introduction

Mixed signal circuits and systems, and especially RFICs, necessitate a re-evaluation the underlying approaches used for modelling, simulation and design for chip- and package-level signal integrity (Table 1). Often the need for re-evaluation does not appear until it is too late to properly prepare for it. The next level of complexity in signal integrity engineering will come about when current quasi-static assumptions start becoming unrealistic, and some degree of full-wave modelling and frequency-dependent simulation becomes important. Preparing for this next level of complexity will take more effort than all the effort spent in the past combined. For example, consider the following electromagnetic effects and the frequency ranges at which they start becoming noticeable:

- *Dielectric losses.* Depending on the substrate material and wire cross-section, dielectric losses start becoming as important as conductor losses in the accurate prediction of multi GHz signal attenuation and distortion. Furthermore, this loss is highly frequency dependent.
- *TE, TM and substrate Modes.* Planar transmission lines are known to suffer from power leakage due to the excitation of non-TEM modes that propagate in the substrate with partial support by adjacent metalization. The generation of such substrate modes becomes an issue of concern for frequencies above 10 GHz. Their accurate modelling is critical for the electromagnetic analysis of global controlled-impedance interconnects used for the propagation of multi-GHz bandwidth signals. The propagation characteristics of these substrate modes exhibit strong frequency dependence that cannot be modelled by the quasi-static tools used for quasi-TEM analysis.

Furthermore, their excitation is strongly dependent on adjacent interconnect topography, thus requiring full three-dimensional electromagnetic analysis. [1].

- *Dispersion.* Other dispersive effects (due to mixed dielectrics, for example) become important above around 14 GHz [1].
- *Proximity Effect.* Current crowding in the return path leading to frequency-dependant RCL(f) starts becoming significant on-chip below 1 GHz. [1].

Future communications systems are expected to contain links operating beyond 5 Gbps, with spectrums containing significant energy beyond 15 GHz. Unfortunately, this problem is not just a modelling problem. The complexity of future communication chips and systems is increasing at the same time. Future methodologies require not just full-wave modelling and full-fidelity simulation but a means to cope with the complexity of a million or more nets on a chip. A very difficult issue will be design convergence – ensuring that each stage of planning is sufficiently accurate to ensure that interconnect delay, noise and RF performance is managed with sufficient accuracy without excessive design margin.

**Table 1.** New issues arise (often by surprise) at every generation. It is now necessary to prepare for the 10 GHz transition.

10 MHz	100MHz	1 GHz	10 GHz
<b>Main Modelling Issues</b>			
Lumped C	RC delay	RCL	Full wave - RCL(f) - dispersion - non quasi-TEM - dielectric losses
<b>Main Simulation Approach/Issues</b>			
Simple calculation	RC delay estimation - Pade approximation - Elmore delay	Full Spice Modelling	Distributed electromagnetic effects/dispersion/retardation
<b>Design Methodology Approach/Issues</b>			
Spreadsheet	Timing management - Extraction	Planning tools - predictors	Design Convergence difficult - Appropriate abstractions - Hierarchical convergence - SoC issues

We believe that on-chip analysis will require distributed electromagnetic analysis to provide the rigor needed. This contrasts with the current quasi-static approach that leads to independent inductance, capacitance and resistance extraction. Such a methodology becomes grossly inaccurate when applied to three-dimensional interconnect structures that are expected to support multi-GHz bandwidth signals. The reason for this is that, even with electromagnetic retardation assumed negligible (as is typically the case for on-chip interconnects), interconnect capacitance extraction assumes that the electric potential remains essentially constant over the entire net for which the self and mutual capacitance with other nets needs to be obtained. Similarly, for interconnect inductance extraction, the assumption is made that interconnect currents flow in physically connected conducting loops, thus neglecting the effect of capacitive coupling on the definition of the return path.

So that the above effects can be properly accounted for we believe 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 modelling has become increasingly complicated in large part because physical behaviour 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.

## **II The Global Modelling Concept**

From the above discussion it is clear that critical to the global modelling of multi-functional, hybrid, mixed-signal systems is a transient, non-linear simulation environment capable of multi-physics abstraction. We believe that the fundamental problem in modelling mixed signal systems using state-of-the-art EDA tools, is that the circuit abstraction that is embedded in Spice (and similar circuit simulation programs) is too restrictive and is based on old technology. We believe that any effort that builds on this framework is destined to lead to further problems and advances to overcoming inherent limitations. The problem is not specific to the computer program Spice, but Spice represents an encapsulation of the computer aided circuit analysis theory of the late 1960s which was largely pioneered by IBM (indeed IBM embodied the concepts of "Spice" before Spice was developed.) We contend that in Spice (and commercial derivatives) circuit analysis uses thirty-year-old technology. 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 modelling." We have been addressing this by developing a Global Modelling Concept and developing a commercial-strength simulator called Transim.

### **A. Comprehensive and Fast Methods for Multi-Physics Modelling and Simulation**

The aforementioned needs for electrical modelling 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

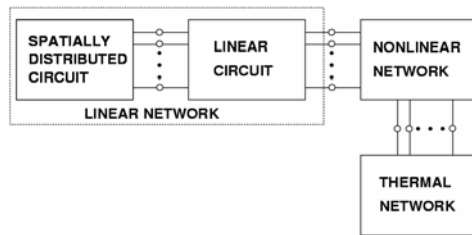
needs to be in a format suitable for network-oriented system-level simulation both in the time and the frequency domain.

Parallel to such advances in fast electromagnetic solvers, advances are needed in the way thermal and mechanical modelling of MEMS is effected in a seamless manner in the aforementioned network-oriented system-level simulation tools for the purposes of global modelling. The same holds true for the optoelectronic devices and components that are expected to be present in future, performance-driven hybrid designs.

For the case of optoelectronic modelling and simulation, the primary handicap of existing CAD tools is their incompatibility with the Spice simulation environment. This prevents the quite sophisticated models for optical wave propagation and spatial processing to be used together with optical sources and detectors as well as the electrical models for the driver/receiver electronics and associated interconnect-induced parasitics for an accurate and meaningful functionality and noise performance analysis and subsequent design optimisation. To overcome this obstacle novel interpretations of optoelectronic elements and devices models in terms of state representations compatible with a network-oriented circuit simulator are needed.

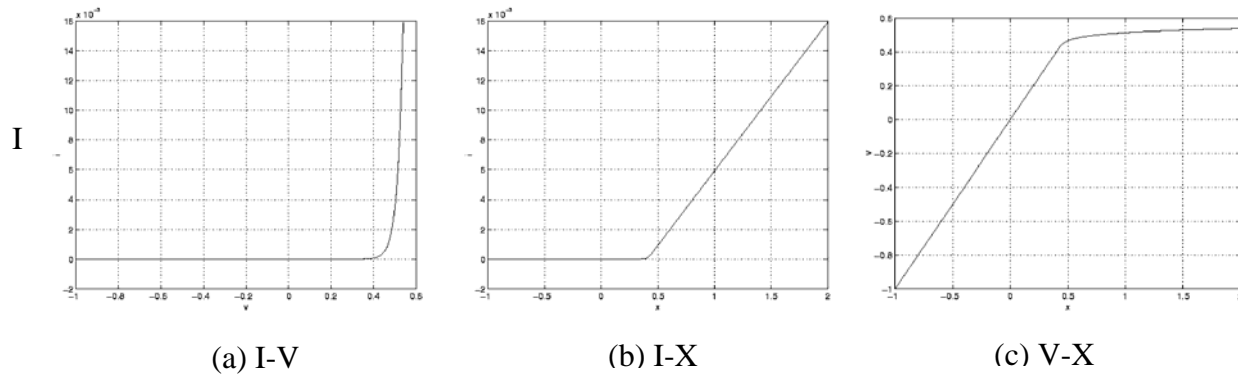
### B. Simulation Approach

Our circuit simulation approach, implemented in the program Transim, realises the first new circuit simulation approach since Spice-like analyses were developed. Concurrent electromagnetic, circuit and thermal analysis are supported. Fig. 2 illustrates the model integration concept. 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 modelled independently and connected to form a thermal circuit.



**Figure 2:** Partition of a system into spatially distributed and lumped linear circuit, nonlinear network, and thermal parts.

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. For example, Fig. 3 illustrates the improvement in the regular nonlinear behaviour of a diode model when using the state variable approach.



**Figure 3:** Parameterisation of the strongly nonlinear characteristics of a diode as shown in (a). The relations plotted in (b) and (c) are well behaved and implement the same I and V relationship.

Virtually any physical model can be implemented in the new modelling 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.

### C. Thermal Modelling

The global modelling approach includes thermal modelling as described the paper in this conference presented by Dr. William Batty and described in References [4–10]. The fundamentally new approach to thermal modelling converts a nonlinear volumetric thermal problem into a linear boundary thermal problem that is several orders of magnitude faster to solve compared to using FEM based schemes. Each boundary of a sub-volume has a thermal terminal to which other sub-volumes and active devices are connected. The technique involves transformed time and transformed temperature concepts but there is no loss of accuracy as the necessary conversions to relate to the real world are handled inside the global modelling simulator. The thermal model is sufficiently fast and robust that electrical and thermal simulations can be undertaken concurrently. The thermal models can be used in both transient and steady state simulations.

### **III 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 modelling 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 modelling strategies. Another technique identifies full-wave interconnect macromodels that can then be used in Transim, instead of full fidelity models.

Reduced order modelling of nonlinear circuitry is essential if whole chips and mixed signal systems are to be modelled. 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 were instrumental in the founding of the IBIS consortium and Steer was the founding librarian of the society. One of the most significant developments (supported by DARPA) was the development of the Spice2IBIS program that automatically develops the IBIS model of a driver or receiver given its Spice netlist (and many hours of Spice simulations). 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.

### **IV Comprehensive and Fast Interconnect Parasitics Extraction**

#### ***A. Parasitic Extraction and Full-Wave Modelling***

We are developing a new class of electromagnetic solvers capable of tackling the complexity of both on-chip and global interconnect networks with the rigor required to capture multi-GHz effects. The new generation of tools will: 1) Extract broadband interconnect transmission performance, and inductive and capacitive coupling properties for very wide buses, accounting for the complex, non-TEM, three-dimensional environment necessitated by the ultra-high wiring densities of future designs. 2) Account rigorously and efficiently for the frequency dependence of interconnect inductance and resistance due to skin effect loss. 3) Account for the dispersion and attenuation introduced by substrate loss. 4) Account for the common-impedance coupling and the frequency dependence of the return path. 5) Account for the frequency dependence of propagation speed and characteristic impedance due to inhomogeneous, high-k, and even anisotropic dielectrics. 6) Model accurately and efficiently semiconductor substrate coupling between analogue and digital blocks integrated on the same semiconductor substrate. 7) Facilitate the design of multi-layered implementations of integrated RF passives with the electromagnetic accuracy needed to meet the narrowband and isolation requirements characteristic of modern wireless applications. Our essential claim is that reduced order models of interconnect parasitics and electromagnetic responses of integrated passives can be developed directly and efficiently through robust, passive model order reduction processes that are embedded in the EM extractors.

#### ***B. Parasitic Extraction and Full-Wave Modelling***

Over the past five years, significant attention has been placed on the advancement of interconnect parasitics modelling tools that exhibit the computational efficiency needed to tackle the complexity of the interconnect network. The results of these efforts include both Fast Multipole Methods (FMM)-based and Pre-Corrected FFT-based capacitance and inductance extractors that have made

possible capacitance and inductance parasitic extraction for interconnects of high-complexity (e.g., [11–16]). However, such a modelling approach is grossly inaccurate when applied to three-dimensional interconnect structures that are expected to support multi-GHz bandwidth signals. The reason for this is that, even with electromagnetic retardation assumed negligible (as is typically the case for on-chip interconnects), interconnect capacitance extraction assumes that the electric potential remains essentially constant over the entire net for which the self and mutual capacitance with other nets needs to be obtained. Similarly, for interconnect inductance extraction, the assumption is made that interconnect currents flow in physically connected conducting loops, thus neglecting the effect of capacitive coupling on the definition of the return path. While for the case of a quasi-TEM analysis of two-dimensional interconnect structures (i.e. interconnects that exhibit strong geometric uniformity along the direction of signal flow) the separation of interconnect parasitic extraction into an electrostatic and a magneto-quasi-static problem is theoretically justified [2], this separation fails for truly three-dimensional structures that are strongly non-TEM. Effects such as current crowding and common-impedance coupling, which have been shown to be of critical importance to the accurate prediction of interconnect-induced delay and crosstalk [3], cannot be quantified properly without the simultaneous modelling of inductive and capacitive coupling. Furthermore, for the case of global interconnects, full-wave electromagnetic modelling is required to account properly for the impact of the chip-to-package discontinuities in signal reflection, radiation loss, and transmission bandwidth reduction. Finally, for the case of integrated passives, in addition to current crowding and its impact on loss, electromagnetic retardation needs to be taken into account for accurate prediction of the useful bandwidth of the component.

The aforementioned shortcomings of existing fast interconnect parasitics extractors are further compounded by the fact that these extractors are, by construction, limited to the modelling of interconnect structures embedded in layered, homogenous, isotropic substrates. Such a limitation is undesirable and is becoming a serious bottleneck as inhomogeneous and anisotropic substrates utilising high-k dielectrics are being considered as means for both improving signal integrity and facilitating the realisation of on-chip integrated passives. Furthermore, for the case of integrated passives, accurate modelling of semiconductor substrate-induced loss is essential for accurate prediction of the quality factor of the device. Finally, as frequency bandwidths extend beyond 10 GHz, the excitation of substrate modes becomes possible. Such modes exhibit strong frequency dependence, act as leakage mechanisms for the interconnect signals, and can be accounted for only through full-wave analysis.

The proposed electromagnetic modelling capability addresses the aforementioned shortcomings through a multiple-task effort that is aimed at the development of a versatile interconnect modelling and simulation capability, capable of delivering the necessary electromagnetic accuracy with the expediency needed for CAD-driven design.

## **V Discussion**

Transim [17–20] implements, or will, 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. It overcomes the fundamental shortcoming of Spice and similar circuit simulation programs; namely, the restrictive model abstraction based on electric circuit elements and associated state variables. This is “old technology” and, considering the rapid growth toward multi-physics (“everything-on-chip”) integrated designs, any effort that builds on this framework is destined to lead to further problems and advances to overcoming inherent limitations. Transim’s approach to model

abstraction provides the versatility needed to tackle the complexity and hybrid nature of such revolutionary integrated systems.

## VI Conclusion

Transim has a GNU public license that facilitates its linkage with the EDA environment of interested industry. Through close collaboration with our industry partners, a paradigm for this linkage will be made. The initial rollout of Transim was supported by DARPA and has been adopted by two EDA companies. It is now supported by an SBIR, an ARO STAS, and private venture funding. The entire team on this proposal has a track record of successful technology transition. For example, Steer and Franzon developed the Spice2IBIS converter that is used throughout the Signal Integrity community to develop behavioural models of digital receivers and off-chip drivers.

## VII References

- [1] A. Deutsch, "Electrical characteristics of interconnections for high-performance systems", Proc. IEEE, Vol. 86, No. 2, February 1998, pp315-355.
- [2] R. Collin, *Field Theory of Guided Waves*, IEEE Press.
- [3] A. Deutsch et al, "Frequency-dependent crosstalk simulation for on-chip interconnects," *IEEE Trans. Advanced Packaging*, vol. 22, no. 3, pp. 292-308, Aug. 1999.
- [4] W. Batty, A. J. Panks, R. G. Johnson and C. M. Snowden, 'Electro-thermal modelling and measurement for spatial power combining at millimeter wavelengths,' *IEEE Trans. Microwave Theory Tech.*, vol. 47, no. 12, pp. 2574-2585, 1999.
- [5] W. Batty, A. J. Panks, R. G. Johnson and C. M. Snowden, 'Electro-thermal modelling of monolithic and hybrid microwave and millimeter wave IC's,' *VLSI Design*, vol. 10, no. 4, pp. 355-389, 2000.
- [6] W. Batty and C. M. Snowden, 'Electro-thermal device and circuit simulation with thermal non linearity due to temperature dependent diffusivity,' *Electron. Lett.*, vol. 36, no. 23, pp. 1966-1968, 2000.
- [7] W. Batty, C. E. Christoffersen, S. David, A. J. Panks, R. G. Johnson and C. M. Snowden, 'Steady-state and transient electro-thermal simulation of power devices and circuits based on a fully physical thermal model,' Proc. 6<sup>th</sup> THERMINIC Workshop, pp. 125--130, Budapest, September 2000.
- [8] W. Batty, C. E. Christoffersen, S. David, A. J. Panks, R. G. Johnson, C. M. Snowden and M. B. Steer, 'Predictive microwave device design by coupled electro-thermal simulation based on a fully physical thermal model,' Proc. 8<sup>th</sup> IEEE Internat. Symp. on High Performance Electron Devices of Microwave and Optoelectronic Applications (EDMO 2000), pp. 111-116, 2000.
- [9] W. Batty, C. E. Christoffersen, S. David, A. J. Panks, R. G. Johnson, C. M. Snowden and M. B. Steer, 'Fully physical, time-dependent compact thermal modelling of complex non linear 3-dimensional systems for device and circuit level electro-thermal CAD,' Proc. 17<sup>th</sup> Annual IEEE Semiconductor Thermal Measurement and Management Symp., San Jose, March 2001 (In Press).
- [10] W. Batty, C. E. Christoffersen, A. J. Panks, S. David, C. M. Snowden and M. B. Steer, 'Electro-Thermal CAD of Power Devices and Circuits with Fully Physical Time-Dependent

Compact Thermal Modelling of Complex 3-D Systems,' IEEE Trans. Comp. Packag. Technol., Sept. 2001.

- [11] K. Nabors and J. White, "FastCap: A multipole accelerated 3D capacitance extraction program, *IEEE Trans. Computer Aided design*, vol. 10, pp. 1447—1459, 1991.
- [12] K. Nabors et al, "Preconditioned, adaptive, multipole-accelerated iterative methods for three-dimensional first-kind integral equations of potential theory," *SIAM J. Scientific and Statistical Computing*, vol. 15, no. 3, pp. 713—735, 1994.
- [13] J. R. Phillips and J.K. White, "Efficient capacitance extraction of 3D structures using generalized pre-corrected FFT methods," *Proc. IEEE 3<sup>rd</sup> Topical Meeting on Electrical Performance of Electronic Packaging*, Nov. 1994.
- [14] J. R. Phillips and J. White, "A precorrected-FFT method for capacitance extraction of complicated 3-D structures," *International Conference on Computer Aided Design*, San Jose, CA, Nov. 1994.
- [15] M. Kammon, *FastHenry User's Guide* (Version 3.0), Nov. 1996.
- [16] M. Bachtold et al, "A system for full-chip and critical net parasitic extraction for ULSI interconnects using a fast 3-D field solver," *IEEE Trans. Computer-Aided Design*, vol. 19, pp. 325-338, Mar. 2000.
- [17] C. E. Christoffersen and M. B. Steer, "State-variable-based transient circuit simulation using wavelets," *IEEE Microwave and Guided Waves Letters*, Vol. 11. 2001, In Press.
- [18] C. E. Christoffersen, U. A. Mughal, and M. B. Steer, "Object oriented microwave circuit simulation," *Int. J. on RF and Microwave Computer Aided Engineering*, Vol. 10, Issue 3, 2000, pp. 164– 182.
- [19] 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, Sept., 1999, pp. 376–384.
- [20] M. B. Steer, J. F. Harvey, J. W. Mink, M. N. Abdulla, C. E. Christoffersen, H. M. Gutierrez, P. L. Heron, C. W. Hicks, A. I. Khalil, U. A. Mughal, S. Nakazawa, T. W. Nuteson, J. Patwardhan, S. G. Skaggs, M. A. Summers, S. Wang, and A. B. Yakovlev, "Global modeling of spatially distributed microwave and millimeter-wave systems," *IEEE Trans. Microwave Theory Techniques*, June 1999, pp. 830–839.