SIMULATION OPTIMIZATION SYSTEMS
Research Laboratory

ELECTROMAGNETIC DESIGN OF MICROWAVE CIRCUITS INTEGRATED WITH SPICE DEVICE MODELS

J.W. Bandler, R.M. Biernacki, Q. Cai,
S.H. Chen and P.A. Grobelny

SOS-95-3-R

February 1995

McMASTER UNIVERSITY
Hamilton, Canada L8S 4L7
Department of Electrical and Computer Engineering

ELECTROMAGNETIC DESIGN OF MICROWAVE CIRCUITS INTEGRATED WITH SPICE DEVICE MODELS

J.W. Bandler, R.M. Biernacki, Q. Cai,
S.H. Chen and P.A. Grobelny

SOS-95-3-R

February 1995

ELECTROMAGNETIC DESIGN OF MICROWAVE CIRCUITS INTEGRATED WITH SPICE DEVICE MODELS

J.W. Bandler, R.M. Biernacki, Q. Cai, S.H. Chen and P.A. Grobelny

Optimization Systems Associates Inc.
P.O. Box 8083, Dundas, Ontario, Canada L9H 5E7

Tel +1 905 628 8228
Fax +1 905 628 8225

Topic Reference Number: 12

SUMMARY

Introduction

Microwave and millimeter-wave circuit designers strive to employ in their design process the best and most accurate simulation techniques available. Recent advances in electromagnetic (EM) field-theory based simulation, in particular the improved efficiency of the algorithms combined with ever increasing computing speed, cause enormous interest in EM design.

EM simulators need to be integrated into circuit optimization in order to fulfil the requirements of automated design. This has already been accomplished for passive microwave and millimeter-wave filters, with either EM simulation in the frequency domain [1, 2] or TLM simulation in the time domain [3]. Recently, this has been extended to nonlinear circuits simulated using the harmonic balance analysis with the passive microstrip structures simulated by Sonnet's ***em*** [4].

EM simulation of passive components and device simulation are of disjoint nature. It makes it necessary to combine the results of both simulations at the circuit level by means of circuit-theoretic analysis. Moreover, both simulations could be carried out by independent simulators. The process of invoking such independent simulators and combining their results must be automated to allow for design optimization. This work breaks the ground for software architecture suitable to address these problems. We focus our attention on employing SPICE for device simulation, ***em*** for simulation of microstrip subcircuits, and OSA90/hope for circuit level simulation, optimization and statistical processing.

Capturing SPICE Device Models

Public domain SPICE by itself does not provide any means for optimization or for statistical representation. Incorporating the results of EM simulations of passive subcircuits into SPICE requires equivalent circuit representation and is not available in an automated fashion for optimization. Rigid structure of commercial versions of SPICE allows for limited optimization only.

SPICE device models are highly regarded by the microwave community. In our work SPICE is interfaced to OSA90/hope as one of many child processes, as shown in Fig. 1. OSA's Datapipe technology based on UNIX pipes is utilized for that purpose. The interface allows for OSA90/hope to drive SPICE in an automated manner, with the SPICE input data determined in OSA90/hope and the SPICE results returned to OSA90/hope.

SPICE is invoked to simulate the device only. The SPICE output is returned to OSA90/hope and further postprocessed. This is achieved using the expression processing capabilities of OSA90/hope, for example simulated node voltages are used to determine the S parameters of the device. In fact, two SPICE simulations are carried out to determine the parameters of a 2-port network.

Statistical SPICE Device Modeling

Statistical modeling from a sample of GaAs MESFET measurement data is performed using the SPICE model. For each device outcome parameter extraction is driven by OSA90/hope's ℓ_1 optimizer with the SPICE MESFET model captured as described in the previous section. The model responses are compared by OSA90/hope against the measured data. This leads to a sample of individually extracted device models. The model statistics including the mean values, standard deviations and the correlation matrix are obtained by postprocessing this sample of models.

Yield Optimization with SPICE device Models and EM Simulation of Passive Components

Using the aforementioned statistical MESFET model we designed a small-signal amplifier shown in Fig. 2. Circuit-level simulation and optimization is performed by OSA90/hope with the SPICE model captured for any specific parameter values needed either in statistical analysis or in optimization. Model statistics are fully described within and all the Monte Carlo outcomes are generated by OSA90/hope. All L, C and R components in Fig. 2 are chosen as design variables. A total of 28 statistical parameters is considered.

EM design integrated with SPICE device simulation is illustrated by yield optimization of a small-signal amplifier (Fig. 3). The microstrip components are simulated by ***em*** and the MESFET model is captured from SPICE (see Fig. 1). The circuit-level simulation and yield optimization are carried out by OSA90/hope. We consider 8 geometrical parameters as design variables. Before optimization yield is 43%. After optimization it is increased to 74%

Conclusions

We have presented a new approach to automated nominal and statistical device modeling and design of microwave circuits. For the first time accurate EM simulation of passive microstrip structures has been integrated with SPICE device modeling and powerful circuit-level design optimization. This work breaks the ground for software architecture suitable to address integration of physical and physics-based device simulators in a similar fashion. Rapid development of such simulators promises to bring the accuracy of device simulation and modeling to a level similar to that of EM simulation of passive structures.

References

[1] J.W. Bandler, R.M. Biernacki, S.H. Chen, D.G. Swanson, Jr., and S. Ye, "Microstrip filter design using direct EM field simulation," *IEEE Trans. Microwave Theory Tech.*, vol. 42, 1994, pp. 1353-1359.

[2] J.W. Bandler, R.M. Biernacki, S.H. Chen, P.A. Grobelny and S. Ye, "Yield-driven electromagnetic optimization via multilevel multidimensional models," *IEEE Trans. Microwave Theory Tech.*, vol. 41, 1993, pp. 2269-2278.

[3] P.P.M. So, W.J.R. Hoefer, J.W. Bandler, R.M. Biernacki and S.H. Chen, "Hybrid frequency/time domain field theory based CAD of microwave circuit," *Proc. 23rd European Microwave Conf.* (Madrid, Spain), 1993, pp. 218-219.

[4] J.W. Bandler, R.M. Biernacki, Q. Cai, S.H. Chen and P.A. Grobelny, "Integrated harmonic balance and electromagnetic optimization with Geometry Capture," *IEEE MTT-S Int. Microwave Symp.* (Orlando, FL), 1995.

[5] R. Griffith, E. Chiprout, Q.J. Zhang and M. Nakhla, "A CAD framework for simulation and optimization of high-speed VLSI interconnections", *IEEE Trans. Circuits and Systems*, vol. 39, 1992, pp. 893-906.

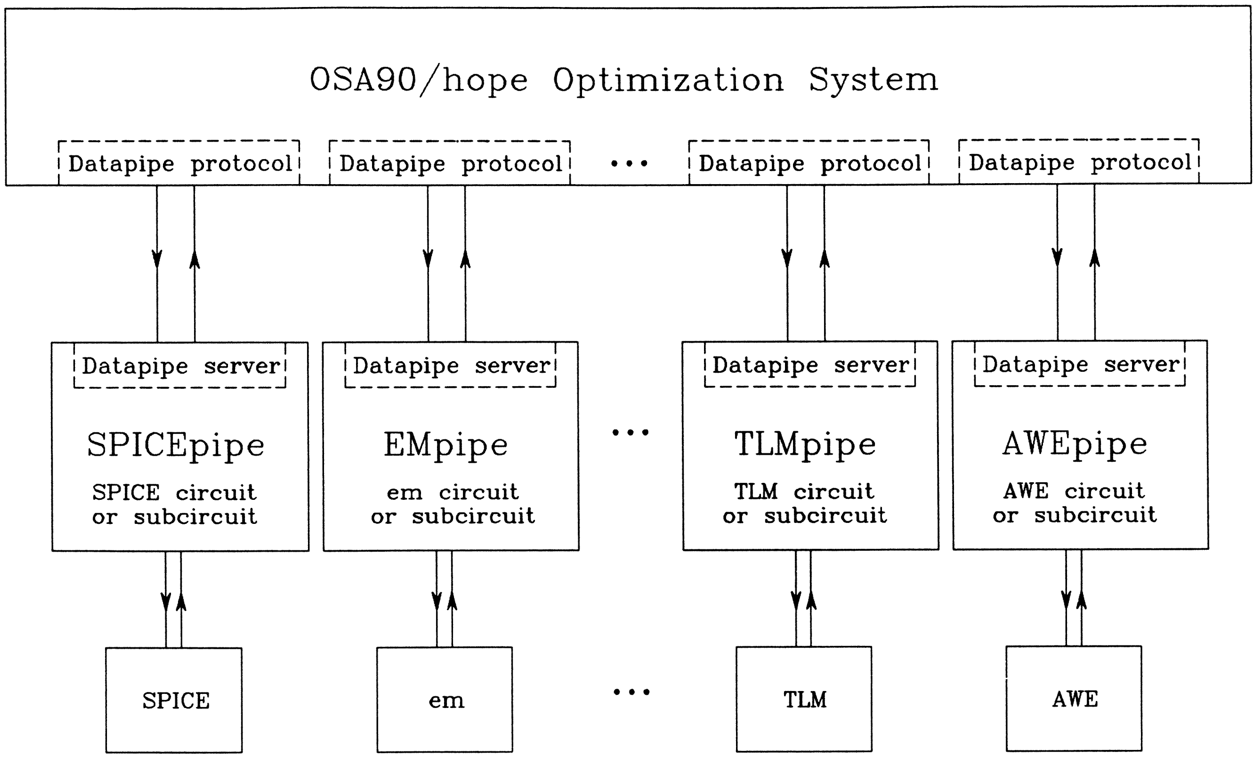

Fig. 1. Datapipe interface between OSA90/hope and several external simulators including SPICE, *em*, TLM [3] and AWE [5].

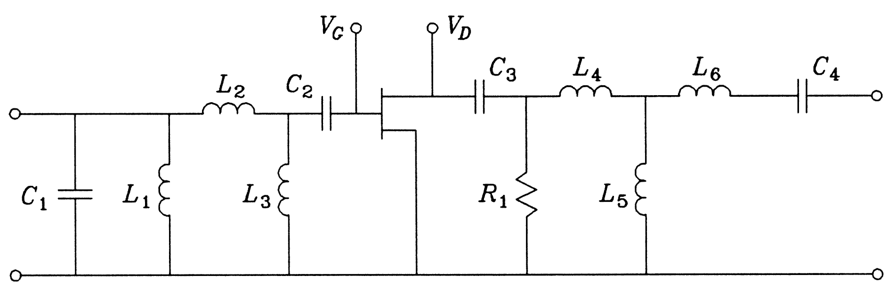

Fig. 2. The small-signal amplifier.

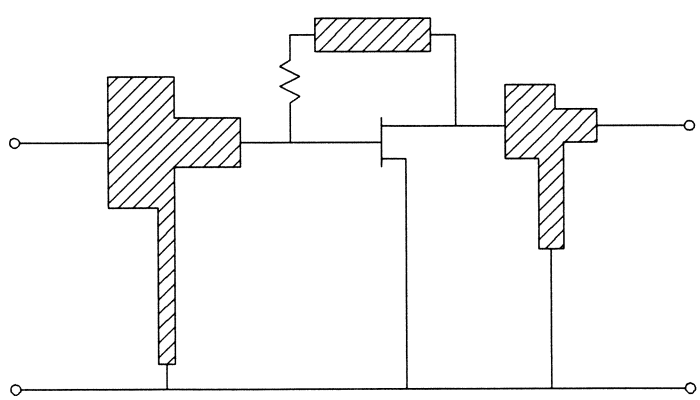

Fig. 3. The broadband small-signal amplifier with microstrip components.