A Unified Technique for PCB/MCM Design by Combining Electromagnetic Field Analysis with Circuit Simulator

Hideaki Kimura Norihito Iyenaga
NTT Access Network Systems Laboratories

Abstract

This paper proposed the unified design technique which combines electromagnetic field analysis [FDTD technique] with circuit simulator [HSPICE]. Proposed technique can analyze the integrated circuits [ICs], multi-chip-module [MCM], and printed circuit board [PCB] design in high-efficiency and high-accuracy including the rounding noise throughout the substrate. Furthermore, this technique can not only analyze the small signal operation but also large signal operation.

I. INTRODUCTION

Recently, the growing demand for multimedia applications requires large-scale integration of highly-accurate analog circuits with many high-speed digital circuit gates. In these mixed-signal ICs, the noise created by the digital circuits passes to the on-chip analog circuits, affecting analog circuit performance. The noise is assumed to be transmitted through various paths. In general, to minimize the coupling of switching noise, power supply lines are separately connected to the digital and analog circuits. On the other hand, in module design, miniaturization of modules is very effective for cost reduction as same as ICs. But miniaturization and high-integration introduce the poor characteristics of modules because of signal coupling between input and output. Therefore, module design technique considering signal coupling and noise passing through the module substrate is important and necessary as same as ICs.

In this paper, we proposed the high-efficiency and high-accuracy design technique for ICs, PCB, and MCM, which can consider all of the electromagnetic field compatibility [EMC] and can analyze both in small signal operation and large signal operation analysis.

II. NEW DESIGN TECHNIQUE

Electromagnetic field analysis can consider the phenomenon originated in physical structure and materials parameters in mounting substrate under consideration, which provides a clear picture of module performance is important and effective way to reduce costs and turn around time. But electromagnetic field analysis, which is called full-wave analysis, needs the high-performance for the computer, because it uses all of electromagnetic filed components. So we propose the effective way using powerful electromagnetic field analysis. Figure 1 shows the concept of proposed design technique considering ICs characteristics, the signal coupling and rounding noise of modules. This technique adopt the finite-difference time-domain [FDTD] [1] method as electromagnetic field analysis for module substrate design and HSPICE as circuit simulator for ICs design.

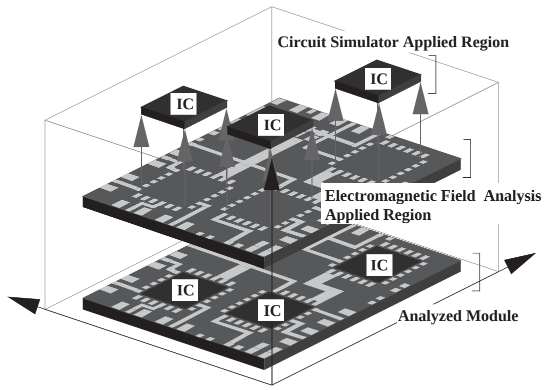


Figure 1 Concept of Proposed Technique

The FDTD method represents true computer simulation that shows field evolution directly in the time domain, and various lumped device elements can be easily modeled and incorporated in the analyzed structure yielding flexibility in representing structures and functions of the circuits. The circuit simulator HSPICE can evaluate the ICs characteristics in frequency domain and time domain efficiently and accurately. Figure 2 shows the procedure of large signal operation analysis, respectively. For example, calculation procedure in small signal operation is as ; (a)first, separate the ICs part and module part, (b)second, apply the FDTD to the module substrate part and the HSPICE to ICs part, (c)third, transform the time domain data obtained by FDTD to the frequency domain data by using the FFT, (d)last, calculate the total performance by combining the frequency domain data of the mounting substrate with the data obtained by HSPICE. As explained, in small signal operation analysis, the characteristics of the module considering the signal coupling and the noise passing the mounting substrate and ICs performance can be obtained at the stage of design. On the other hand, in large signal operation analysis, the proposed design technique can calculate the FDTD and HSPICE alternately as synchronizing the time step of FDTD and HSPICE as shown in Fig.2.

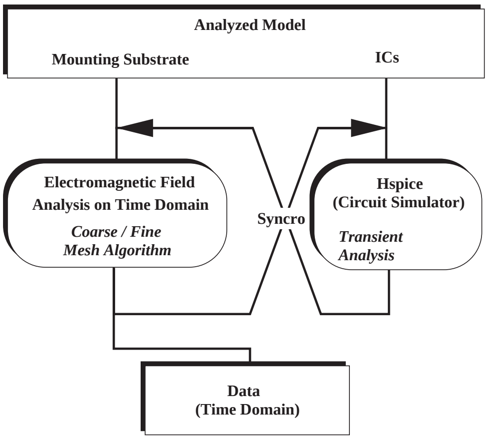


Figure 2 Large signal operation analysis

III. VALIDITY OF PROPOSED TECHNIQUE

For examining the validity of this technique, we applied the proposed technique to optical transmitter/receiver module design. Figure 3 shows the optical transceiver module.

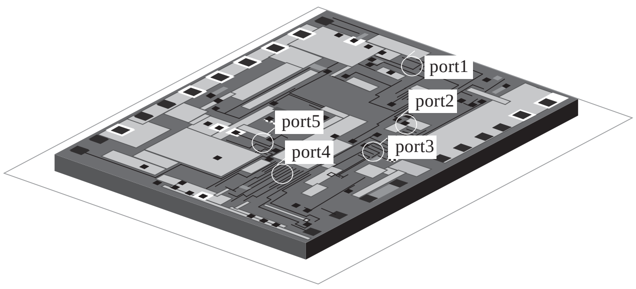
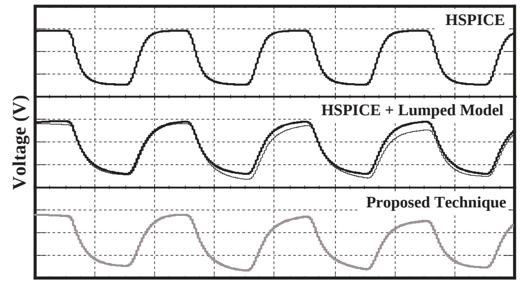
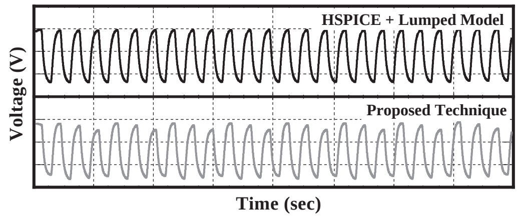


Figure 3 Optical transmitter / receiver module

Figure 4 shows the wave forms at port 1 shown in Fig.3. As shown in Fig.4, the proposed technique can consider the low frequency signal rounding which can not be estimated in the combination model HSPICE with lumped model. Table 1 shows the comparison simulation results calculated by proposed technique with the experimental results, but we assumed the isolation characteristics relates the minimum optical received power with S/N ratio. As shown in Table.1, simulation results show good agreement with the experimental results. These results show the validity of proposed technique.



(a) Expanded Wave Form



(b) Wave Form

Figure 4 Wave forms

IV. CONCLUSIONS

In this paper, we proposed the unified design technique combining electromagnetic field analysis with circuit simulator, which can analyze in small signal and large signal operation. We expect proposed design technique considering all of EMC phenomenon and the ICs performance will be very important and necessary in ICs, PCB and MCM design.

Mounting Substrate Type	Minimum Optical Received Power BER=1×10^{-8}@156Mbps	
	Simulation (Proposed Technique)	Experimental Results
Sub1	-32.7dBm	-31.6dBm
Sub2	-31.5dBm	-31.2dBm
Sub3	-30.1dBm	-29.5dBm

Table 1 Comparison simulation with experimental results

REFERENCES

[1]T.Shibata and H.Kimura,'Computer-aided engineering for microwave and millimeter-wave circuits using the FD-TD technique of field simulations,' in Int.J.Microwave MM-Wave Computer.-Aided Eng., vol.3,pp.238-250, July 1993.