Simulation Method for Quasi-Static Solver to Effectively Model Parasitic Components Between Package and PCB

(Article begins on next page)

Simulation method for Quasi-static solver to effectively model parasitic components between Package and PCB

Silvia Simone[1], Fabio Pareschi[1], Davide Lena[2], and Gianluca Setti[3]
[1]DET, Politecnico of Torino, corso Duca degli Abruzzi 24, 10129 Torino, Italy.
[2]STMicroelectronics s.r.l., Torino, Italy.
[3]CEMSE, King Abdullah University of Science and Technology (KAUST), Saudi Arabia.
Email: {silvia_simone, fabio.pareschi}@polito.it, davide.lena@st.com, gianluca.setti@kaust.edu.sa.

***Abstract*—A methodology to simulate Package and PCB with a quasi-static solver is developed. The widespread cascade method, where Package and PCB are first simulated standalone to reduce the required computational effort and then recombined, is improved to consider electromagnetic interactions between the two systems. The proposed approach provides results very similar to that achieved with the full system simulation, but with a computational cost that is very similar to that of the standard cascade method.**

***Index Terms*—quasi-static solver, electromagnetic simulation, Q3D, Package, Printed Circuit Board (PCB), parasitic components, power integrity.**

I. Introduction

Nowadays, considering the effect of parasitic components in the early design phase is fundamental for a thorough analysis that guarantees compliance with the electromagnetic compatibility standard regulation, but also to reduce the risk of iteration of the production process, ensuring that all constraints are met. In particular for power integrity analysis, knowing the values of parasitic elements in a very accurate way is essential for design considerations. For that reason, electromagnetic simulation of Printed Circuit board (PCB) and Package (PKG) of the device under test must be performed.

For design of converters and power electronic equipment a quasi-static tool is adequate when the structure is not electrically large, therefore when the dimension of the structure is approximately lower than $\lambda/10$, where λ is the signal wavelength, to precisely model the parasitic components for the co-simulation with the electrical circuit [1]. However, performing a complete simulation of PCB and PKG can be time and memory consuming. Especially if the system consists of components of different dimensions subdividing it into sections with similar size can be useful.

The current industry adopted method is to model PCB and PKG separately due to the limited computational resources [2]. Some investigations on the difference between considering the full model and the cascade of the separate objects are also considered [3], instead a technique to extract only the value of interconnections is developed [4]. For high frequencies applications, mainly for signal integrity analysis, some techniques have also been studied [5]–[8]. However, all aforementioned methodologies involve full wave solver, or results are compared with a hybrid solver, cascading the different analyzed elements [9].

In this work the quasi-static solver of Ansys, Q3D, is used. It can be successfully employed when the frequencies involved and the dimensions of the device satisfy the explained solver validity condition. A typical application is a DC/DC converter with a switching frequency in the range of kHz-MHz, where lumped elements of parasitic components can be used for power integrity analysis. With the adopted tool, a methodology to reduce the computational effort of electromagnetic simulations is explained, obtaining more accurate values respect to simply cascading the two separately simulated elements, that is the easiest methodology to adopt but which neglects the electromagnetic contribution due to the interaction between the two systems. Instead, with the proposed solution the accuracy of the obtained results is comparable to the simulation of the full system, nevertheless maintaining a significant reduction in the employed resources. In Section II the method used for analyzing the PCB and PKG with a quasi-static solver is explained, and it is applied to a practical case showing the results in Section III. Finally, the conclusion is drawn.

II. Adopted methodology

When considering a structure consisting of PCB and PKG, most accurate results are obtained with the simulation of the whole system (i.e., PCB + PKG) represented in Figure 1.

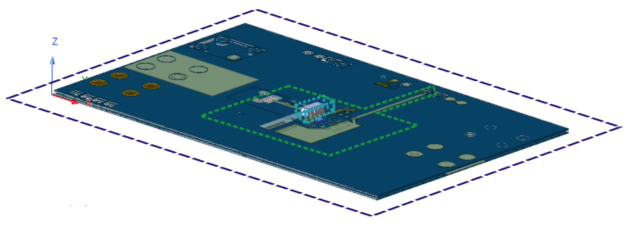

Fig. 1. Complete model of PCB highlighted in dark blue, signal traces in green and PKG in light blue, used for the simulation of whole system to obtain parasitic components considering the interaction between the structures.

979-8-3503-5123-1/24/$31.00 ©2024 IEEE

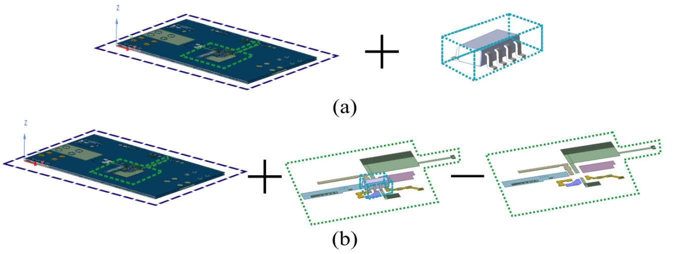

Fig. 2. Cascade method of T-parameters (a): simulation of PCB and simulation of PKG; proposed method (b): simulation of PCB, simulation of PKG+signal traces connected to it, simulation of signal traces only.

However, due to the limited computational resources, the most widespread technique consists in simulating the PCB and the PKG separately, as represented in Figure 2(a), and subsequently cascading the two matrices of S-parameters obtained from the electromagnetic simulations, conventionally using ABCD or T-parameters.

In this paper, we propose first to simulate the PCB alone, but instead of cascading results with that obtained from the simulation of the PKG alone, a simulation of the system composed of the PKG plus the PCB traces connected to it is performed and results are combined. Since in this way the PCB traces are present twice, to eliminate the contribution a simulation of only the traces is performed and subtracted from the previous results. This procedure is schematized in Figure 2(b).

To be able to correctly add and subtract the different contributions of the S-parameters obtained as a result of the individual electromagnetic simulations, the following considerations are taken into account. Q3D simulator defines *nets* that indicate the electrical connections of conductors [10], in each net a *sink* and multiple *sources* can be defined, they represent terminals connecting points where currents can go in or out. For each net the inductance and resistance between each source and the sink can be obtained converting the S-parameters into Z-matrix and closing the corresponding sink terminal to GND, as in Figure 3(a).

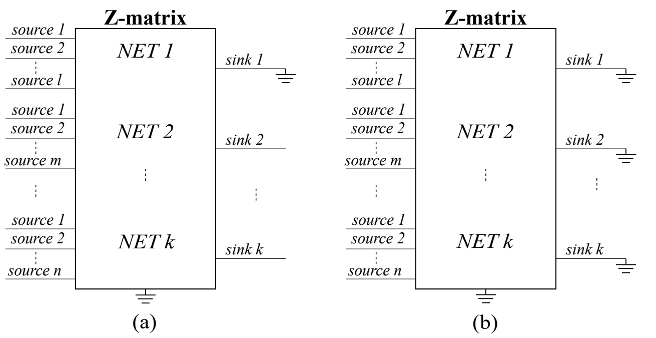

Fig. 3. Z-matrix converted from S-matrix obtained from electromagnetic simulation. (a): closing alternately the sink of each net to GND to compute the parasitic values of that net, (b): closing sink of every net to GND to compute the parasitic values between different nets.

For each net, the direct values of parasitic inductance and resistance between a source and sink defined in that same net (self-inductance and self-resistance) are obtained considering the imaginary part $\mathrm{Im}(\cdot)$ and real part $\mathrm{Re}(\cdot)$ of the specific element of the Z-matrix of the modified system as:

$$L_{i,i} = \mathrm{Im}\left(\frac{Z_{mod_{i,i}}}{2\pi f}\right), \quad R_{i,i} = \mathrm{Re}(Z_{mod_{i,i}}) \tag{1}$$

whereas for the parasitic components defined between different sources on the same net (mutual-inductance and mutual-resistance) we have:

$$L_{i,j} = \mathrm{Im}\left(\frac{Z_{mod_{i,j}}}{2\pi f}\right), \quad R_{i,j} = \mathrm{Re}(Z_{mod_{i,j}}) \tag{2}$$

The S-parameters obtained for the three simulations in Figure 2(b) are converted into Z-parameters considering for each net the sink terminal connected to GND. The obtained parameters of the PCB are summed to that of the PKG+traces simulation, while the matrix obtained from the traces simulation is subtracted. With the final matrix obtained by using equations (1) and (2) it is possible to compute the corresponding parasitic values for each net.

Conversely, it is also possible to consider the parasitic resistances and inductances between different sources on different nets (mutual-inductance and mutual-resistance). In this case, the matrix is computed as represented in Figure 3(b). All the sinks of the nets are now connected to GND and, in the same way as before but starting from different matrices, the parameters obtained from the PCB simulation are added to the parameters of the PKG+traces simulation and subtracted to the traces one. In this case we are interested in the mutual parasitic resistance and inductance between terminals in different nets only, so that equations (2) need to be used.

Reducing the Z-matrix as explained allows to isolate only the quantities of interest, therefore the new matrix of each system can be directly added and subtracted, and subsequently the mathematical operations to derive L and R can be performed.

III. Obtained Results

The explained method is applied to the simulation of a model of PCB and PKG with the quasi static-solver Q3D. Results are compared to the case of the simulation of the complete system which takes into account all the interactions between PCB and PKG, and to the most widespread method, i.e., the case of cascading PCB and PKG considering transmission parameters. Terms of comparison are the simulation time of each block according to the methodology adopted, and the accuracy computed by means of the maximum relative error between the resistance (or inductance) computed with the approximated method and with the full simulation, and averaged for the different considered cases. Data are collected in Table I. It is possible to observe that the simulation time of the proposed method is similar with that of the cascade one, and significantly reduced compared to the whole system simulation, decreasing the resource effort required by the electromagnetic simulation. Furthermore, with the proposed

This is the author's version of the article that has been presented at IEEE EPEPS2024
The editorial version of the paper is available at http://dx.doi.org/10.1109/EPEPS61853.2024.10754476

TABLE I
PERFORMANCE COMPARISON BETWEEN THE FULL SYSTEM SIMULATION, THE STANDARD CASCADE APPROACH AND THE PROPOSED APPROACH.

Methodology	Accuracy	Simulation	Time
Full system	100%	PCB+PKG	32h5min
Cascade	74%	PCB	18h18min
		PKG	6min
Proposed	90%	PCB	18h18min
		PKG+Traces	14min
		Traces	10min

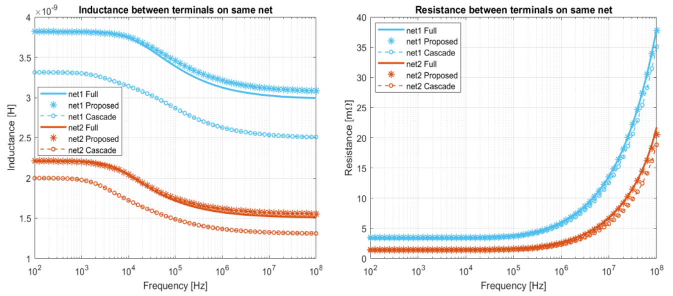

Fig. 4. Comparison between (self) parasitic inductance and (self) resistance on the same net (computed for source1 on net1 and for source1 on net2) starting from Z-matrix in Figure 3(a) using the three different methods explained.

approach, the parasitic effect between PCB and PKG is not completely neglected, as it happens in the case of the cascade approach, but it is possible to obtain an accuracy comparable to the full method. The values of parasitic inductances and resistances are computed for the case of the whole simulation, the cascade, and the proposed one starting from the electromagnetic simulation results. Parasitic inductance and resistance between sink and source in the same net, considering two nets, are represented in Figure 4. Parasitic inductance and resistance between sources on the two different nets previously considered are depicted in Figure 5. From the comparison it is possible to observe that with the proposed methodology the worst accuracy achieved at low frequencies is 98%, with the cascade method instead the value is 75%. Considering the entire frequency range the lowest accuracy for the presented method is about 70%, while for the other case is about 45%.

IV. CONCLUSION

A new methodology to model PCB and PKG in order to consider the interaction between the two elements in a quasi-static simulator is described, reducing the computational time by approximately half the time required for the full system simulation. Furthermore, better results are obtained compared to the most common method used that is the cascade of the two elements, where the interaction between PCB and PKG is neglected. The results obtained show comparable values respect to the complete simulation in the entire range of frequencies valid for the quasi-static simulator, and in particular completely overlapping at low frequencies, maintaining the computational cost of the cascade method.

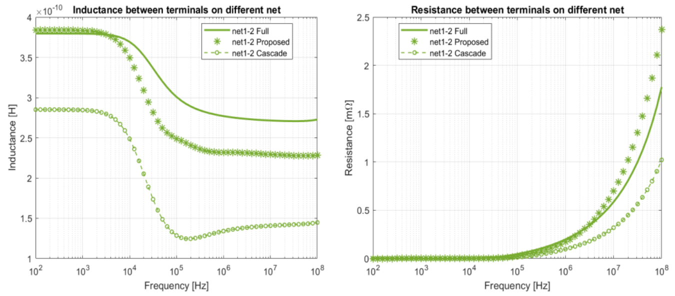

Fig. 5. Comparison between (mutual) parasitic inductance and (mutual) resistance between two sources on different nets (from source1 on net1 to source1 on net2) obtained starting from Z-matrix in Figure 3(b) using the three different methods explained.

ACKNOWLEDGMENT

This publication is part of the project PNRR-NGEU which has received funding from the MUR - DM 352 / 2022.

REFERENCES

[1] S. Simone, F. Pareschi, G. Setti, and D. Lena, "Equivalent skin effect model for time-domain analysis starting from electromagnetic simulator values," in *2024 25th International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Microelectronics and Microsystems (EuroSimE)*, Apr. 2024, pp. 1–6, doi:10.1109/EuroSimE60745.2024.10491514.

[2] Z. Chen, "A general co-design approach to multi-level package modeling based on individual single-level package full-wave s-parameter modeling including signal and power/ground ports," in *2012 IEEE 62nd Electronic Components and Technology Conference*, May 2012, pp. 1687–1694, doi:10.1109/ECTC.2012.6249066.

[3] J. Zhang, A. C. Scogna, T. Vincent, H. Shi, and H. Liu, "System level simulation solutions for high-speed channels - package and PCB interface," in *2013 IEEE International Symposium on Electromagnetic Compatibility*, Aug. 2013, pp. 593–598, doi:10.1109/ISEMC.2013.6670481.

[4] Y.-C. Chang, S. S. H. Hsu, D.-C. Chang, J.-H. Lee, S.-G. Lin, and Y.-Z. Juang, "A de-embedding method for extracting s-parameters of vertical interconnect in advanced packaging," in *2011 IEEE 20th Conference on Electrical Performance of Electronic Packaging and Systems*, Oct. 2011, pp. 219–222, doi:10.1109/EPEPS.2011.6100231.

[5] L. Teoh, C. Chiang, A. C. Scogna, K. Khrone, and H. Lee, "De-embedding method and segmentation approach using full wave solver for high speed channels in PCB/package co-design," in *2014 IEEE Electrical Design of Advanced Packaging & Systems Symposium (EDAPS)*. IEEE, Dec. 2012, pp. 33–36, doi:10.1109/EDAPS.2014.7030808.

[6] H. Dsilva, A. Jain, S. J, and A. Kumar, "Effective segmentation approach of package-to-PCB modeling using full-wave EM field solver," in *2020 IEEE Electrical Design of Advanced Packaging and Systems (EDAPS)*, Dec. 2020, pp. 1–3, doi:10.1109/EDAPS50281.2020.9312900.

[7] A. C. Scogna, C. Chiang, L. Lau, L. Teoh, and H. Lee, "Modeling methodologies for multi level PCB-package co-simulation & co-design," in *2013 IEEE 22nd Conference on Electrical Performance of Electronic Packaging and Systems*, Oct. 2013, pp. 57–57, doi:10.1109/EPEPS.2013.6703466.

[8] Z. Chen, "A lumped/discrete port de-embedding method by port connection error-cancelling network in full-wave electromagnetic modeling of 3d integration and packaging with vertical interconnects," in *2013 IEEE 63rd Electronic Components and Technology Conference*, May 2013, pp. 1980–1987, doi:10.1109/ECTC.2013.6575850.

[9] A. C. Scogna, C. T. Chiang, K. Krohne, L. K. Teoh, and H.-Y. Lee, "Signal integrity analysis for high speed channels in pcb/package co-design interface: 3d full wave vs. 2d/hybrid approach & full model vs. segmentation approach," in *2013 IEEE 15th Electronics Packaging Technology Conference (EPTC 2013)*, Dec. 2013, pp. 585–588, doi:10.1109/EPTC.2013.6745787.

[10] "Q3d extractor help," 2024.