ANALYSIS OF 3D CONJUGATE HEAT TRANSFERS IN ELECTRONICS

Fradin J.P., Molla L. and Desaunettes B.

EPSILON INGENIERIE, California - Voie 5 - BP:653 - 31319 Labège Cedex
tel : (33)-05-62-24-31-07, fax :(33)-05-61-39-99-16, e-mail : fradin@epsilon.fr

Abstract

An efficient method for the analysis of real 3D conjugate heat transfer for electronic devices is presented. This methodology is based on the coupling of two software : a conductive software based on the Boundary Element Method (REBECA-3D®) and a convective software based on the Volume Finite Method (FLUENT). The methodology is tested on a Multi Chip Module (CPGA224) for which experiments have been performed by the CNRS (French National Center for Scientific Research).

1. Introduction

In order to provide the electronic function and to improve reliability and performances of electronic packages, one of the objective of the electronic companies is to accurately predict the operating temperatures of critical parts. Then, the current miniaturisation of electronic devices and the higherpower density have introduced increasing of the temperature of the chips. Forced convection by flowing air over the component is also mandatory.

In the domain of the conjugate heat transfers, all physical phenomena must be taken into account as well the complexity of the flow (real 3D, recirculation, wakes interacting with obstacle) as the high gradients inside the chips. To reach these objectives, a new methodology is used, which combine two different industrial modules : a conductive software (REBECA-3D) and a computational fluid dynamics (CFD) module (FLUENT).

In this paper, we first present the selected numerical technique before testing and validating it on an industrial example on which experiments have been performed by CNRS.

2. Numerical techniques

2.1 Why to develop a new approach?

In many cases, the global and classical numerical methods to compute conjugate heat transfers are cumbersome to use when great accuracy is required.

This problem is all the more important since it is linked to scale difference, for example between the thickness of the chip and the length of the Printed Circuit Board, on which the package is connected. Global codes are usually based on Finite Volume Method combined with multigrid technique for which similar sizes of meshes are required. To reach a sufficient accuracy by such a global approach, the number of nodes must be counted in hundreds of thousands of nodes and even often in millions of nodes. This is an obvious obstacle to industrial applications.

As such a global and accurate process is still complex and slow for designing, many engineers prefer to use approximate method (which is more or less reliable but very fast) or a global approach with very few nodes. This last technique does not provide a good accuracy and above all makes it impossible to quantify the introduced errors.

This drawback is particularly disturbing for designing as the process usually involves a series of modifications at the level of geometry, boundary conditions, conductivities... because many calculations are required.

To be able to solve a complex problem with a good accuracy in a reasonable time, a coupled approach is better [1]. The conductive and convective transfers have to be coupled at the boundary condition level. Our objective is then to consider the best software depending on the heat transfer mode : one for conduction and one for convection. The next step is to couple them and to achieve the convergence.

ED&TC '97 on CD-ROM

2.2 The conductive software : REBECA-3D

Developed by Epsilon Ingenierie, REBECA-3D (**RE**liability **B**oundary **E**lement **C**onductive **A**nalyser in **3D**) is a thermal software. It allows the determination of temperature fields and flux distributions for 3D systems when thermal transfers are driven by conductive exchanges [2] [3].

REBECA-3D is based on the use of the Boundary Element Method whose principal property is to provide a reduction in dimension, so that three-dimensional problems are reduced to a sequence of two-dimensional problems, involving only surface integration. So, boundary temperatures and fluxes are calculated without the discretization of the domain. The saving of nodes allows to solve more complex problems than by using more classical methods such as Finite Difference Method, Finite Element Method... The advantage is the CPU time saved by REBECA-3D.

The Boundary Element Method is also a very powerful tool that can be used to carry out a great number of parametric studies with a very few calculations. The conception of the software is carried out in accordance with these requirements.

REBECA-3D is then an ideal tool for engineering design. It makes it easier to generate the data required to run a problem and to carry out the modifications needed to achieve an optimum design.

2.3 Convective software

For the modelling of the convective transfers, a commercial computational fluid dynamics (CFD) code has been used [4]. This flow solver is based on Finite Volume Method combined with Multigrid Method. Physical models include laminar or turbulent incompressible flow, natural convection and boundary convection for fluid flows.

2.4 Coupling conduction and flow

Coupling of the conductive and convective transfers is performed by passing energy between both solvers at the fluid/solid interface. Although there are in fact two disjoint meshes at the interface between fluid/solid, (a convective mesh and a conductive mesh), it is possible to transfer energy at the boundary condition level with the notion of mean value on a facet (geometrical surface on which only one boundary condition is applied).

Due to discontinuity and convergence, the possibilities of algorithm are reduced. The use of the mean convective transfer coefficient by facet is the best solution. This is the easiest way to transfer energy without any mesh problem.

On the conductive model, Robin boundary conditions are prescribed as follows :

⇨ for the first iteration, all the convective coefficients are identical and equal to an estimated value. The mean temperature of the fluid is also predicted.

⇨ for the next iterations, the convective coefficients of each facet and the fluid mean temperature are calculated by the CFD software. Each mean convective coefficient is computed by dividing the total flux on the facet by the difference between the fluid mean temperature and the wall temperature.

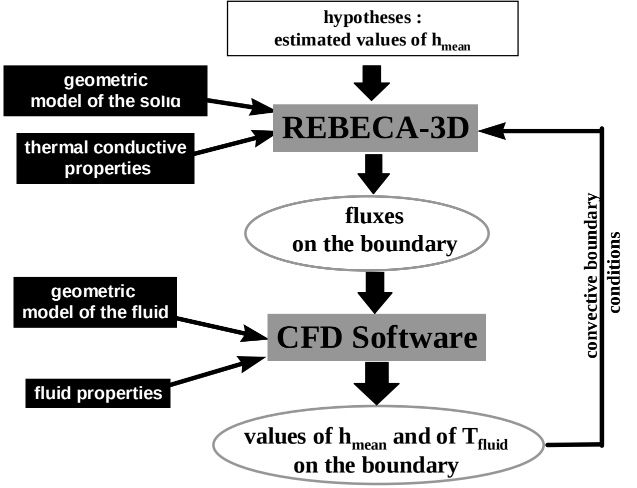


fig. 1 : Schematization of the methodology

With these convective boundary conditions, REBECA-3D computes the conductive field of the package and gives the flux density and the temperature of each mesh on the boundary and then the mean temperature and the total flux of each facet. These fluxes are then prescribed on the interface facets of the convective model.

When values of change become identical from an iteration to another with regard to a software, the convergence of the coupling is achieved. The test of convergence may then be performed between the temperatures of the wall computed by both software.

2.5 Validation on an academic example

The methodology has been first validated on an academic example that a CNRS laboratory has studied using a global method (Finite Volume combined with multigrid technique). This problem of validation simulates in a realistic but academic way the cooling of an electronic equipment in forced convection. An obstacle on a PCB with the presence of a small dissipating chip is considered [5]. Comparisons between the results given by the reference code and our approach allowed to chose the methodology relating to the mean convective heat transfers.

It is interesting to notice that the coupling is set up with mean values at the facet level. So if the zones are not thermally homogeneous, the transfer of parameters may introduce some small differences between conductive and convective results. It is obvious that the more thermally homogeneous the facets are, the better the results become. Choosing a reference temperature close to the facet temperature is also preferable. In the case of an automatic coupling, it would be better to transfer the mesh convective coefficients and thereference temperatures, equal to these of the first cells. Such an approach is actually forbidden by the limitation of the convective software.

The use of two software gives a very good accuracy for the temperature of the chip with very few steps of iterations. In general, less than 5 iterations are required. To converge the fluid mechanical equations, CPU time consumption is required only for the first calculation.

As meshes are disjoint on the geometry, the required model mesh size is dramatically reduced. In this study, the objective was only to test the methodology, to validate it and to apply it on an industrial and realistic example. So, at present, two models are built : a conductive and a convective model. The final objective is to substitute the prototype of interface for a user-friendly one without manual action to transfer parameters

3. Application to a MCM

This example deals with the cooling of a MCM (Multi Chip Module : CPGA224) mounted on a Printed Circuit Board and made up with with 16 semiconductor chips.

3.1 Description of the example

To demonstrate the whole interest of the methodology on an real and industrial example, the attention has been focused on the thermal behaviour of a Multi-Chip-Module : the CPGA224. This structure has been studied in the context of the European collaborative project named **DELPHI** (for **DE**velopment of **L**ibraries of **PH**ysical models for an **I**ntegrated design environment) under contract with Alcatel Espace. The project consists of a mix of industrial companies manufacturing a range of electronic equipments, a software supplier and a university-linked research institute, namely : Alcatel BELL, Alcatel Espace, NMRC, Philips, Thomson CSF, Flomerics. The project is concerned with the development and experimental validation of thermal models of a variety of electronic parts.

The electronic package is a block which is about 45mm long and 3mm deep (cf. fig. 2). The studied structure is really 3D with the same dimensions in length and in width. It consists of 16 semiconductor chips mounted in a ceramic chip carrier. The chips are stuck by their inferior face in the ceramic cavity and dissipate a power P. They are grouped by 4 to form 4 zones. Heat dissipation within the silicon die is assumed to be volumetrically uniform.

The small cavity of the chip carrier is filled with air and covered with a lid. As it is only about 1mm high, no mass flow exists and the cavity can be modelled as a conductive volume. The package is mounted on an epoxy substrate which is 1.5mm thick (size : 160mm x 120mm) with pins.

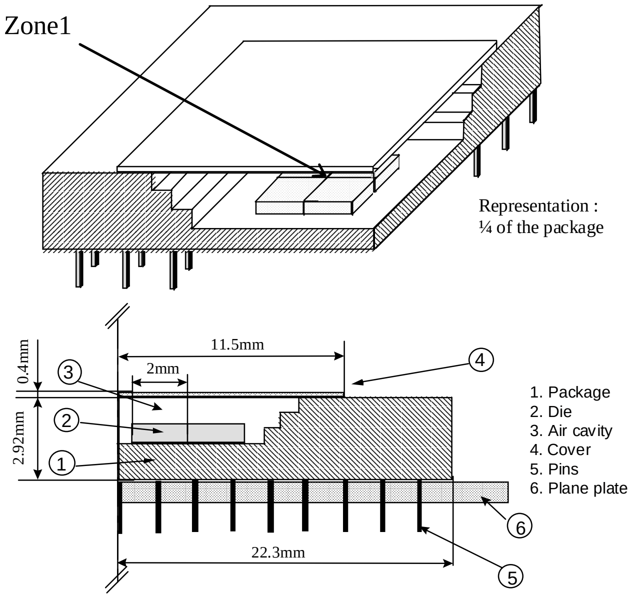


fig. 2 : Structure of the mounted component

Some thermal material properties such as the thickness of the glue and the conductivity of the ceramic have been characterised by a first experiment based on a double jet in a bath. In this case, because of the high value of the convective coefficient (heat coefficient in the order of 20000W/m²K), only conductive aspects needed to be modelled and no coupling is necessary [6].

The thermal conductivities of the involved materials are listed in table 1 :

material	ceramic	chip	lid	glue	pin	plate
conductivity	22.0	155	130	1.0	17	0.5

tab. 1 : Thermal conductivities (W/mK)

For the computational study, the thickness of the glue has been identified to 10µm. It is obviously one of the most difficult parameter to quantify. Because of its slight thickness, the glue has not been modelized by a domain but only by a contact resistance.

3.2 Testing in rough water

This experiment (performed by a CNRS laboratory (L.E.T. from Poitiers) for Alcatel Espace) deals with the cooling of the CPGA224 in a bath of rough water. So, in this study, the contribution of radiative heat transfers may be neglected.

The geometry of the experimental setup is given on figure 3. The sample is put in water bath in vertical position with the top and bottom surfaces of the package parallel to the gravity and to the outlet of the pump.

The package is cooling by forced convection. The agitator generates a flow rate of 6 l/mn in a diameter of 8 mm. The inlet velocity of the water at the exit of the agitator is equal to 2,65 m/s. The mean temperature of the fluid is 20,4 °C.

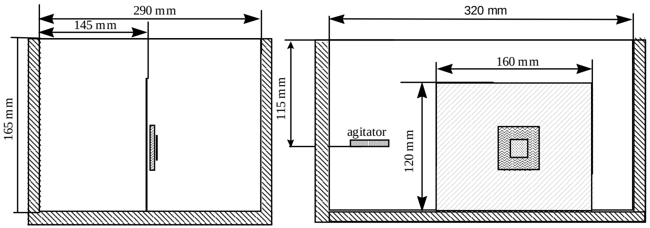


fig. 3 : Schematic view of the experimental setup

In the table 2, we give the differences between the measured mean temperatures and the fluid temperatures:

	Zone 1	Zone 2	Zone 3	Zone 4
Powers (W)	5,83	3,08	4,51	4,12
(T_{mean}-T_{fluid}) (°C)	32,4	21,2	23,9	22,3

tab. 2 : (T_{mean} - T_{fluid}) (°C) of the different zones

3.3 Modelling of the pins

In this experiment, the main problem deals with the modelling of all the pins which link the package on its support. Due to their small size and their important number, it is impossible to exactly represent them.

When the cooling is performed with a fluid such as water, the fluid mechanics may not be neglicted between the package and the plate (cf. fig. 4). Thus, we have to consider an appropriate modelling dealing with the conductivity of the pins and the velocities in this zone.

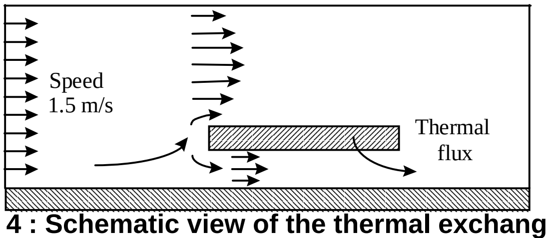
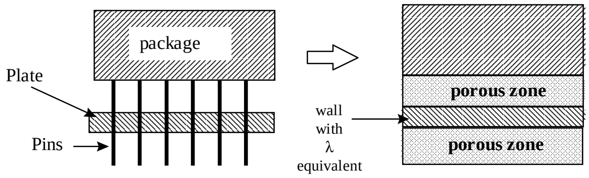


fig. 4 : Schematic view of the thermal exchanges

The chosen methodology consists in modelling the pins with FLUENT by a porous medium characterised by the permeability (pressure drop of the medium by unit of length, 10 Pa/m), the porosity α (ratio of the fluid volume to the total volume, α=0.974) and the conductivity λ_s of the solid part of the medium. Through the porous medium, FLUENT adds a term of pressure drop in the governing momentum equations of fluid mechanics and calculates a equivalent conductivity $\lambda_{eq} = \alpha \cdot \lambda_{fluid} + (1-\alpha) \cdot \lambda_{solid}$. These porous media are prescribed on both sides of the plate (cf. fig. 5).



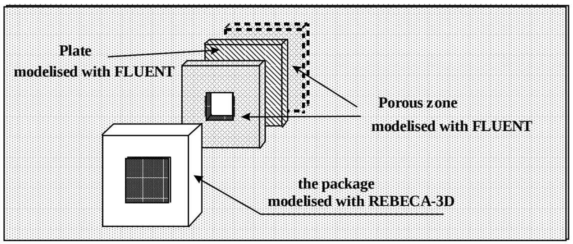


fig. 5 : Principle of the modelling of the pins

In the part where the plate is crossed by the pins, the conductivity of the plate is modified. An equivalent conductivity is calculated as follows :

$$\lambda_{eq} = \alpha \cdot \lambda_{plate} + (1-\alpha) \cdot \lambda_{pins} = 1W/mK .$$

As the interface between the package and the flat plate has been modelled by FLUENT, the conduction in the package and in the plate has to be divided in two parts. The selected solution consists in modelling the conduction in the plate with FLUENT. The introduced inaccuracies are insignificant faced with the uncertainties of the model. Because of the dimensions of the package, as previously said, the only way to accurately model it, is to use the Boundary Element Method and REBECA-3D.

3.4 Results

When the convergence of the model is performed, the mean temperature difference between the numerical and experimental result is equal to 1.8°C (error 5%).

	Zone 1	Zone 2	Zone 3	Zone 4
Numerical	33,1	18.8	26,4	24,0
Experimental	32,4	21,2	23,9	22,3

tab. 3 : Comparison of the results : (T_{mean}-T_{fluid})(°C)

Like in the academic example, the convergence of the methodology is performed in less than 5 iterations. For FLUENT, the convergence of the mechanical equations is long to be set up contrary to the thermal equations. This explains the difference in CPU between the first FLUENT calculations (some hours) and the following ones (only few minutes). After the first step of iteration, mechanics is less sensitive to thermics and modifications of boundary conditions do not imply problems of convergence.

3.5 Views

Figure 6 shows the gradient of temperatures of the chips given by REBECA-3D.

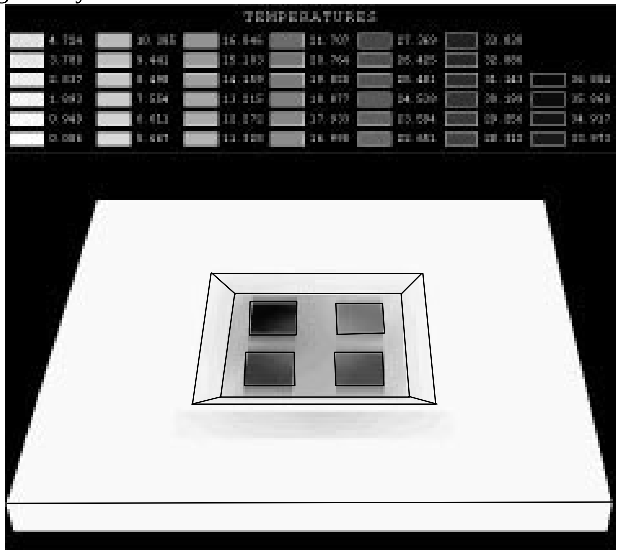


fig. 6 : Conductive temperatures (REBECA-3D)

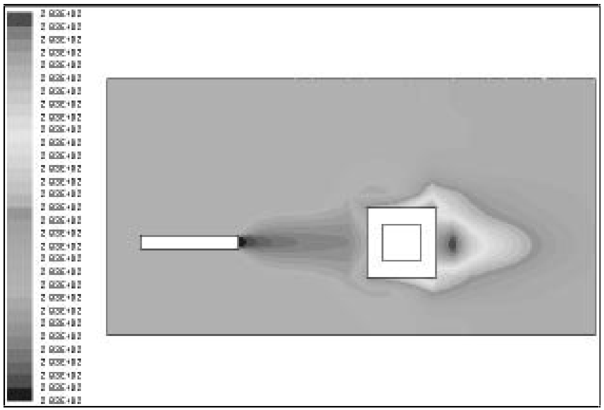


fig. 7a : Temperature field in a rough bath

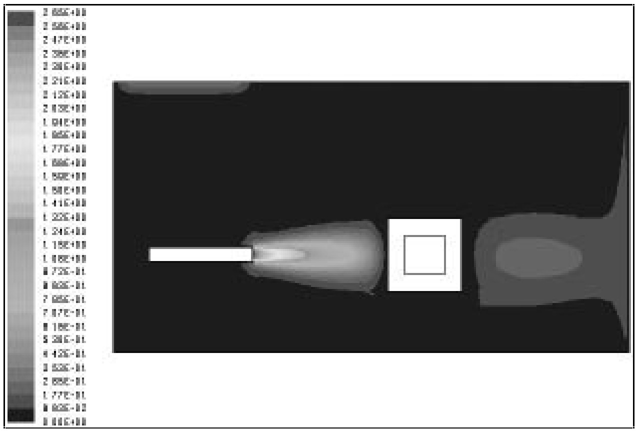


fig. 7b : Velocity field in a rough bath

Figure 7a shows the temperature field in the rough bath and the increasing of temperature behind the package due to the heating of the fluid. Figure 7b shows the velocity field in the rough bath and the zone of separation just behind the package. After this zone, there is an increasing of the velocity because the fluid has ended to lap the package.

4. Conclusion

Experimental investigation has been carried out to test and compare the numerical results. Our methodology allows to tune the results with less than two degrees when few parameters are previously identified. This has been sucessfully performed thanks to the accuracy of the modelling of the conductive and convective transfers.

We have found very interesting to modelize the complex mechanical and thermal influence of the pins while using the notion of porous surface.

Due to the high value of the convective coefficients in water, it is useless to take into account the radiation modelling. However, if there is a need, as for example in calm air, a correct term may be added at the boundary condition level (cf. a next paper).

Today, this coupled method seems to be one of the best approach to modelize conjugate heat transfers.

5. Acknowledgements

The Multi Chip Module CPGA224 has been studied under contract for Alcatel Espace in the context of the DELPHI Project. The authors wish to thank C. Lacaze (of Alcatel Espace). J.J. Vullierme and R. Zugari from the L.E.T. (C.N.R.S.) of Poitiers have performed bath tests.

6. References

[1] Sugavanam R.; Ortega A.; Choi C, "Numerical Investigation of Conjugate Heat Transfert from a Flush Heat Source on a Conductive Board in Laminar Channel Flow", Int. J. of Heat and Mass Transf, Vol.38, No.16; pp 2969-2984; . Nov 1995.
[2] J.P. Fradin and B. Desaunettes., "REBECA-3D : The Software for Conductive Fluxes and Temperature Fields Prediction", 26th ICES, Monterey, July 8-11, 1996
[3] J.P. Fradin and B. Desaunettes - "Calculations of the Temperature Distribution Throughout Electronic Equipments by the B.E.M. with the REBECA-3D Software", Eurotherm 45 : Thermal Management of electronic Systems, Sept 1995
[4] Technical documentation of FLUENT
[5] B. Fourka & J.B. Saulnier, The Multigrid Technique : An Efficient Tool for the Analysis of 3D Conjugate Heat Transfert in Electronic Boards, Eurotherm Seminar 36, Poitiers, 1994.
[6] L. Molla, Etude du couplage thermique conductif-convectif, Sept. 1996, EPSILON Ing. RE001/STA/0996/LM/1.0
[7] J.J. Vullierme, R. Zugari, Etude numérique et expérimentale du CPGA180, L.E.T., Alcatel Espace,1995