Application of the CFD Method for Heat Transfer Simulation

Anna V.Andonova[1] and Nadejda M. Kafadarova[2]

Abstract – **Modern CFD codes now are used for microsystem heat transfer simulation. The outputs of these codes are 3D maps of temperature distributions. These codes can often predict a hot spot with good accuracy, but offer no diagnostic help in determining what to change for the next trial.**

This paper proposes a set of diagnostic parameters that should be calculated for each critical component. From these parameters, it could be answer, during the heat transfer simulation process not only whether an overheating problem exists but what is the most likely cause.

Conjugate transient heat transfer from a sample carrier mounted microsystem component was numerically modeled using a CFD code dedicated to the thermal analysis of electronic system. It is used to evaluate the potential for improving module thermal design

Keywords – **Computational Fluids Dynamics (CFD), Simulation, Heat transfer, Thermal management, Reliability, Virtual prototyping**

I. INTRODUCTION

The need to reduce development cycle times and costs has lead to increased efforts to use virtual prototyping during the design phase in electronics microsystems. As the complexity of these systems rise, addressing thermal management issues has become increasingly important. The ultimate goal of system thermal design is not the prediction of component temperatures, but rather the reduction of thermally associated risk to the product [1]. Many tools exist to assist thermal design engineers during this process, including heat transfer correlations, Flow Network Modeling (FNM), Computational Fluid Dynamics (CFD), and experimental measurement techniques. The key to efficient and comprehensive thermal design is not, necessarily, the choice of the "best" tool for design, but rather the optimized integration of available tools.

The flow and heat transfer situations encountered in microelectronics cooling are very tricky .

There are many ways in which an "acceptable' thermal design can fail: inadequate or poor flow distribution, thermal stratification (poor mixing), low heat transfer coefficient, or power dissipation higher than expected. Heat transfer simulation relies heavily on CFD codes that predict the flow and temperature distributions using more or less elegant turbulent flow models.

The applicability of CFD analysis to predict steady-state, single-component PCB heat transfer in free and forced convection has been well established [2].

The use of such CFD tools enable the following: l) better prediction of component, board, and system temperature, 2) reduction in design-cycle time, 3) reduction in experimental testing, and 3) parametric materials investigation for a number of boundary conditions. This paper proposes diagnostic parameters that should be calculated for each critical component. From these parameters, it will be said not only whether an overheating problem exists but what is the most likely cause.

The design analysis was performed early in the module design process at a time when knowledge of the thermal characteristics of a module design is most useful. An example for a sample was demonstrate.

II. PRINCIPLE COMPUTATIONAL REQUIREMENT

A. Component level modeling

To create a good model of an entire microsystem, it is essential to develop accurate models of the components that are contained within the system. The underlying aim for this analysis approach is to create behavioral models based on the known characteristics of the component.

On the electrical side, an extensive library of component models for systems simulations has been created. These models are based on the electrical characteristics of the devices as supplied by the manufacturer or as determined through electrical testing. To develop thermal models of the system's components, the devices are subjected to thorough testing during which the power dissipated and the temperatures attained are carefully monitored for various loading scenarios. This test data is then used to design a thermal model of the device that reflects the observed response as closely as possible over a broad range of electrical loading scenarios. This automated approach also allows the development of models that are highly accurate as well as computationally efficient.

Though a variety of microsystem layouts are possible, it will simplify the discussion to limit these considerations to a current standard system design.

[1] Department of Microelectronics, Technical University of Sofia, FETT, 8 Kl. Ohridski Str., 1797 Sofia, Bulgaria, E-mail: ava@ecad.tu-sofia.bg

[2.] Technical University - Sofia, Plovdiv Branch, 25 Tsanko Diustabanov Str., 4000 Plovdiv, Bulgaria, E-mail: n_kafadarova@yahoo.com

Accurate prediction of components junction temperatures and the overall system temperature field is necessary to allow for adequate reliability and performance analysis in the design process. Ideally, computational fluid dynamics (CFD) simulations of the electronic microsystem, with detailed modeling of the components and conjugate heat transfer interactions, would be utilized to predict temperatures at the component and system level, but these simulations require significant computational resources. This computational expense is the result of large variability in length scales at the component and system level. In conventional discretization schemes, the smaller package length scales result in a larger number of control volumes in the surrounding fluid regions where fewer volumes might suffice. This is because the discretization required in the package translates into the fluid regions, and vice versa. System level compact thermal modeling attempts to eliminate the small length scales associated with modeling the details of the package by using models with length scales comparable to those required by the system level CFD simulation.

B. Diagnostic parameters for each critical component.

Table I summarizes the input data and results generated by the CFD and FE analyses of the commercial CFD software, FLOTHERM ®. Geometry, grid and temperature information is passed from the CFD solver to the FE solver at the end of the CFD calculation. The outputs of these codes are "maps" of flow and temperature distribution. For identifying potential problems, these maps provide little or no guidance as to how to solve the problems they reveal.

TABLE I
INPUT DATA AND RESULTS

CFD Thermal Tool
Data and Solution:
· Fluid and Solid Regions
· Structured Mesh in Solid and Fluid Regions
· *Finite Volume Solver Results:*
· Fluid Velocities
· Temperatures
Derived Quantities:
· Fan Operating Point

It is hard to develop a systematic approach to problem solving where everything changes when anything changes. Classical theory for internal passage heat transfer uses a heat transfer coefficient based on the mean fluid temperature:

$$q_{conv} = h_m A(T_0 - T_m) \quad (1)$$

$$T_0 = T_m + q_{conv}/h_m A, \quad (2)$$

where A is the area, m^2, h_m is the h based on (T_o-T_{mean}), W/m^{2}K, h is the heat transfer coefficient, W/m^{2}K, q_{conv} is the convective heat transfer rate, W.

Unfortunately, electronics cooling situations are characterized by abrupt changes in wall temperature, from one component to the next. Under those conditions, h_m can range between +∞ and -∞ . The experimentally measured heat transfer coefficients presented in the electronics cooling literature are not values of h_m, they are referenced to the adiabatic temperature of the component, If those values are used in Eq.1, the actual temperature rise will be higher than the calculated one, sometimes by as much as 40%.

When h is defined in terms of the mean temperature (as it may be by a CFD code), then the value of h depends on the power distribution on the board and will change when the power distribution changes.

If it is introduced a more suitable definition of h for electronics cooling applications in [3]:

$$q_{conv} = h_m A_{ad}(T_0 - T_{ad}) \quad (3)$$

$$T_0 = T_{ad} + q_{conv}/h_{ad} A \quad (4)$$

where T_{ad} is the temperature the component attains with no power applied and with no radiation or conduction to it.

Thinking in terms of h_{ad} instead of h_m gives the designer more tools to work with. This can be seen by comparing Eq.2 and Eq. 5. Most importantly, however, h_{ad} is independent of the temperature distribution on the heated wall being determined by geometry and flow field alone. This allows the heat transfer engineer to deal with the overall temperature rise as the sum of a set of linearly independent temperature rise components that can be individually measured (or computed).

Thefour energy flows acting on a componen are shown on Fig.1.

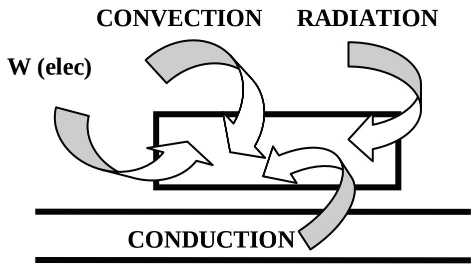

Fig. 1. Energy balance terms for an active component.

If the component is running too hot, it is important to know which of these four terms is causing the trouble. Starting from the energy balance equation, Eq. 6, the terms can be arranged into groups that represent the individual contributions to the component temperature:

$$q_{conv} + q_{rad} + q_{cond} + W_{elec} = 0 \quad (6)$$

Eq. 7 describes the four factors that affect the temperature rise of a component:

$$\Delta T_{overall} = (\Delta T_{ad}/\Delta T_m)\Delta T_m + \Delta T_{cond+rad} + \Delta T_{elec} \quad (7)$$

The CFD code calculates the operating temperature, T_o, in its normal mode of operation. The diagnostic benefit comes from breaking up the total temperature rise at operating conditions to reveal these four factors, using Eq. 7 as a guide.

B. Example

Consider a microsystem consisting of regular array of components. Most are un-powered but in one region, a 10 watt component is installed immediately downstream of one dissipating 18 watts. Carrier spacing is 2.25 times the component height, and the inlet velocity (measured upwind of the first row of components) is 3.07 m/s. The CFD code predicts an operating temperature of 115 ^{0}C in a 20 ^{0}C ambient. The following information is needed for each component under normal operating conditions:

Intel Temperature – 20^{0}C
Adiabatic temperature – 45,8^{0}C
The mixed-mean coolant temperature – 24,7^{0}C
The value of h_{ad} – 51W/m^{2}C
The pressure drop per row – 1,61 Pa
Operating temperature – 115 ^{0}C

The adiabatic temperature rise is 5.5 times the mixed-mean coolant rise. The coolant flow is badly stratified. If it were perfectly mixed, the operating temperature would drop by more than 20.

III. Module Design Process

To be most beneficial, thermal design analysis should be performed early in the module design cycle during component placement studies.

Electronic modules may be destined for use in a wide variety of applications and environments. Module operational times may be brief and therefore component package and local PCB/carrier heat capacitance effects may be dominant thermal concerns. In other cases, module operational time may be lengthy, in which case the proper design of heat conduction paths from the components to heat sinks may be paramount.

Frequently, active cooling systems are used, and temperature control is accomplished by direct immersion into a quiescent or circulating coolant. In low atmospheric pressure or space applications, radiation may be a significant factor as well. Thus, a very wide variety of applications are possible, each requirements was developed and translated into with varying modeling requirements. With these and other considerations in mind, a set of design analysis requirements for modeling in FLOTHERM.

The initial attempts, featured numerous internal plates and external walls, which led to many questions as to how the plates and walls would interact, and whether the whole process would prove too confusing for the module designer to use effectively. The following process allows an Auto Therm generated lib file to be analyzed within FLOTHERM using prescribed heat transfer coefficients in a conduction only mode. Auto Therm can generate a 3D model of a circuit board with components as shown in Figure 2. It consists of: a cuboid block representing the circuit board and a series of cuboid blocks representing the components with non-conducting internal plates insulating the sides, and conducting plates setting the $\Theta_{j\text{-}c}$ and $\Theta_{j\text{-}t}$ values.

The principle of the approach is the translation of the resistance of the surface-fluid heat transfer coefficient into a thermal resistance created by an internal plate with a k, *t* combination through the relationship:

$$Q = hA\Delta T = kA\Delta T / t \text{ hence } h = k/t.$$

Table II
Input data and results with heat source

Geometry Model	Type	X Position (m)	Y Position (m)	Z Position (m)	X Size (m)	Y Size (m)	Z Size (m)
Root Assembly	Assembly						
Structure	Assembly	0	0	0	0.25	0.075	0.3
Chassis	Enclosure	0	0	0	0.25	0.075	0.3
Low Y Plate	Perforated Plate	0.02	0	0.02	0.19	0	0.08
Low Z Plate	Perforated Plate	0.02	0.01	0.3	0.19	0.04	0
High Z Plate	Perforated Plate	0.02	0.01	0	0.19	0.04	0
Box Temperature	Monitor Point	0.125	0.0375	0.15	0	0	0
PSU	Assembly	0.145	0.01	0.235	0.075	0.04	0.05
PSU Heat	Source	0	0	0	0.075	0.04	0.05
PSU Temperature	Monitor Point	0.0375	0.02	0.025	0	0	0

Table III
Input data and results without heat source

Geometry Model	Type	X Position (m)	Y Position (m)	Z Position (m)	X Size (m)	Y Size (m)	Z Size (m)
Root Assembly	Assembly						
Structure	Assembly	0	0	0	0.25	0.075	0.3
Chassis	Enclosure	0	0	0	0.25	0.075	0.3
Low Y Plate	Perforated Plate	0.02	0	0.02	0.19	0	0.08
Low Z Plate	Perforated Plate	0.02	0.01	0.3	0.19	0.04	0
High Z Plate	Perforated Plate	0.02	0.01	0	0.19	0.04	0
Box Temperature	Monitor Point	0.125	0.0375	0.15	0	0	0
Electronics	Assembly	0	0	0	0.19	0.0016	0.21
PCB 1	PCB	0.02	0.01	0.23	0.19	0.0016	0.21
Component	PCB Component	0	0	0	0	0	0

The final stage of the process is to import the “plated” lib file into a pre-prepared project file consisting of: highly conductive (hence isothermal) cuboid and a number of planar sources fixing the temperature of the cuboid to the “ambient” temperature (T_a).

With the assurance that the model making process was manageable, calculate the steady state temperature distribution

To illustrate the model making process a sample module was selected randomly as that described above, which had not been incorporated into the program to demonstrate calculation of the steady state temperature distribution. The data are shown in Table II and Table III with and without heat source respectively.

Arbitrary, typical thermal parameters were applied to each component. These features require editing in FLOTHERM if they are to be incorporated and thermal boundary conditions were applied. It was decided totally arbitrarily to calculate the steady state temperature distribution within a control volume for a high temperature free convection cooling case.

Fig. 2 shows the temperature distributions calculated for that case, and Fig. 3 shows the temperature profiles for the case with and without heat source respectively.

The design analysis can be performed early in the module design process at a time when knowledge of the thermal characteristics of a module design is most useful.

Thermal designers can participate closely in the overall process, by establishing and maintaining the component model databases and thermal parameter characterizations. These parameters would be predetermined to coincide with those manufacturing.

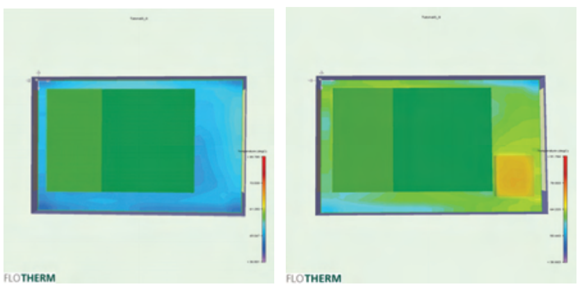

Fig. 2. Free convection temperature prediction for sample modules with and without heat source

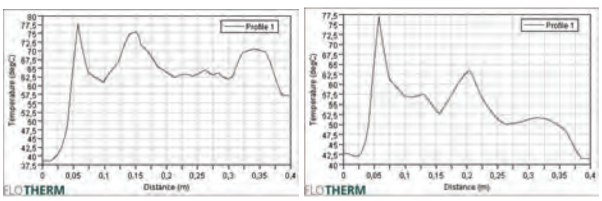

Fig. 3. Profile of the temperature vs. distance

IV. Conclusion

Numerical methods have demonstrated their ability to produce useful and generally reliable estimates of the temperatures in electronic cooling situations.

These codes can often predict a hot spot with good accuracy, but offer no diagnostic help in determining what to change for the next trial.

This paper shows that the usefulness of a commercial code could be extended, and its value to the user greatly enhanced, paper shows that the usefulness of a commercial code could be extended, and its value to the user greatly enhanced, by equipping it to generate the diagnostic information needed by the heat transfer engineer.

The following information is needed for each component under normal operating conditions:

- Its operating temperature
- Its adiabatic temperature
- The mixed-mean coolant temperature approaching it
- Its net radiation and conduction inputs
- Its electrical power dissipation
- The value of *had*
- The pressure drop per row in the vicinity

With these values known, the heat transfer engineer can diagnose the cause of overheating and more quickly arrive at an appropriate re-

The electronics cooling CFD package FLOTHERM is a computational fluid and heat transfer analysis and design package specifically for the analysis of electronic equipment. FLOTHERM makes use of the finite volume method to analyze three-dimensional geometries from chip level to system level.

These can all be obtained without undue effort either by integration of values already available over appropriate areas or by “freezing” the flow calculation and sequentially changing the power distribution (relying on the linearity of the energy equation and the fact that h_{ad} does not change when the temperature distribution changes).

Acknowledgement

Research in this subject has recently commenced through a NIS TUS funded project, which started in 2007.

References

[1] J. H, Lienhard, *A Heat Transfer Textbook*, third edition, Phlogiston Press, Cambridge Massachusetts, 2006.

[2] P. Gauche, “A Design Approach to Thermal Characterization of Forced Convection Systems Using Superposition in CFD” IMAPS, Conference Proceedings, pp. 18-22, Boston, USA, 2000.

[3] D. E. Arvizu and R.J. Moffat, “The Use of Superposition in Calculating Cooling Requirements for Circuit-Board-Mounted Electronic Components” Proceedings of the 32nd Electronic Components Conference, pp. 133-144, San Diego, USA, 1982.

[4] A. Andonova, N. Kafadarova, G. Pavlov, “Investigation of the Effective Thermal Conductivity of the PCB”, Годишник на ТУ-София, том 56, кн.1, стр. 120-125, 2006.G. Blad, D. Klepacki, J. Potencki,, “Application of Electro-thermal Analogy for Complex Simulation of Hybrid Power Controllers”, ICEST, Conference Proceedings, pp. 262-265 Sofia, Bulgaria, 2006,.