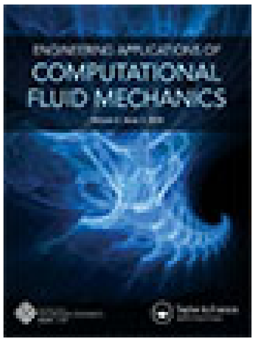

Engineering Applications of Computational Fluid Mechanics

ISSN: 1994-2060 (Print) 1997-003X (Online) Journal homepage: https://www.tandfonline.com/loi/tcfm20

CFD Modeling of Forced Cooling of Computer Chassis

Emre Ozturk & Ilker Tari

To cite this article: Emre Ozturk & Ilker Tari (2007) CFD Modeling of Forced Cooling of Computer Chassis, Engineering Applications of Computational Fluid Mechanics, 1:4, 304-313, DOI: 10.1080/19942060.2007.11015201

To link to this article: https://doi.org/10.1080/19942060.2007.11015201

CFD MODELING OF FORCED COOLING OF COMPUTER CHASSIS

Emre Ozturk* and Ilker Tari**

** Anova CAD-CAE-TEST, ODTU-OSTIM Teknokent Z-3 Yenimahalle, Ankara 06370, Turkey*
*** Middle East Technical University, Mechanical Engineering Department, E206, Ankara 06531, Turkey*
E-Mail: itari@metu.edu.tr (Corresponding Author)

ABSTRACT: In this study, Computational Fluid Dynamics, which has taken its position in the thermal design of electronic packages, was used in order to draw a CFD road map for forced cooling conjugate heat transfer analyses in heat generating electronic systems. The main sources of error in CFD analyses arise from inappropriate numerical models including turbulence models, radiation modeling and discretization schemes, insufficient grid resolution, and lack of convergence. A complete computer chassis model with heat sinks and fans inside was created and parametric analyses were performed to compare the effects of different turbulence models, discretization schemes, mesh resolutions, convergence criteria, and radiative heat transfer. Two commercially available CFD software packages were used, ANSYS Icepak for preprocessing and FLUENT for solution and postprocessing. The road map was applied to three different heat sinks modeled into the full chassis. Numerical results were compared with the available experimental data and they were in good agreement.

Keywords: CPU cooling, computational fluid dynamics, conjugate heat transfer, forced convection

1. INTRODUCTION

CFD is gaining popularity especially as a decision support tool for the product design process. Even a decade ago, commercial CFD software packages were rarely used for heat transfer analyses, owing to a lack of tools or a lack of details for heat transfer in them. Especially, analyzing conduction and radiation together with convection was not possible. Currently, all popular CFD software packages have tools for handling coupled conduction, convection and radiation problems.

CFD software companies try to expand the applicability of their software to as many fields as possible. As a result, the software gets more and more complex. It may include many options that are useful for a particular application but useless and confusing for others. Users may take a long time learning and effectively using it.

Most of the commercial CFD packages in the market today include supplementary tools for electronics cooling applications. As the software gets more complex, these front-end or preprocessing tools become more useful in modeling. They have many modules to model most of the common components in electronics packages. One can easily construct a model of an electronics package combining offered component modules. The seemingly simplified and quicker modeling process sometimes may have hidden weaknesses that the user may not be aware of. The underlying complexity and sometimes the overly simplified modeling process increase the need for road maps for specific applications. In this study, taking a common electronics cooling application, we try to draw such a road map. The selected application is the CPU cooling of a standard desktop PC. The model is a full chassis model including every important component inside the chassis.

2. LITERATURE

Moffat (2002) claims that the flow and heat transfer situations encountered in electronics cooling applications are much more challenging than those in heat exchangers and as complex as those encountered in gas turbine blade cooling. Since it is almost impossible to get a detailed solution of the thermal and flow fields in a complicated electronic box, like a computer chassis, new efforts are emerging for thermal design of such systems. Although known for years, Computational Fluid Dynamics simulations have seldom been applied to the field of electronics cooling. Before the last

Received: 6 Feb. 2007; Revised: 27 Jun. 2007; Accepted: 23 Jul. 2007

A Preliminary version of this work appeared in Proceedings of ASME IMECE 2006, 5-10 November 2006, Chicago IL, USA.

decade, it was very expensive to perform CFD calculations. But with the introduction of high power workstations and personal computers, the cost of such computations has been drastically reduced (Behnia, Nakayama and Wang 1998).

Several researchers have worked on conjugate heat transfer in electronic systems via CFD. Among others, Yu and Webb (2001) simulated a complete desktop computer system which used an 80 W CPU. With the addition of other components (memory, chipset, AGP, PCI cards, floppy drives), a total of 313 W of heat was dissipated into the system. They solved the whole domain with a commercially available software, ANSYS Icepak. To decrease the complexity of their model, they modeled the CPU heat sink as a volume resistance having the same impedance as the detailed geometry. They improved the cooling of PCI cards with PCI side vents and baffle.

Biswas et al. (1999) also used ANSYS Icepak to study the airflow in a compact electronic enclosure. Their aim was to investigate the pressure loss due to the presence of the inlet and outlet grilles. They considered the use of fan curves obtained from the manufacturer since the fan curve might need to be modified if the fan was not closely ducted.

Argento, Joshi and Osterman (1996) not only studied system level electronic packaging thermal design computationally but also verified it experimentally. After the verification they worked on redesigning of an inlet plenum. Their implemented modification resulted in 56% reduction of the surface temperature.

Some relatively older studies used CFD for heat sink simulations only. Linton and Agonafer (1995) compared the results of detailed CFD modeling of a heat sink with experimental data. Then they presented a technique for representing the heat sink in a coarse manner for simulations that were less time-consuming. Their coarse model agreed well with the detailed model without losing the characteristics of the heat sink.

Sathyamurthy, Runstadler and Lee (1996) studied planar and staggered heat sink performance using FLUENT. Their computational results agreed well with the experimental ones. They found that the thermal performance of staggered fin configuration was superior over planar fin configuration. However the pressure drop requirements for the staggered fin heat sink was greater than those for the planar case.

Eveloy, Rodgers and Hashmi (2003) used Flotherm software to provide a perspective on the current capabilities of CFD as a design tool to predict component temperature on printed circuit boards. Their computations predicted the component operating temperature in an accuracy range of 3 ºC to 22 ºC, with up to 35% error. They suggested that component junction temperature would need to be measured experimentally when used for strategic product design decisions. They thought that the source of error was due to the turbulence models employed. They suggested using flow visualization in the early design phase to identify aerodynamically sensitive regions on the board, where temperature distributions should be handled with care.

The present study makes use of CFD for the conjugate heat transfer simulations in a whole computer chassis with an aim of drawing a road map based on experiences gained during the process. ANSYS Icepak is used for preprocessing and FLUENT is used for solution and postprocessing.

3. CFD SIMULATION APPROACH

The simulation approach for the full computer chassis model is discussed briefly in this section. The details related to the model and the software can be found in the work of Öztürk (2004) and the FLUENT 6.1 Users Guide, respectively.

3.1 Computer chassis model

The model studied consists of the following objects:

- **Computer chassis**
  It defines the computational domain. No mesh is generated outside the computer chassis. Some non-critical parts of the computer chassis are not meshed and modeled by hollow blocks.
- **CPU**
  It is the main heat source in the model. The CPU is modeled as a 2-dimensional area, which dissipates 70 W for each model. The CPU dimensions are equivalent to a commercially available CPU, i.e., AMD 2000+.
- **CPU Heat sink**
  The most important object in the computer chassis is the heat sink. Various models have been created for the investigation of cooling characteristics of different heat sinks. In every

model, only the heat sink geometry is changed; all the other objects remained unchanged. This is the most complicated object of all models; therefore it takes more time to create this geometry. Since there is no CAD geometry available, some models are created by measuring the dimensions of the actual heat sink and/or by using the dimensions given by the manufacturers.

- **CPU Fan**
  The fan is modeled as a lumped parameter model. It does not have geometrically defined blades. Only the fan curve is defined from which the operating point is calculated.
- **AGP**
  It is the graphics card, which is also a considerable heat source. Therefore it has a heat sink and a fan cooling it. AGP is modeled as a 2D source like the CPU.
- **AGP Heat sink and fan**
  It is a simple extruded aluminum channel heat sink. The heat sink is much smaller than the one on the CPU, since the heat dissipated by the AGP is much lesser than that dissipated by the CPU. A sucking fan with a linear fan curve is placed on the AGP heat sink.
- **Chipset**
  It is one of the main heat sources on the mainboard. A heat sink is mounted on the chipset, but there is no fan on it, making it a passive heat sink. However it does not work like a passive heat sink since the airflow in the chassis makes it forced cooled. Compared to the CPU, chipset can withstand higher temperatures.
- **Mainboard**
  It is the main card on which the CPU, chipset and other cards are placed. It is modeled with its thickness. The material of the mainboard is FR4 and copper.
- **RAM**
  Two RAM cards are placed on the mainboard. They are also heat sources and the spacing between these two cards is critical. This spacing consideration is beyond the scope of this study and therefore, a fixed, typical spacing is used for all models.
- **Other cards**
  Other miscellaneous cards are also modeled. Since there are a lot of very small electronic components on these cards, the cards are also heat sources. The details of these cards are not modeled but, for each case, heat dissipation is added and distributed over the whole card.
- **Power supply**
  Power supply is an important heat source in the case. Also it affects the air flow in the domain. However it is not feasible to model it exactly. Instead, lumped parameter models are used. The power supply with all the built-in components, cables and small openings on the outer walls are modeled as a single object, which is a resistance. The 3D fan placed behind it is the main fan of the computer case.
- **Grilles**
  The computer cases have small holes on them, which make air inlet or outlet possible. Since modeling these holes make the model computationally expensive, lumped parameter models are used again. They act like resistance to the flow according to the specified free area ratios.
- **Floppy and hard drives**
  CD-Rom, DVD-Rom, diskette drive and hard disk are also modeled. They are modeled as 3D blocks with heat dissipation. While they may be neglected with respect to their heat dissipation rates, they are included since they affect air flow in the system.

Fig. 1 shows the components of the chassis. The chassis is modeled using dimensions of a common ATX chassis (H×W×D = 444 mm × 424 mm × 187 mm), and all the components inside the chassis are standard-sized components that are used in most desktop computers. During modeling, exact dimensions of the components were either sourced from manufacturer specifications or obtained by measurement. Fig. 2 shows the detailed model of one of the heat sinks considered. Geometric details and measurements of the heat sinks were obtained from manufacturers.

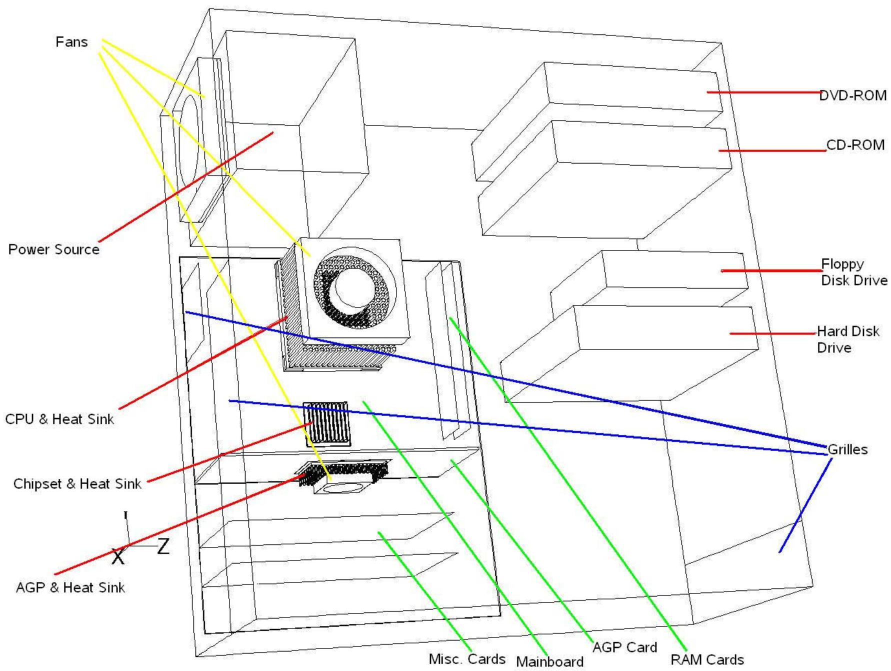

Fig. 1 Computer chassis model (H×W×D (in Y×Z×X directions) = 444 mm × 424 mm × 187 mm).

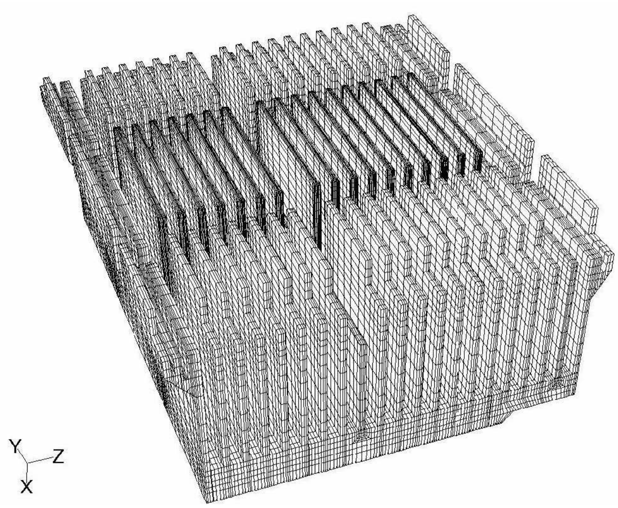

Fig. 2 Surface grid on one of the CPU heat sinks.

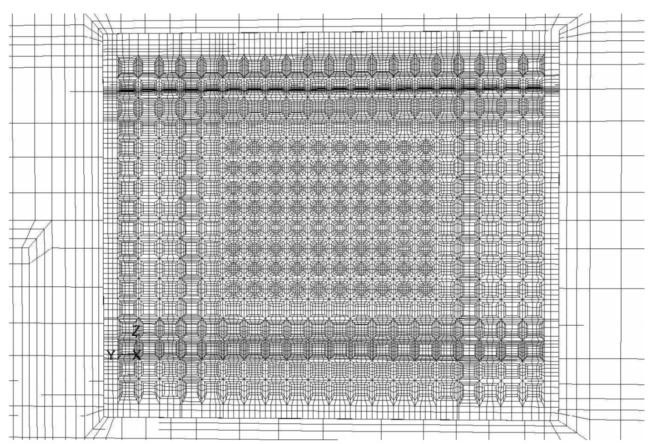

Fig. 3 Non-conformal grid interface.

3.2 Mesh generation

The mesh is the key component of a high quality solution. In our simulations hexahedral unstructured meshing is used. It is important to have a good mesh to have an accurate solution. There are some general guidelines called rules of QRST, standing for, Quality, Resolution, Smoothness and Total cell count. Mesh resolution is determined by showing grid independence with respect to coarser and finer meshing. Mesh quality is automatically determined by ANSYS Icepak considering face alignment, skewness, aspect ratio and cell size. Smoothness in mesh distribution is assured by keeping the growth rate of the cells from finer mesh to coarser mesh below 20%. Considering our computer resources, the total number of cells generated is kept around one million for the entire model.

Due to the complexity of the computer chassis, the fine mesh in and around the heat sink geometry cannot be carried out for the entire model and therefore, non-conformal mesh is used. It is the "hanging node mode" for which the nodes on the two sides of the non-conformal mesh interface do not match. Unless there is an abrupt change in the cell size, it is useful to use this kind of mesh to decrease the total cell count. Fig. 3 shows a cut section of the mesh in and around the CPU heat sink assembly. Mesh is fine inside the assembly where the heat sink locates.

The following is a set of procedures followed for meshing. It can be considered as a road map for meshing of similar electronic components.

- A first cut mesh is generated using the default coarse mesh parameters.
- Mesh quality is examined using surface plots and cut planes. It is important to have at least 4–5 elements on flow boundaries like grilles and fans. Also at least 3 elements should be placed between the fins of heat sinks. The smoothness of cell distribution and number of cells are the other issues checked.
- For all the cases it is seen that without using assemblies, which are virtual boxes whose inside and outside differ in mesh density, the number of cells would exceed the limit that the available computer resources can solve, even in the case when default coarse mesh parameters are used. This happens because of the detailed heat sink on the CPU. Therefore heat sinks are put in assemblies and the hanging node mode, i.e., non-conformal grid interface, is used for all the cases investigated.
- The default coarse mesh parameters are applied again to create a new mesh. These parameters make sure that there exist at least 2 cells in narrowest air gap and 1 cell across solid blocks. Again the mesh is examined for quality, resolution, smoothness and total cell count. This time, the mesh was far too coarse to resolve the flow features.
- Mesh is refined in regions of high velocity and thermal gradients. Maximum cell size in all directions is constrained to one twentieth of the domain size in order to avoid very large cells at the corners of the computer chassis.
- Highly skewed cells are generated in some models. These cells are generally created in the spaces between the heat sinks and the fans cooling them. The reason is that the fans are circular but the heat sinks are rectangular in fin shape, so a bad transition occurs from the fans to the heat sink in the limited space between them. Therefore additional virtual geometries are placed in those gaps. This is a way to increase the mesh quality in those regions since the user can control the meshing parameters for these additional geometries whereas it is impossible to control the default fluid mesh parameters in all the regions of the domain.
- When the mesh obeys the quality, resolution, smoothness and total cell count parameters, the boundary conditions and solution parameters can then be defined.

3.3 Governing equations

Time independent flow equations with turbulence are to be solved. The viscous dissipation term will be omitted. Therefore the governing equations for the fluid flow are the following forms of the incompressible continuity equations, Navier-Stokes equations—x-y-and z-direction momentum, and energy equation together with the equation of state:

$$\nabla.(\rho \vec{V}) = 0 \tag{1}$$

$$\nabla.(\rho u \vec{V}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \tag{2}$$

$$\nabla.(\rho v \vec{V}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + S_{My} \tag{3}$$

$$\nabla.(\rho w \vec{V}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + S_{Mz} \tag{4}$$

$$\nabla.(\rho h_0 \vec{V}) = -p\nabla.\vec{V} + \nabla.(k_{eff}\nabla T) + S_h \tag{5}$$

$$p = \rho R T \tag{6}$$

where (u, v, w) are components of fluid velocity $\vec{V}$ in (x, y, z) directions; ρ is density; p is pressure; T is temperature, h_0 is total enthalpy and R is ideal gas constant; S and τ are directional body force and shear stresses.

In the energy equation, Eq. (5), effective thermal conductivity is defined as $k_{eff} = k + k_t$ where k_t is turbulent conductivity term.

To handle turbulence, Reynolds averaging technique is employed. In Reynolds averaging, the solution variables are decomposed into mean and fluctuating components. For the velocity components, $u = \overline{u} + u'$, where $\overline{u}$ and u' are the mean and fluctuating velocity components for x-direction. Likewise, for pressure and other scalar quantities: $\phi = \overline{\phi} + \phi'$, where ϕ is a scalar such as pressure or energy.

The Reynolds Averaged Navier-Stokes (RANS) equations are solved together with Boussinesq approximation.

3.4 Boundary conditions

Since Navier-Stokes equations are solved inside the domain, no-slip boundary condition is applied to all walls in the domain. Therefore, at all surfaces $u = v = w = 0$.

It is assumed that the system fan does not drive a flow cell around the computer chassis and the heat transfer mechanism at the chassis outer walls is natural convection. Heat transfer coefficients at the outer walls are estimated from the correlations. In order to use the correlations, the average wall temperature must be prescribed. To do that, a first cut analysis must be run. As the typical values of the natural convection heat transfer coefficient lie between 2 and 25 W/m²K, a value of 5 W/m²K is selected to be the heat transfer coefficient at the computer chassis walls. The analysis results by taking ambient temperature as 30 °C give an average wall temperature of 36 °C at the walls and then heat transfer coefficients are calculated using this value and the available correlations with definitions of Rayleigh and average Nusselt numbers as:

$$Ra_L = Gr_L Pr = \frac{g\beta(T_s - T_\infty)L^3}{\nu\alpha} \quad \text{and} \quad \overline{Nu}_L = \frac{hL}{k}$$

where L is the characteristic length; h, k, g, β, ν and α are convection heat transfer coefficient, fluid thermal conductivity, gravitational acceleration, volumetric thermal expansion coefficient, kinematic viscosity and thermal diffusivity, respectively. T_s and T_∞ are the surface and ambient temperatures. Here, Ra is less than 10^9, for both vertical and horizontal surfaces, therefore the flow is laminar. For laminar natural convection over both vertical top surface and horizontal side surfaces of the chassis, two heat transfer coefficients (one for vertical and one for horizontal directions) are calculated from empirical correlations. Using correlations for laminar natural convection on the vertical plate, the thermal conductivity of air is taken as $k = 27\times10^{-3}$ W/mK and heat transfer coefficient $h \approx 3$ W/m²K. For the horizontal top plate, the Rayleigh number is calculated as 1.5×10^5 where the characteristic length is calculated from *L=A/P*, where *A* is the plate surface area and *P* is the plate perimeter. Nusselt number for the corresponding Rayleigh number is defined as $\overline{Nu}_L = 0.54 Ra_L^{1/4}$ which gives $h \approx 0.05$ W/m²K. The calculated heat transfer coefficients are applied to all exterior walls of the chassis except the bottom horizontal wall which sits on the ground.

3.5 Convergence issues

Only a well converged, well posed and grid independent simulation can give reliable results. Convergence is determined by the order of magnitude of the residues. Two different convergence tolerances are compared, one is 10^{-3} for flow and 10^{-6} for energy, and the other is 10^{-4} for flow and 10^{-7} for energy. Running the solver such that residuals fall one more order of magnitude means that more iterations are done to improve the solution quality. It should be noted that convergence criteria must assure that the results do not change as the iterations proceed. A common way of implementing this is to monitor the changes in some scalar values like temperature as well as the residue monitors. When the scalar values stay at a certain number and do not change as the iterations continue, then it can be stated that the solution has converged. It was seen that this criterion was satisfied when the continuity and momentum residues fell below 10^{-4} and energy residue fell below 10^{-7} . Therefore, all the models use the convergence criteria of 10^{-4} for the flow variables and 10^{-7} for the energy.

3.6 Turbulence modeling

The default turbulence model of all calculations is Algebraic Turbulence Model. It is a zero-equation model and computationally least expensive since no extra equations are solved in addition to continuity, momentum and energy equations. However, in order to rely on the results that algebraic model gives, it should be validated with higher-order turbulence models. RNG k-ε model was used as a test case. The temperature distributions and velocity fields are compared. The results show acceptable agreement (Öztürk, 2004). Therefore, the Algebraic Turbulence Model is suitable for use. Using RNG k-ε model, which is a two-equation model, doubles the solution time.

3.7 Radiation effects

Alpha heat sink was analysed to investigate the radiation effects (Öztürk, 2004). Radiation heat transfer helped the Alpha heat sink cool by less than additional 0.5 K. Therefore, it is concluded that radiation could be ignored for forced cooling of CPUs.

4. RESULTS AND DISCUSSIONS

The simulations are done using the chassis model with three different commercial heat sinks. Details related to heat sink models, turbulence models, radiation modeling and grid independence studies can be found in the work of Öztürk (2004) and Öztürk and Tari (2005). Only the part of the results that are relevant to the final computations are presented here.
The first group of results is for temperature distributions of all three heat sinks for the same conditions and same model. The results are obtained by changing the heat sink model while keeping the rest of the computational domain the same. The second group of results is for comparison with available experimental data.

4.1 Temperature distributions

For the three heat sinks considered, the temperature distributions are shown in Fig. 4.
It is obvious from Fig. 4 and Table 1 that Alpha heat sink outperforms the other two. The main reason for this is that it is a bigger heat sink with more heat transfer area. Evercool performs better than Coolermaster. Although the heat sink dimensions are similar for these two heat sinks, Evercool has an embedded copper base which enables higher conduction rates and heat is conducted to the whole heat sink in a more efficient way. For all heat sinks, it can be stated that the centre of the heat sink is the hot spot since the heat source corresponds to the proximity of the base centre. The fans installed on the heat sinks are identical in dimensions and fan curve. The fans have hubs where air cannot pass through and this makes the centre hotter. In the current simulations, the swirl of the fan is not modeled since the fans are lumped parameter models. In reality, with the fan swirling, the centre would not be as hot as the simulations predicts.
Heat dissipation rates of modeled components are given in Table 2.

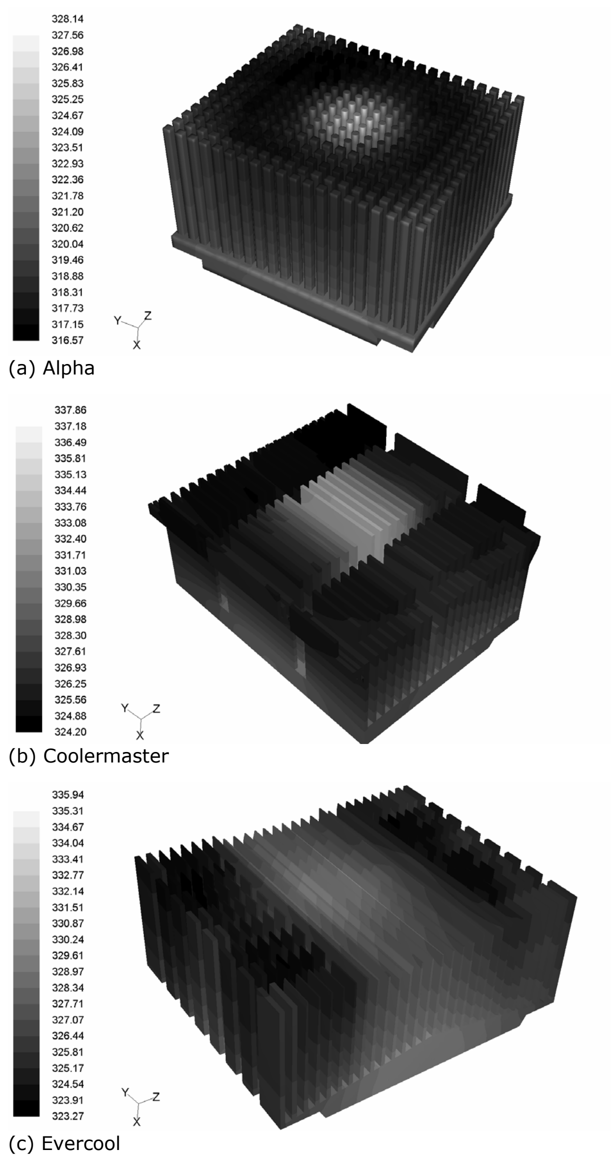

Fig. 4 Temperature distributions (K) on different CPU heat sinks (70 W CPU case).

Table 1 Maximum and minimum temperatures for the three heat sinks.

	Alpha	Coolermaster	Evercool
Tmax (K)	328	338	336
Tmin (K)	316	324	323
ΔT	12	14	13

Table 2 Heat dissipation of modeled components.

Object Name	Material	Heat Dissipation Rates (W)
CPU	Silicon	70
AGP	Silicon	25
CD	Al	10
DVD	Al	10
Hard drive	Al	20
Floppy drive	Al	-
Chipset	Silicon	10
CPU heat sink	Al-Cu	-
AGP heat sink	Al	-
Chipset heat sink	Al	-
Power supply	Porous	75
Memory cards	FR4	6x2
Misc. cards	FR4	10x2
Mainboard	FR4	-

4.2 Comparison with experimental data

There were some experiments in the literature that have been conducted on CPU heat sinks. Among them, the data obtained by Frostytech (2004) (experimental results in Table 3) are used for comparison. Their test setup is not the whole computer chassis system but some smaller domain in order to simplify the experiments. They have prepared a copper block to install the heat sink over and heated the block with two different heat loads, 50 W and 100 W. Then the rise above ambient temperature values was recorded. In our simulations, average temperature that the fan blows is calculated. This value is used as the corresponding ambient temperature of the test setup. Since the test setup is an open domain, the ambient temperature is the temperature of the air blown on to the heat sink. However in our simulations, ambient temperature is the temperature outside the domain, so the temperature of air blown by the CPU fan is considerably higher than the outside temperature. This necessitates the calculation of average temperature at the fan exit.

Table 3 Comparison of experimental (Frostytech, 2004) and numerical results. Errors are relative to experimental results.

		Alpha	Coolermaster	Evercool
50 W Heat Load	ΔT Exp.	12.7	23.3	19.1
	ΔT Num.	13.1	22.8	17.4
	Error	-3.1 %	2.1 %	8.9 %
100 W Heat Load	ΔT Exp.	25.4	34.4	38.5
	ΔT Num.	27.2	33.9	35.3
	Error	-7.1 %	1.5 %	8.3 %

Although the comparison was made quantitatively in Table 3, it would better be considered as a qualitative one. For a good quantitative comparison, the test setup should be modeled precisely. However, most of the heat sinks are shipped with their specific fans already installed. These fans have slightly different rotational speeds, blade geometries and dimensions than the ones used in this study. Some heat sinks are designed for low thermal resistance whereas some are designed for low noise. Therefore although the test setup is modeled correctly, different fans specifications make it meaningless to compare the heat sink performance. In our models identical fans were used for all three heat sinks. It was impossible to model the experimental setup of Frostytech for two reasons: i) it was an open setup and therefore, air flow inside the room was uncontrolled and unknown to the authors; ii) the specifications of the hot plate that was used in place of the CPU in their experiments were unavailable. Their experimental results cannot be considered as scientific because details of the experimental conditions and uncertainties of measurements are not provided. Nevertheless, it was clear from both their experiments and our analyses that Alpha outperforms the other two heat sinks.

5. ROAD MAP AND CONCLUSIONS

In this study, CPU cooling has been investigated in a complete computer chassis with different heat sinks and the performance of the three heat sinks is compared. The comparison of the heat sink temperature difference results were made with the available experimental results. The results showed agreement with the experimental data.

While performing the CFD simulations, considerable insight was gained for drawing the following road map:

- Mesh should be generated such that 3 cells in the narrowest air gap and 2 cells in conduction zones are enough. Unnecessarily finer mesh distorts the stability of the model with the penalty of larger run times. The reduction in stability for finer mesh in FLUENT solution is due to the slower convergence of Gauss-Seidel in error reduction in multi-grid mesh. This can be prevented by increasing the multi-grid levels with an associated extra computational cost (FLUENT 6.1 Users Guide).
- Non-conformal mesh can be used so that the fine mesh parameters are stored inside the non-conformal mesh interfaces where heat sinks are located. This prevents the generation of an excessive number of cells which increases the computational cost. If the analysis is going to be running on a single computer instead of a parallel cluster, non-conformal mesh is inevitable.
- First-order discretization scheme is enough if the grid is dominated by hexahedral cells. Convergence is faster with this scheme.
- Zero-equation turbulence model is suitable for analysing fluid flow and heat transfer in a computer chassis. Higher-order turbulence models are unnecessarily expensive.
- Radiation effects can be ignored because of the domination of forced convection and relatively low temperature differences inside the chassis.
- Convergence must be assured by checking the residuals and also the temperature monitors. It is necessary to let the residuals to drop more than three orders of magnitude.
- Experimental verification is important for CFD analyses. Verification of a base model can be a reference for the consequent simulations.

Although this road map is drawn specifically for our computer chassis model, it can be further generalized for electronics boxes of similar size. Together with the mesh generation approach presented in section 3.2, it can be used as a guide especially for FLUENT or in general for finite volume CFD simulations.

REFERENCES

1. Argento CW, Joshi YK and Osterman MD (1996). Forced convection air-cooling of a commercial electronic chassis: an experimental and computational case study. *IEEE Transactions on Components, Packaging, and Manufacturing Technology—Part A* 19(2):248–257.
2. Behnia M, Nakayama W and Wang J (1998). CFD simulations of heat transfer from a heated module in an air stream: comparison with experiments and a parametric study. *Proceedings of 6th I-THERM*. 27–30 May 1998, Seattle WA, USA, 143–151.
3. Biswas R, Agarwal RB, Goswami A and Mansingh V (1999). Evaluation of airflow prediction methods in compact electronic enclosures. *Proceedings of 15th SEMI-THERM*. 9–11 March 1999, San Diego CA, USA, 48–53.
4. Eveloy V, Rodgers P and Hashmi MSJ (2003). Numerical prediction of electronic component heat transfer: an industry perspective. *Proceedings of 19th SEMI-THERM*. 11–13 March 2003, San Jose CA, USA, 14–26.
5. FLUENT 6.1 Users Guide.
6. Frostytech (2004). Heat Sink Reviews. June 2004. www.frostytech.com.
7. Linton RL and Agonefer D (1995). Coarse and detailed CFD modelling of a finned heat sink. *IEEE Transactions on Components, Packaging, and Manufacturing Technology—Part A* 18(3):517–520.
8. Moffat RJ (2002). Getting the most out of your CFD program. *Proceedings of 8th I-THERM*. 30 May–1 June 2002, San Diego CA, USA, 9–14.
9. Öztürk E (2004). *CFD Analyses of heat sinks for CPU cooling with FLUENT*. M.S. thesis, Middle East Technical University, Ankara, Turkey.
10. Öztürk E and Tarı I (2005). CFD Analyses of heat sinks for CPU cooling with FLUENT. *Proceedings of ASME 2005 Summer Heat Transfer Conference*. 17–22 July 2005, San Francisco CA, USA.
11. Sathyamurthy P, Runstadler PW and Lee S (1996). Numerical and experimental evaluation of planar and staggered heat sinks. *Proceedings of 5th I-THERM*. 29 May–1 June 1996, Orlando FL, USA, 132–139.

12. Yu CW and Webb RL (2001). Thermal design of a desktop computer system using CFD analysis. *Proceedings of 17th SEMI-THERM*. 20–22 March 2001, San Jose CA, USA, 18–26.