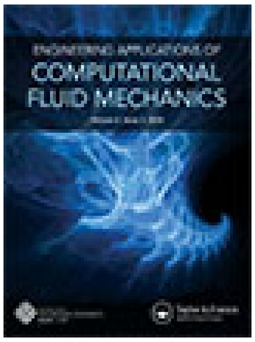


Taylor & Francis
Taylor & Francis Group

Engineering Applications of Computational Fluid Mechanics

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

Numerical Investigation on Cooling of Small form Factor Computer Cases

Ömer Emre Orhan & Ilker Tari

To cite this article: Ömer Emre Orhan & Ilker Tari (2008) Numerical Investigation on Cooling of Small form Factor Computer Cases, Engineering Applications of Computational Fluid Mechanics, 2:4, 427-435, DOI: 10.1080/19942060.2008.11015242

To link to this article: https://doi.org/10.1080/19942060.2008.11015242

Published online: 19 Nov 2014.

Engineering Applications of Computational Fluid Mechanics Vol. 2, No. 4, pp. 427–435 (2008)

NUMERICAL INVESTIGATION ON COOLING OF SMALL FORM FACTOR COMPUTER CASES†

Ömer Emre Orhan* and Ilker Tari**

** Alstom Hydro Power, Armada Is Merkezi A Blok Kat:19, Ankara 06520, Turkey*
*** Middle East Technical University, Mechanical Engineering Department, E206, Ankara 06531, Turkey*
E-Mail: itari@metu.edu.tr (*Corresponding Author*)

ABSTRACT: In this study, cooling of small form factor computers is numerically investigated. The problem is a conjugate heat transfer problem in which ambient air is the final heat transfer medium. In modeling the problem, heat transfer using heat pipes running from the CPU to the heat exchanger in the back end of the chassis, forced convection inside the chassis, ventilation of the chassis air, conduction paths inside the chassis, and natural convection from the chassis walls to the ambient air are considered. The numerical model is analyzed using a commercial finite volume computational fluid dynamics software package. The effects of grid selection, discretization schemes and turbulence model selection on simulation results are discussed. In addition, recirculation and relaminarization are addressed briefly as examples of physical phenomena affecting cooling. For a comparison with the computational fluid dynamics results, an experiment is conducted and some temperature measurements are obtained from critical locations inside the chassis. The computational results were found to be in good agreement with the experimental ones.

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

1. INTRODUCTION

Small Form Factor is a general term which implies that this form factor (computer case and motherboard size) is smaller than the standard form factors like micro-ATX and BTX. The size of a typical SFF computer is as big as a shoebox with a square front profile. The primary advantage of SFF computers is its size. They are used in areas where space limitation is a major concern. However, due to the small size, they cannot be cooled as easily as the regular systems are. Generally, SFF computers are cooled using heat pipe systems. Investigation of a common SFF cooling scenario using CFD is the objective of this work.

In this study, two SFF computers, Shuttle SK21G and Shuttle SS59GV2, are used. Detailed full chassis computational models are generated in Icepak™ and simulations are performed. Also, some temperature measurements are taken for comparing with the CFD results.

2. LITERATURE

CFD is a very powerful tool in the sense that qualitative information and quantitative data can be obtained if it is complemented by experiments. The amount of data that one can obtain from experiments is limited, but CFD is very useful in offering a large set of data. Therefore, CFD can be used to minimize the number of experiments and design alternatives. Previous works with CFD show that there have been successful applications of CFD for design purposes in electronics cooling field.

Tucker (1997) compared several commercial and non-commercial CFD codes related to electronics cooling. General thermo-fluid capabilities, user friendliness and peripheral aspects were checked and for all comparisons with measurement, agreement was found to be within 30%.

Yu and Webb (2001) used Icepak™ to analyze the flow and heat transfer inside a desktop computer which had an 80 W CPU. The design was for a total chassis power of 313 W. In their model, motherboard, PCI/AGP cards and memory were modeled as zero thickness rectangular plates with heat generated uniformly on the component side. The HDD and DVD were modeled as solid blocks generating a specified amount of heat inside the volume. The power supply and the CPU heat sink were modeled as a volume resistance. In their study, the key design parameter to minimize was chassis air flow.

† A Preliminary version of this work appeared in Proceedings of ASME IMECE 2007, 11–15 November 2007, Seattle WA, USA.

Received: 29 Jan. 2008; Revised: 17 Apr. 2008; Accepted: 2 May 2008

Yeh (2002) employed CFD to check the feasibility of a proposed cooling scheme that used a fan card to cool the printed circuit boards within a rack. A commercial CFD package, Ideas-ESC, was used for the analysis. The main idea of the cooling scheme was to utilize cooling fans to keep the boards at a specific temperature. Since the air flow through the printed circuit boards were unknown, CFD analysis was necessary.
With advancement in processor technology, processor powers increase considerably. Roknaldin and Sahan (2003) discussed heat sink optimization for the final design process in which heat sink space and the cost of thermal solution are big concerns. They modeled a 1U server including inlet/exhaust vents, hard drives, blower/fans, power supply, boards and components, heat sink and PCI cards using Icepak™. They worked on two new strategies for heat sink optimization and managed to obtain a cheaper solution for a cooler interior.
Icepak™ was used to simulate a compact two phase cooling system for a laptop computer by Ali (2004). In this study, two alternative cooling systems for a laptop computer were designed. Some parameters that affected the thermal performance of the system like fill ratio of the coolant, initial system pressure and pump flow rate were examined using CFD. Between two choices, the better one is identified using the CFD results.
The work of Lin and Huang (2002) is related to generating a small forward-curved fan for the cooling management of laptop computers. This is an integrated study including fan design, mockup manufacturing, experimental verification and numerical simulation. The fan performances were verified by both experimental and numerical approaches. A commercial code for the numerical simulations, STAR-CD was used and numerical results were in good agreement with the experimental ones. The flow rate was found to be within an error range of 2%–6%.
Öztürk (2004) and Öztürk and Tari (2007) investigated the forced cooling of heat sinks mounted on CPUs. Thermal parameters such as heat sink effectiveness, turbulence models, radiation and geometry of heat sinks were analyzed using commercial CFD programs, Fluent and Icepak™. Some improvements on heat sinks were suggested and tested. After carrying out CFD simulations, the results were found to be in good agreement with the experimental values.
Due to its small size, cooling of an SFF computer is more challenging compared to that of other computer cases. However, in the literature, there is a lack of detailed information about the cooling of SFF computers. Hence, the main motivation of this study is to understand the cooling phenomena inside SFF computers using CFD.

3. CFD SIMULATION APPROACH

Detailed models of the selected SFF systems and related modeling details are presented in this section. Special attention is paid to include in the models as much detail as possible and to assign the right power dissipation values to the included components.

3.1 Modeling

After the selection of the parts in the computer chassis that will be modeled such as hard disk, DVD-ROM and heat pipe, the dimensions of these parts are carefully measured using a caliper. A photograph of SK21G together with the measured outside dimensions can be seen in Fig. 1.

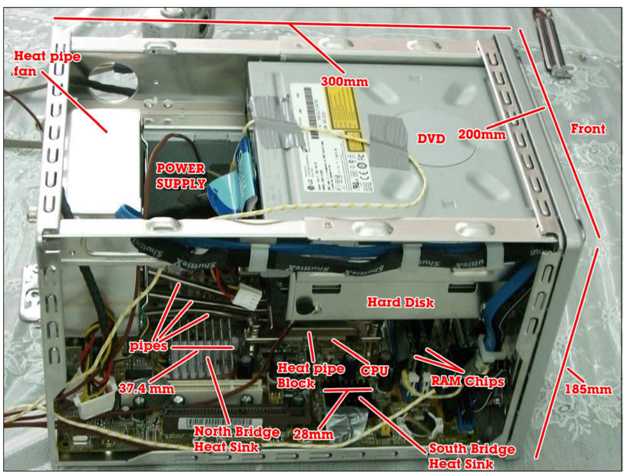


Fig. 1 Shuttle XPC SK21G used in this study.

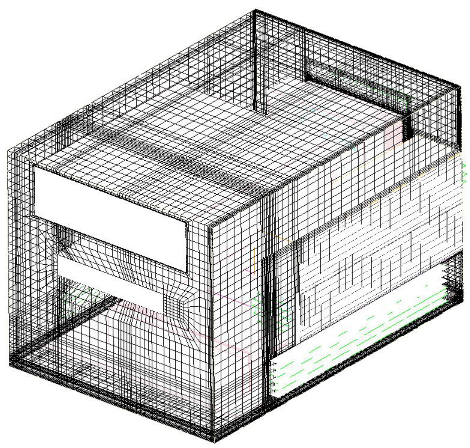


Fig. 2 Computational grid of the exterior of the chassis.

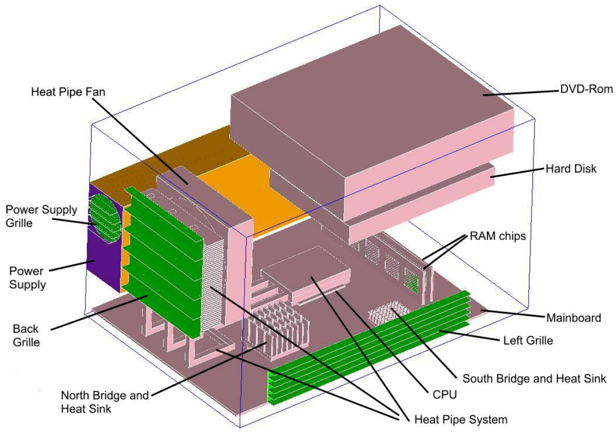


Fig. 3 Model of SK21G in Icepak™.

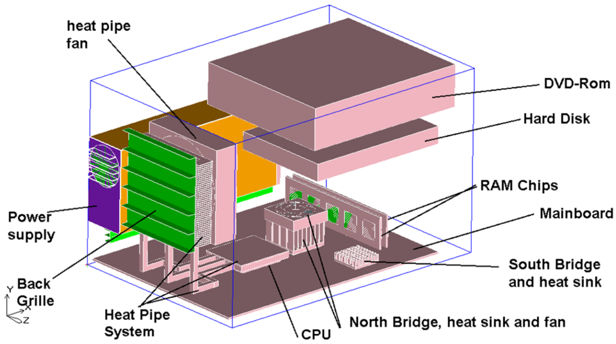


Fig. 4 Model of the SS59GV2 in Icepak™.

In this study, the computational domain is taken as the whole computer chassis. Although there are natural convection effects outside the chassis, these effects are taken into account using the lumped parameter modeling approach of Icepak™ and a suitable heat transfer coefficient is assigned for the outside surfaces of the chassis walls. Inside the chassis, only the parts that have major effect on the heat transfer, such as the hard disk, CPU, chipset and RAM, are modeled. However, components like transistors, capacitors, resistors, cables, etc. are neglected.

The density and quality of the computational grid is an important issue. The density of the mesh is increased in the areas where the resolution of the flow is important. However, a non-conformal computational grid is created in order to be able to decrease the computation time as much as possible. Finally, a high quality, unstructured hexahedral mesh is generated before the characteristic equations are solved and converged. Fig. 2 shows the mesh around the computer chassis.

Computational domain is taken as the 3D computer chassis in this study. CPU, CPU heat pipe, chipset (including north bridge and south bridge and their heat sinks), power supply, RAM, hard disk, DVD-ROM, mainboard, fans and grilles are modeled. Details of the computational domain for SK21G and for SS59GV2 can be seen in Figs. 3 and 4, respectively. Since both of them are G-type Shuttle chassis, outer chassis dimensions of both systems are the same. However, there are some major differences in motherboard components, CPU and heat pipe system.

With the assumption that chassis fan cannot create a convection loop outside of the chassis, the outside ambient air is not modeled and the thermal boundary conditions for the chassis walls are set by defining their convection heat transfer coefficients. The natural convection heat transfer coefficient values for the side walls and the top surface of the chassis were estimated to be 3 and 4.2 W/(m^{2}K) by using the laminar average Nusselt number correlations in the literature. The calculations were performed by taking ambient air temperature as 28°C and iteratively determining the wall temperature as 34°C (details are presented in Orhan (2007)). The bottom of the chassis is defined as an adiabatic surface, since it is touching or very close to its support.

3.2 Heat pipe system

Heat pipes are passive devices that transport heat from a heat source (evaporator) to a heat sink (condenser) via the latent heat of evaporation of a working fluid. The heat pipe system of SK21G is modeled as seen in Fig. 5. The heat pipe is a phase change device; evaporation at the CPU end and condensation at the heat exchanger (a finned heat sink) end are the important concerns. However, due to the difficulties associated with phase change phenomena modeling, in the case of electronics cooling applications, investigators prefer to model heat pipes as highly conducting solid parts. The major advantage of having a heat pipe is to achieve high heat transfer rates between its ends. Hence, heat pipes in the system are modeled as solid blocks with very high conductivities in the main heat transfer direction. This conductivity should be large enough to result in the same heat rate as that taken by the boiling heat pipe fluid. These conductivity values can be taken typically between 20,000 and 50,000 W/m·K. In this model, this conductivity value is chosen as 20,000 W/m·K. This value was also checked, indirectly, during the performed experiments. The conductivity values in the other directions are taken as the conductivity of the heat pipe material (aluminum) which is 300 W/m·K. In addition to the pipes of the system, there are also very thin aluminum plate fins (heat exchanger) where condensation takes place. These 44 plate fins (measured 90 mm $\times$ 17 mm) are 0.4 mm thick and packed parallel to each other with a 1.45 mm spacing between them to provide enough area for heat transfer. The chassis fan of 90 mm diameter blows chassis air onto the heat sink plates of the heat pipe system which is defined inside the chassis boundary. The fan curve is obtained from the manufacturer specifications and is entered into the preprocessing step of Icepack by defining a 3-D fan model. This fan is a dual purpose fan that also circulates the air inside the chassis and ventilates the chassis. SS59GV2 heat pipe system is a similar one with shorter pipes.

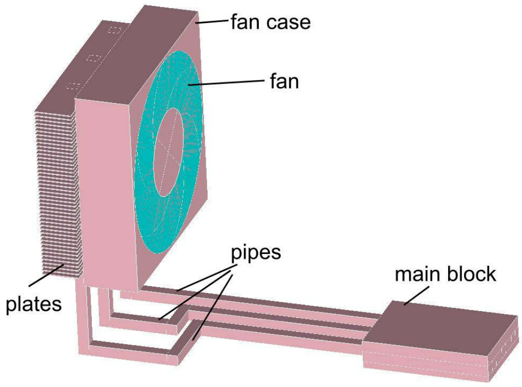


Fig. 5 Details of the heat pipe system of SK21G (as modeled).

3.3 Governing equations

The governing equations of the flow are modified according to the conditions of the simulated case. Since the problem is assumed to be steady state with low velocities, time dependent parameters, together with the viscous dissipation term, are dropped from the equations. The resulting equations are:

- Conservation of mass:

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

- x-momentum:

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

- y-momentum:

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

- z-momentum:

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

- Thermal energy:

$$\nabla.(\rho e\vec{V}) = -p\nabla.\vec{V} + \nabla.(k\nabla T) + \dot{q} \tag{5}$$

where u, v and w are the components of velocity $\vec{V}$ in x, y and z directions; f and τ are body force and shear stress components; $\dot{q}$ is heat generation rate; and p, T, ρ and k are pressure, temperature, density and thermal conductivity of the fluid, respectively.

After defining all of the major component geometries in the chassis according to their precise measurements, no slip boundary condition is applied to all of the surfaces together with the thermal boundary conditions discussed in Section 3.1. The grills or the array of small holes on the side surfaces are defined with their free area ratios to allow air circulation through them.

Since the flow in this study is turbulent, turbulence effects should be taken into account using turbulence modeling.

Table 1 Power budget and materials of the components.

Object Name	Material	Heat Dissipation Rates (W)	
		SK21G (AMD)	SS59GV2 (Pentium)
CPU	Package	25	50
DVD	Al	5	5
Hard drive	Al	15	15
North Bridge	Package	2.5	3.5
South Bridge	Package	1.25	1.5
North Bridge heat sink	Al	–	–
South Bridge heat sink	Al	–	–
Heat pipe system	Al-Cu	–	–
Power supply	Porous	25	30
Mainboard	FR4	–	–
DIMM	FR4	0.5×2	0.5×2

3.4 Heat dissipation rates of components

Heat dissipation rates of the components inside the chassis are given in Table 1. The power ratings of the CPU, north and south bridge chips are obtained after an iterative process since the real heat dissipation values of these components are not known to anyone other than the manufacturer. The heat dissipation of the power supply is also assigned after some discussions and a literature survey (details are in Orhan (2007)). These values are assigned by comparing the experimental temperature measurements with the CFD results. Power ratings of the DVD-ROM and the hard drive are taken from the study of Öztürk (2007). During normal operation of the computer, both the DVD-ROM and the hard drive do not dissipate their rated maximum values of heat. To investigate the sensitivity of the results to these heat dissipation values, we established a case in which both of these values were taken as zero and we observed that the effect on the rest of the chassis components and the flow was negligible (Orhan, 2007).

4. RESULTS AND DISCUSSIONS

In this section, the results of the CFD analyses are presented and compared with the experimental results. In CFD calculations, there are several sources of computational errors. One of them is caused by the adopted discretization method. Others result from the incorrect choices of boundary condition definition and turbulence model. Since boundary conditions and heat dissipation rates should closely represent the real case conditions, it is very important to assign them as accurately as possible. In addition, the selected turbulence modeling involves some errors since it is not possible to model turbulence exactly. Therefore, selecting the right turbulence model is very important for minimizing the errors. Other than computational error, convergence and associated tolerances are also important concerns. There should be convergence criteria to decide where to stop the solution. In Icepak™ convergence is decided by checking the residuals. In order to obtain an accurate solution, enough number of iterations should be run and the residual of a variable should fall below some predetermined value. This value is assigned by the user and in this study, it is chosen as 10^{-4} for the continuity and momentum equations and 10^{-7} for the energy equation. However, residual checking is not enough to decide whether a solution has converged or not. Several critical locations in the computational domain should be monitored for velocity, pressure and temperature. These monitors are also indicators of convergence if they do not change significantly for successive iterations.

After the modeling phase is completed, a mesh selection should be made. For this, three different computational grids were generated: coarse, base and fine. The results of these three computational grids were compared; since there was no significant difference between the fine and base grids, the base grid was used as a basis for all simulations (Orhan, 2007).

The next aspect to consider is the application of different discretization schemes. There are several discretization schemes available, and in this study only first and second order upwind discretization schemes are considered. The results from both schemes were compared and no significant difference was found, therefore first order discretization is selected (Orhan, 2007).

Base grid, together with first order discretization turbulence model, is adopted in this study. Since the flow inside the chassis is mildly turbulent, comparison of different turbulence models should be considered. Other than the zero equation model, k-ε turbulence model was employed and the comparison of the results showed that zero equation model is sufficient for this study (Orhan, 2007).

Recirculation and relaminarization affect the heat transfer and the characteristics of the flow. The flow structure should be examined to identify any zones where these might occur. Relaminarization is not observed in any candidate locations such as the space between the north bridge heat sink fins. There are several observed recirculation zones as shown in Fig. 6. Detailed particle traces in these recirculation zones are given in Figs. 7 and 8. By making small changes in chassis configuration these recirculation loops may be avoided. This example also shows the advantage of having a very detailed flow field simulation in a CFD study, in contrast to the experimental results that are limited to instrument locations.

The last step is the comparison of CFD results with experimental data. In this part, two experiments are conducted and temperatures are measured at specific locations inside the two Shuttle computers, SK21G and SS59GV2. There are four different temperature measurement locations for each of the computers. The results of the CFD analyses are also given in these four points for comparison purpose.

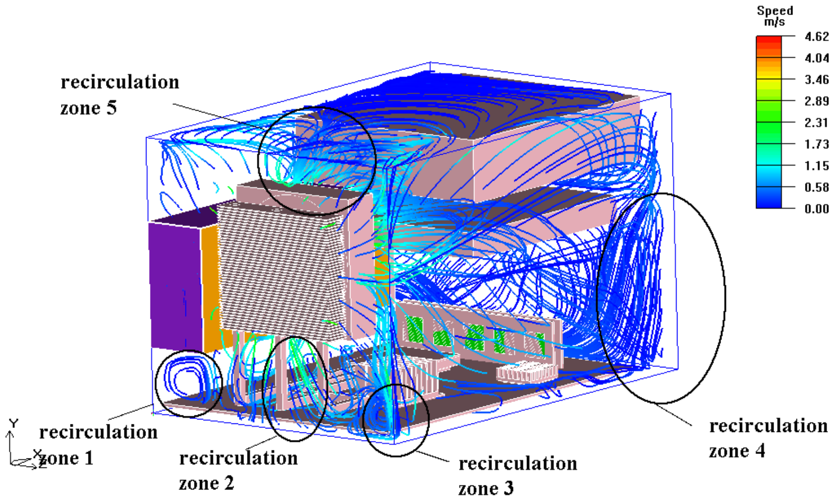


Fig. 6 General visualization of the flow field inside the computer chassis and the recirculation zones.

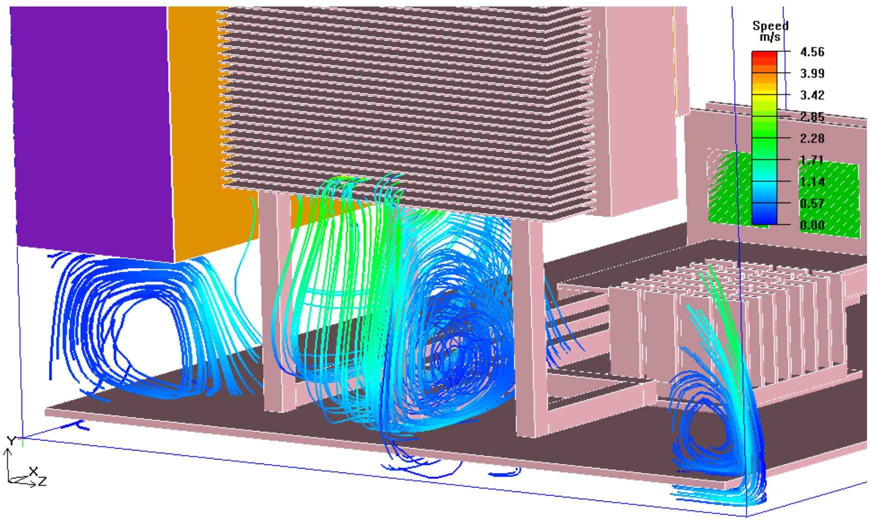


Fig. 7 Particle traces of recirculation zones 1–3.

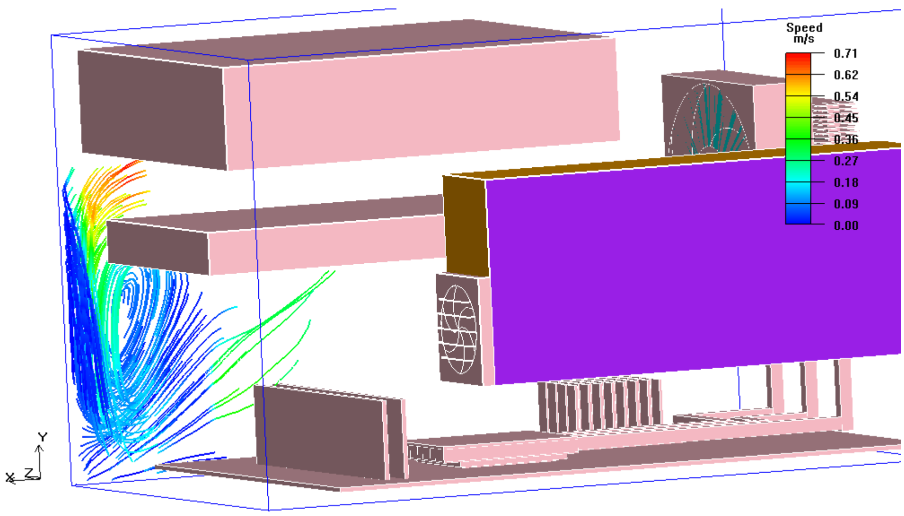


Fig. 8 Particle traces of recirculation zone 4.

The experiments are conducted for two different computer configurations. The first configuration is Shuttle SK21G with an AMD Sempron 3000+ processor. The other one is SS59GV2 with an Intel Pentium 4 3.2 GHz CPU. Mainly, the configurations are similar; however, there are some basic differences. Other than the CPUs, the chipsets are also different. Although the north bridge and south bridge heat sinks are almost the same, there is a fan at the top of the north bridge heat sink which makes it an active heat sink for SS59GV2. This is a major difference affecting the flow field inside the chassis.

The experiments are not conducted in fully controlled environments. Therefore, these experiments cannot be used to verify thoroughly the CFD results. However, they can give an idea on a comparative basis. Tables 2 and 3 are the comparisons of CFD results with experimental result for Shuttle SK21G and Shuttle SS59GV2. The comparison is based on the temperature measurements taken from four different locations inside the chassis. The thermocouple insertion locations for the CPU and south bridge heat sinks are shown in Fig. 9. The percent relative errors are calculated taking the experimental results as true values. The largest error is the calculated value for the heat pipe block of Shuttle SK21G. However, for nearly the same location on the heat pipe of Shuttle SS59GV2, the magnitude of the error differs. This may result from the modeling of the heat pipe system. Due to the geometric modeling limitations, the circular cross section of heat pipes is modeled with rectangular cross section blocks. Heat pipe in SK21G is longer than the one in SS59GV2. Therefore, modeling of pipes as rectangular solid blocks in Icepak™ brings out additional thermal resistance in SK21G compared to SS59GV2.

Table 2 Comparison of experimental results of Shuttle SK21G with CFD results for an ambient temperature of 28°C.

Point		Temperature Values, ΔT (°C)		
		ΔT Experiment	ΔT CFD	% Error
1	North Bridge Heat Sink	6.6	7	5.7
2	South Bridge Heat Sink	15.1	14	7.3
3	Heat Pipe Block	10.4	11.4	9.6
4	Power Supply	3.9	4.2	7.7

Table 3 Comparison of experimental results of Shuttle SS59GV2 with CFD results for ambient temperature of 28°C.

Point		Temperature Values, ΔT (°C)		
		ΔT Experiment	ΔT CFD	% Error
1	North Bridge Heat Sink	8.4	8.5	1.2
2	South Bridge Heat Sink	15.6	16.1	3.2
3	Heat Pipe Block	19.3	19.1	1
4	Power Supply	8.9	8.1	9

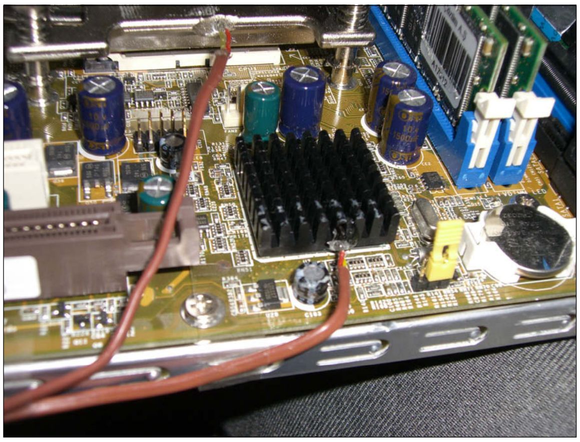
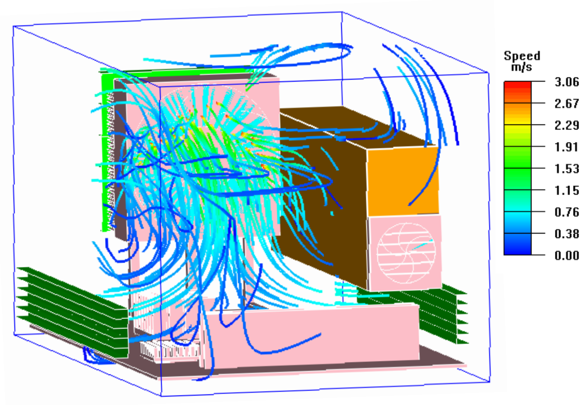


Fig. 9 Thermocouples inside the heat pipe block above the CPU and south bridge heat sink base.



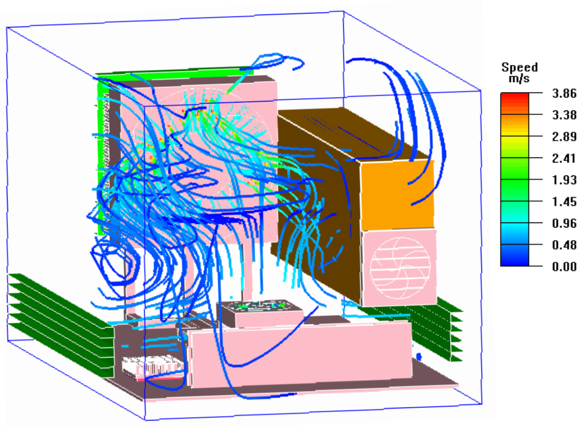


Fig. 10 Particle traces for SK21G (top) and SS59GV2 (bottom).

Although the power ratings of the power supplies are close, the temperatures on the power supplies are considerably different. This is an indication of dissimilar flow fields inside the chassis and around the power supplies as expected. Fig. 10 shows the particle traces originating from the same cut plane. The flow fields in the middle of the chassis are different, due to different arrangements of the motherboard components. The intensity and the velocity magnitudes of the particle traces around the power supplies are also different which affect the temperature distribution. Thermocouple attachments on the power supplies are not as reliable as the other attachment locations. Since it is difficult to attach the thermocouple to the outer surface of the power supply, temperatures are recorded with the thermocouple cable glued on the plate. Therefore, relatively large errors are obtained for the measurements on the power supply.

In regard of the temperature values, there is a considerable difference between the heat pipes. This is mainly due to the higher power consumption of the Pentium 4 CPUs. The power consumption of the Pentium 4 CPU is taken as 50 W, while the power consumption of the AMD Sempron 3000+ CPU is just 25 W. Although the power rating of the Pentium 4 CPU is twice that of the AMD Sempron counterpart, the system can cool it to reasonable temperatures. This is also an indication of the effectiveness of the cooling system.

5. CONCLUSIONS

This study shows that it is possible to simulate a very complicated model using CFD, although there are some limitations and assumptions involved. CFD is a very powerful tool in the sense that it minimizes the design time and cost effectively. It provides large amount of data that an experiment can never provide. It is especially useful for identifying design weaknesses noted in recirculation zones. Therefore, it is a complementary tool that needs to be used in different stages of engineering design. And this study is an example of how CFD and experiments can complement each other in resolving the cooling problem of an SFF computer. Since, in this study and in similar studies, heat pipe systems are modeled as directional high thermal conductors, the value of that directional thermal conductivity should be determined experimentally.

NOMENCLATURE

ATX	Advanced Technology Extended form factor
BTX	Balanced Technology Extended form factor
CFD	Computational Fluid Dynamics
CPU	Computer Processing Unit
HDD	Hard Disk Drive
SFF	Small Form Factor
T	temperature (K)
$\vec{V}$	velocity vector
f	body forces
h	convection heat transfer coefficient (W/m^2 K)
k	thermal conductivity (W/m·K)
p	pressure (Pa)
$\dot{q}$	heat generation rate (W/m^3)
t	time (s)
u, v, w	velocity components (m/s)
x, y, z	position coordinates
ρ	density (kg/m^3)
τ	shear stress (N/m^2)

REFERENCES

1. Ali AA (2004). *Design and Analysis of a Compact Two Phase Cooling System for a Laptop Computer*. M.S. thesis, Georgia Institute of Technology, Georgia, USA.
2. Lin SC, Huang CL (2002). An Integrated Experimental and Numerical Study of Forward Curved Centrifugal Fan. *Experimental Thermal and Fluid Science* 26(5):421–434.
3. Orhan ÖE (2007). *Numerical Investigation on Cooling of Small Form Factor Computer Cases*. M.S. thesis, Middle East Technical University, Ankara, Turkey.
4. Öztürk E (2004). *CFD Analyses of Heat Sinks for CPU Cooling with FLUENT*. M.S. thesis, Middle East Technical University, Ankara, Turkey.
5. Öztürk E, Tari I (2007). CFD Modelling of Forced Cooling of Computer Chassis. *Engineering Applications of Computational Fluid Mechanics* 1(4):304–313.
6. Roknaldin F, Sahan RA (2003). Cooling Solution for Next Generation High Power Processor Boards in 1U Computer Servers. *International Electronic Packaging Technical Conference and Exhibition*, July 6–11, 1–6.
7. Tucker PG (1997). CFD Applied to Electronic Systems. *IEEE Transactions on Components, Packaging, and Manufacturing Technology—Part A* 20(4):518–529.
8. Yeh LT (2002). A CFD Analysis of an Electronic Box. *ITherm 2002, Eighth Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems*, 30 May–1 June 2002, 710–717.
9. Yu CW, 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.