Ninth International Conference on Computational Fluid Dynamics (ICCFD9), Istanbul, Turkey, July 11-15, 2016

ICCFD9-0111

Thermal and Flow Field Analysis of Electronic Components Inside a Desktop Computer Chassis

Kutay ÇETİN[1], Kürşad Melih GÜLEREN[2]

[1]Faculty of Aeronautics and Astronautics, University of Turkish Aeronautical Association, 06790, Ankara, TURKEY
[2]Faculty of Aeronautics and Astronautics, Anadolu University, 26470, Eskişehir, TURKEY
Corresponding author: kmguleren@anadolu.edu.tr

Abstract: The reliability of electronic components inside a computer chassis is getting affected in a critical manner by temperature. The purpose of this study is to analyze the three-dimensional thermal dissipation process of a desktop computer which uses 85 W CPU by means of a CFD software package. Air cooling is used for heat generating and affected components such as the main chassis, CPU and GPU. The temperature distribution of the components and flow fields are investigated to understand the effectiveness of the cooling system. Based on CFD results a new design of component arrangement will be provided.

Keywords: Computational Fluid Dynamics, Thermal Analysis, Cooling, Computer Chassis

1. Introduction

Heat transfer has an important role in the design of electronics cooling and heating, ventilating, and air conditioning systems. Heat transfer also plays a significant role in many other applications, such as turbofan engines, heat exchangers, inhabiting, and even people. Thereby, the computer systems that used nowadays are tried to improve in terms of multi-tasking capacity and process speed. Herein, scientist and engineers encounters with a significant problem which is over-heating an electronic component. The main reason of temperature rising is very high voltages to get through of the electronic components. When the temperature reached the manufacturer's threshold temperature, system starts to give errors. Consequently, heat transfer inside a computer chassis and heat transfer between electronic components must be examined before moving on to application.

Air cooling is not very effective all the while. There are many ways that air cooling may fail such as weak flow distribution, poor mixing of cold and hot air flow or unexpectedly higher power generation. Computational fluid dynamics is a well approach for this kind of problems. Since the desktop computers are the most common computer type worldwide, ensuring these computer more energy efficient come out with micro and macro effects such as decrement in electric consumption which effect the climate change in a good way.

Many scientists have investigated the heat transfer across the computer or sub components. Kim et al. [1] used heat pipes while Yu and Webb [2] used ANSYS to simulate an entire computer chassis and in this study heat generated components such as central processing unit, graphics processing unit, floppy drives were included to the analysis. 313 W of heat was dissipated in this study to the computational domain. Dirker et al. [3] is used embedded solids while Amon [4] destabilize force convection of the fan flow. Choi et al. [5] has investigated the impact of turbulent models on convective heat transfer in a computer chassis and results are describing the effects of aspect ratio. Ozturk E et al. [6] tracks a special computational fluid dynamics road map for forced cooling conjugate heat transfer in a rectangular computer case. The last two papers present a study on a level of computer chassis. Biswas et al. [7] has studied on the airflow in the computer chassis and their objective was examined the pressure loss due to occurrence of grills at the top and bottom on the chassis. Argento et al. [8] was moved forward the similar experiments by verifying the analyze result experimentally. Their modification was so good that surface temperature decreased as 56 percent. Linton and Agonafer [9] compared the experimental data with analyze results of the heat sinks in the detailed CFD model. Their coarse model settled well with the detailed arrangement without losing the features of the heat sink. Sathyamurthy et al. [10] investigated staggered heat sink performance using the software FLUENT. In this study experimental results were agreed well with FLUENT results.

In the industry there are many different design for the location of fans in the computer chassis. Thereby, the main aim in this study to find optimal outline by modifying the location, number and power of the fans, opening additional exit holes for the high temperature flow in the chassis without changing the arrangements of other components in the chassis. Mainly the FLUENT is used across the study and it should be noted that this analyze not consider the chip level heat transfer.

2. Problem Statement

This main objective of this present work is to investigate well-known commercial application solved by computational fluid dynamics technique and to analyze thermal and flow fields in a CPU cabinet. The geometry is simplified to get rid of complexities which are effect grid generation and solution time around the sub components. There are many boundary conditions and more than one heat sources therefore multi block approach is necessary while generating the grid. Active cooling means that to cool the chassis inlet and exhaust fans are used. In many cases CPU (Central Processing Unit) has its own fan but in this study, closed loop liquid cooling is applied for CPU. General case of problem is shown in Fig 1.

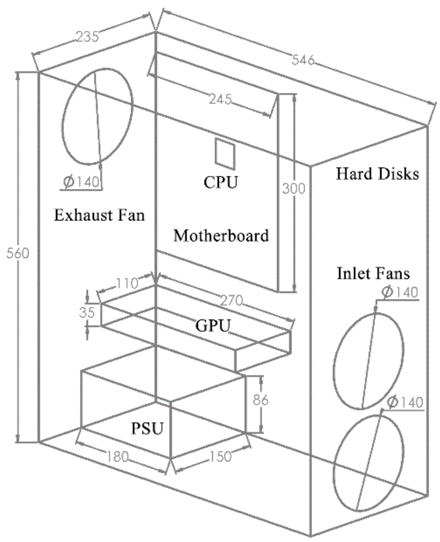

Figure 1. Computational Domain (Dimensions are in mm).

The present study takes into account an ATX (Advanced Technology Extended motherboard) computer chassis. Fourth generation 22 nm INTEL 4790K central processing unit is used in this configuration. Its thermal design power is 85 W which means this processor dissipates 85W heat with all cores loaded. In the computer chassis as a graphic processing unit NVIDIA GeForce 780 Ti Edition is used. It has an 875 MHz base clock and 928 MHz boost clock. Maximum operating temperature is 100 degrees in Celsius with 70W heat dissipation. In studied configuration, 4 units of Corsair 4 GB DDR 3 1600 MHz random access memory are used. One unit has 8W heat dissipation. This computer powered by Corsair RM 750 PSU with 75W heat dissipation. Two units of Seagate 500 GB SATA hard drive with 30 W heat dissipation are used for storage. Inlets and outlet fans have selected from Corsair's Airflow series and they have 140mm diameter and 75 CFM airflow.

2.1. Governing Equations

RANS (Reynolds-averaged Navier-Stroke) equations are going to be solved for time independent flow. The equations for continuity, momentum and energy are shown below. The viscous dissipation term will be misplaced. Equation 1 is described as continuity equation and equations 2, 3 and 4 are described as the momentum equations in x, y and z directions, respectively and the last two equations are described as the energy equation and equation of state.

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

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

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

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

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

$$p = \rho RT \tag{6}$$

where (u, v, w) are components of velocity V_r 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 is directional body force and τ is shear stress.

2.2. Turbulence Modeling

The default turbulence model of all calculations is SST k-w Turbulence Model. The SST k-ω turbulence model (Menter, 1993) is a two-equation eddy-viscosity model and this model recently has become more popular. In the inner parts k- ω formulation is used for the boundary layer and this helps the SST k-ω model can be used as a Low-Reynolds turbulence model with no necessary any additional damping function.

2.3. Radiation Effects

Alpha heat sink was analyzed by Öztürk (2004) to study the radiation effects and it was seen that effect is too small. Thereby, it is resulted that radiation could be ignored for this kind of studies i.e. forced cooling of CPUs.

2.4. Boundary Conditions

Subsequently Navier-Stokes equations are solved inward the domain, no-slip boundary condition is applied to all walls in the dominion. Therefore, at all surfaces u = v = w = 0. It is assumed that the heat transfer between chassis and ambient is natural convection and a value of 5 W/m²K is selected as a coefficient between the ambient and computer chassis walls. Other coefficient are calculated with definitions of Rayleigh and average Nusselt number:

$$Ra_L = Gr_L Pr = \frac{g\beta(T_s - T_\infty)L^3}{\nu\alpha} \tag{7}$$

$$h = \frac{\overline{Nu_L}k}{L} \tag{8}$$

where L is the characteristic length; h is convection heat transfer coefficient, k is fluid thermal conductivity, g is gravitational acceleration, β is volumetric thermal expansion coefficient, ν is kinematic viscosity and α is thermal diffusivity, respectively. T_S and T_∞ are the surface and ambient temperatures. It can be seen that calculations gives the result of Rayleigh number is less than 10^9 for all the surfaces which means that the flow is laminar. Using correlations, the thermal conductivity of air is taken as $k = 27\times10^{-3}$ W/mK and heat transfer coefficient $h \approx 3$ W/m^{2}K. These heat transfer coefficients are applied to all chassis walls except the bottom wall since it stands on the ground.

Table 1. Heat Dissipation Rates of Components

Component Name	Material	Heat Dissipation Rates
CPU	Silicon	70 W
GPU (PCI)	Silicon	80 W
Hard Disk Drive	Al	15x2 W
Chipset	Silicon	15 W
CPU Heat Sink	Al-Cu	-
GPU Heat Sink	Al	-
Power Supply	Al - Porous	65 W
Memory Cards	FR4	10x2 W
Motherboard	FR4	-

The thermal boundary conditions for the components inside the chassis are listed on Table 1. A total amount of 280 W is dissipated.

2.5. Mesh Generation

The high quality solution is only comes with high quality mesh. In this study, tetrahedral unstructured mesher is used. To obtain an accurate solution having a good mesh is important in terms of quality, resolution, smoothness and cell count. Mesh quality is determined by ANSYS FLUENT software with parameters Skewness, aspect ratio and cell size. Total amount of cell size for this computer chassis kept around 1.3 million.

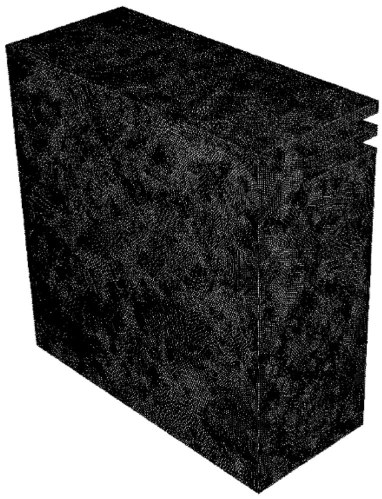

Figure 2. Grid on Computer Chassis Walls

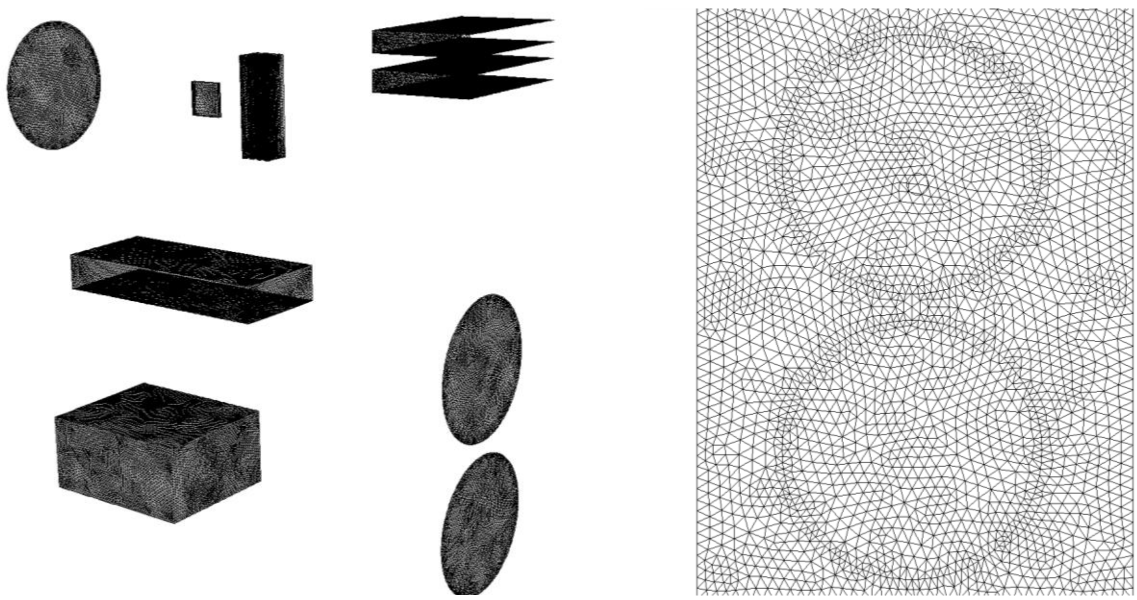

Figure 3. A Closer Look to the Mesh at the Components (on the Left) and Inlets (on the Right)

3. Results and Discussions

Necessary graphs, contours, vector fields and streamlines are showed in next figures. It should be note that in this illustration Iso-Mid plane is used. Iso-Mid plane is the plane which is exactly at the middle of the chassis cutting by x direction

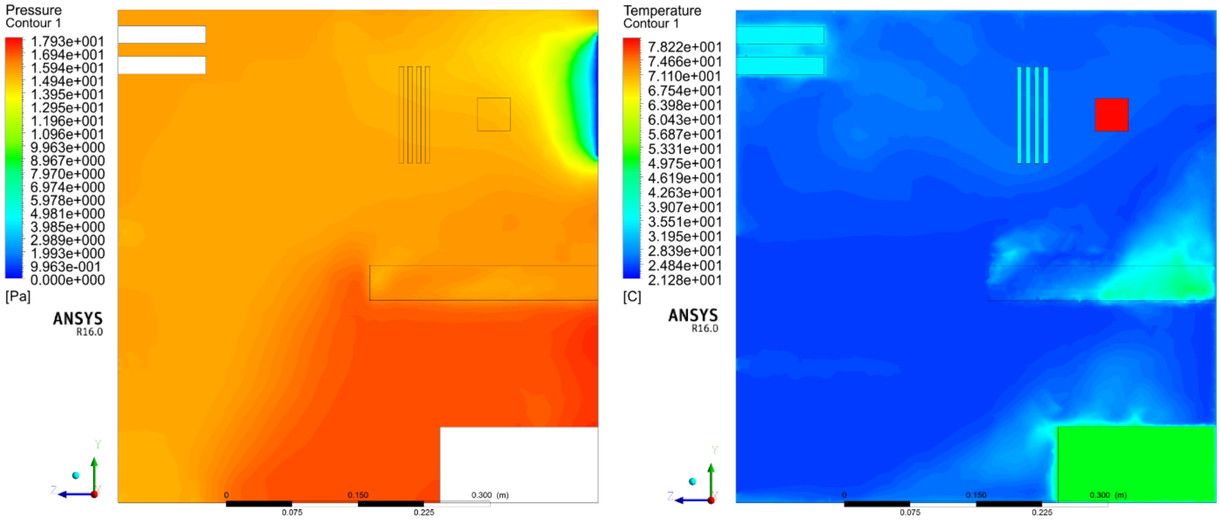

Figure 4. Pressure Distribution (on the Left) and Temperature Distribution (on the Right) Contours throughout the Iso-Mid Plane

It can be seen that pressure distribution is directly proportional with flow intention which make sense because of inverse ratio between velocity and pressure. Flow is accumulated most at the top of PSU. Because of the location of GPU, flow cannot drain through exhaust fan. This is the main reason of accumulation of the flow at the top of PSU. On the other hand the location and sufficient distance between hard drives let the flow leak between disc and this is the reason of the low pressure distribution

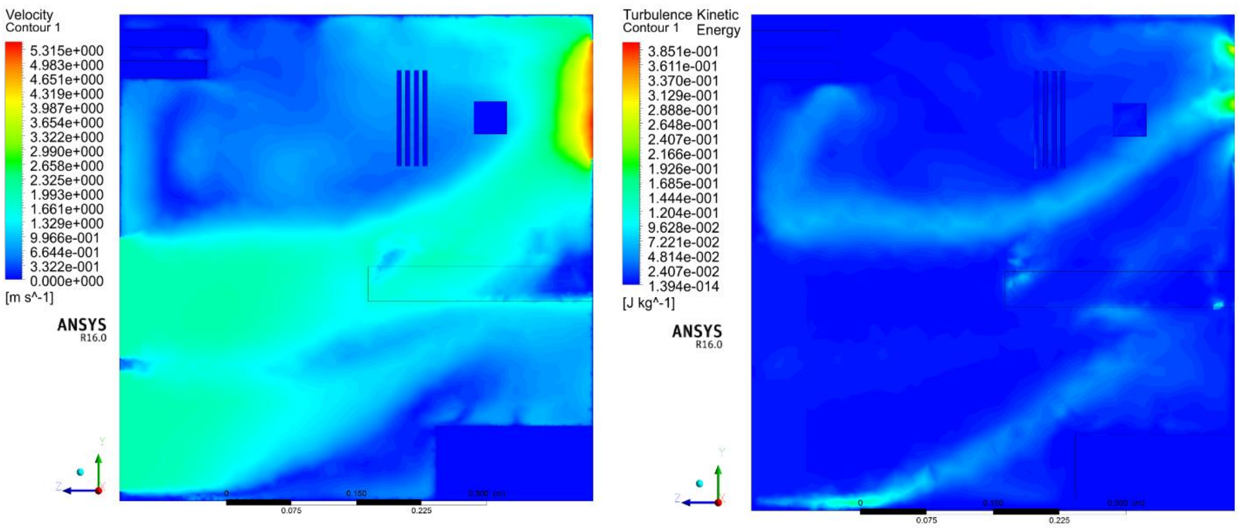

Figure 5. Velocity Distribution (on the Left) and Turbulence Kinetic Energy Distribution (on the Right) Contours throughout the Iso-Mid Plane

In the previous figures turbulence kinetic energy of the flow is represented. At the mid plane, kinetic energy condensed through flow direction. On the front side, through top inlet to exhaust fan there is a significant increase in turbulence kinetic energy.

Bottom inlet has the same flow rate with top fan and it can be agreed that from pressure distribution contours flow is concentrated on the top of PSU. The same thing is proved by turbulence kinetic energy contour. Temperature is accumulated on the top of GPU. This is because of high heat generation in the GPU. Heated flow is choked between the GPU and back side of computer chassis. This is an undesirable result which is investigated and solved in subsequent section.

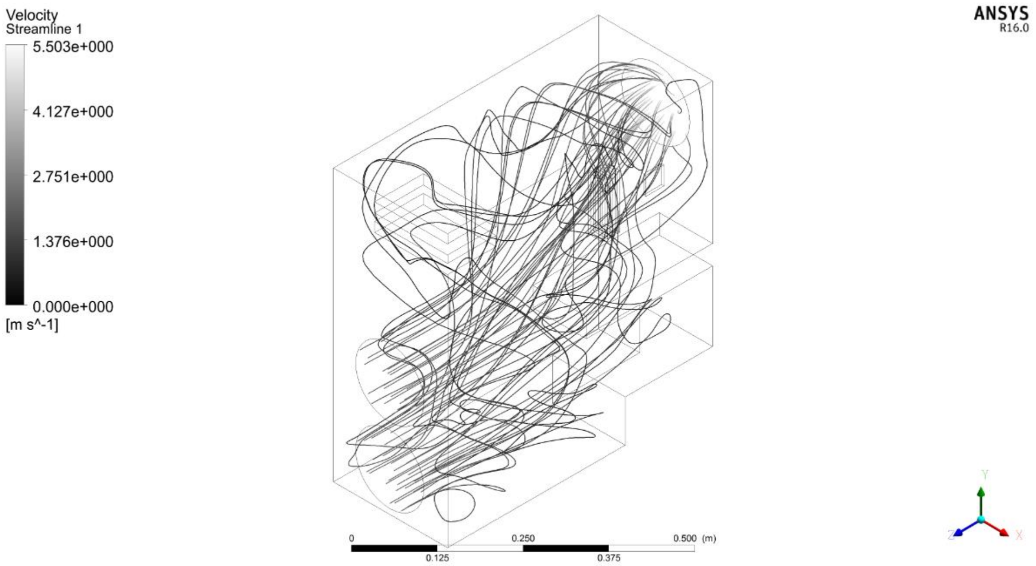

Figure 6. Flow Streamlines along the Chassis

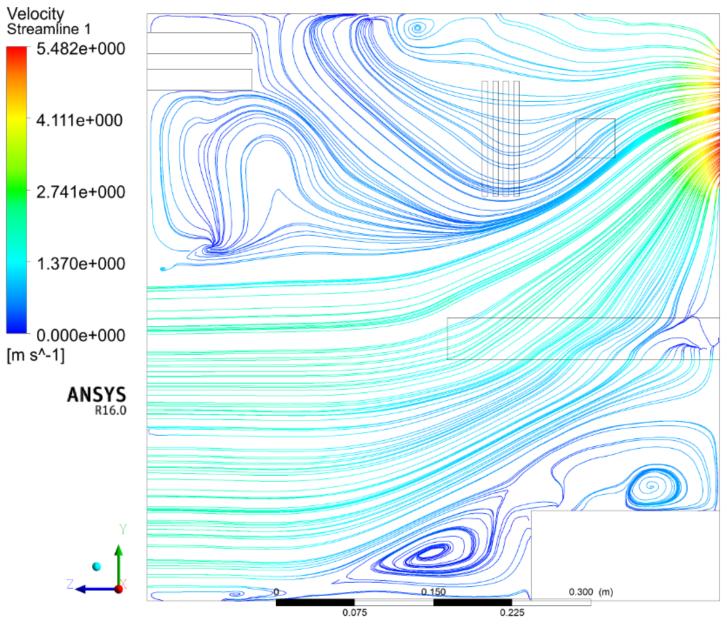

Figure 7. Flow Streamlines all along the Iso-Mid Plane

In the first streamline graph, 3D streamlines can be seen from inlet fans to exhaust fan. This is a complicated view which routes us to look at streamlines at the iso planes. Second streamline graph is taken from the mid plane and shows the flow of the air from inlet to exhaust fan. The flow is smoothly drains from inlet fans to exhaust fan since there is no blockage. Near power supply unit there are two turbulated flow and besides, flow directly drains on the random access memories and central processing unit. It can be understood from last figures that velocity and the flow spreads in a good way from inlet fans to exhaust fan. On the other hand antepenultimate figure proves that GPU blocks the flow distribution which comes from inlet towards to exhaust fan. Since the location of mainboard cannot changeable, currently there is no solution for this blockage.

3.1. Improvements in Design

In this study, flow thermal and flow field in a computer chassis are investigated. At the end of analysis it is seen that such improvements are needed. In this section improvements and upgrades are discussed.

Temperature is accumulated on the top of GPU. Heated flow is choked between the GPU and back side of computer chassis. A mechanism or such a thing is needed to get over this choked flow. The simplest improvement that can be materialized is the back chassis holes. The honeycombed shaped holes helps the choked and heated flow move away. This is an also another good improvement for cooling the chassis interior and has an effect to all of the components inside the chassis.

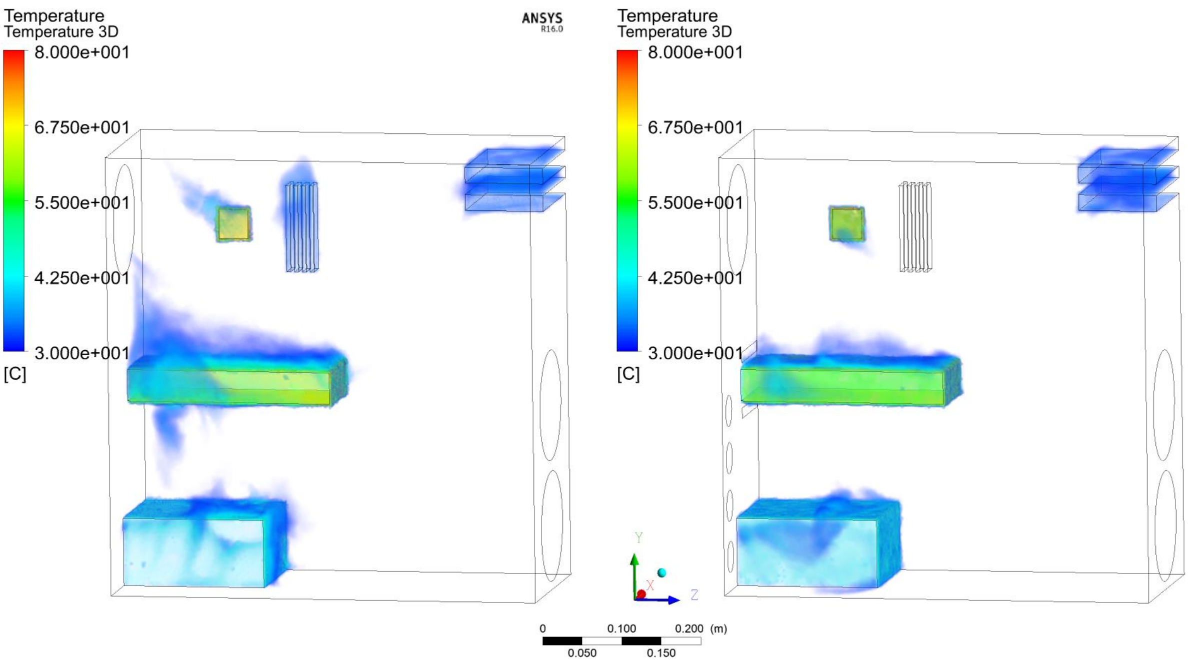

Figure 8. Choked Flow before (on the left) and after (on the right) the Chassis Improvements

In the last figure the temperatures are showed in discretized way. This mode is specially selected to see better the high and choked distribution around the GPU, CPU, RAMs and PSU. It is seen that in the gap between hard drives collects high amount of heat. As we said before, at the top of the GPU, flow with high temperature is flow slowly to exhaust but it is a sufficient flow to cool the GPU.

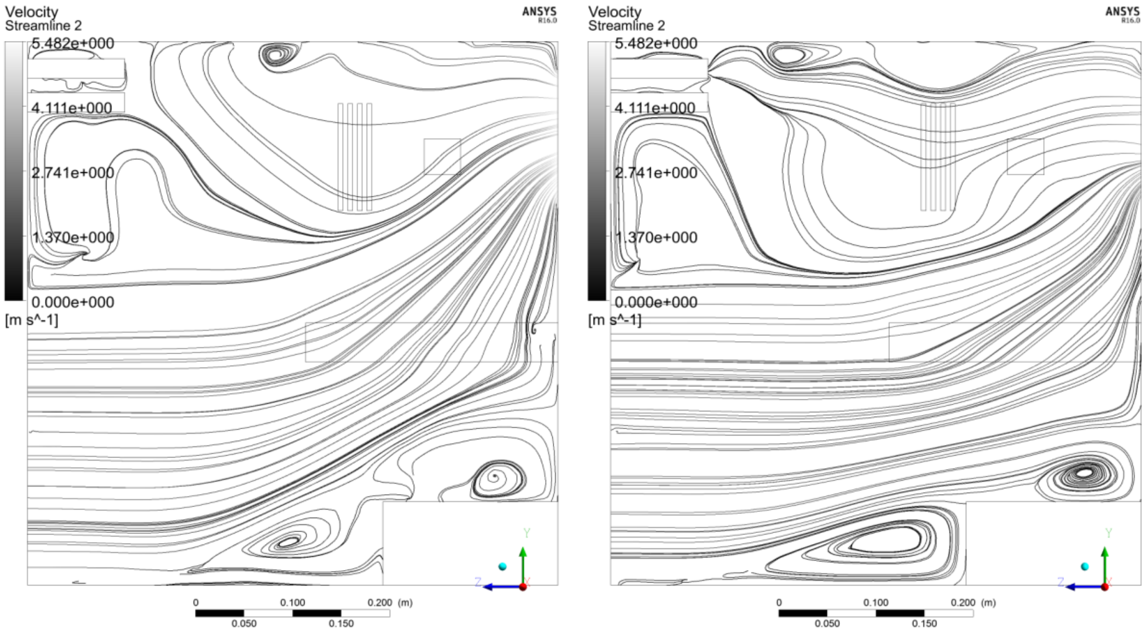

Figure 9. Streamlines before (on the Left) and after (on the Right) Chassis Improvements on the Iso-Mid Panel

4. Conclusion

A road map has been established for simulating the computer chassis. The mesh resolution, turbulence model choice, convergence criteria and discretization schemes are examined to obtain the best model with slightest computational expense. The numerical methods exposed agreement with the experimental data. Nevertheless, the comparison was qualitative. In order to make improved comparisons, the experiments should be achieved on a computer chassis bearing in mind the full model. In this study, since it is not practicable to model the fans and resistances with their exact geometry, lumped parameter models are used. This always leads into some error. Also the atmosphere exterior the computer chassis is not modelled; thus there is one more approximation, for the heat transfer outside the chassis. After improvements are done, significant amount of choked flow warded-off from inside the chassis. This helps the sub components work under cooler environment.

This study investigates the capability of CFD software in calculating flow field and heat transfer in an active cooled computer chassis. According to the results, additional holes at the back of chassis is necessary to discard hot choked flow. Besides this, GPU needs an additional hollows so that hot flow transfused out of chassis easily. An exit hollow is also necessary in the regions that vortices occurred.

References

[1] Kim, K.S., Won, M.H., Kim, J.W., and Back, B.J., 2003, “Heat Pipe Cooling Technology for Desktop PC CPU,” Applied Thermal Engineering, vol. 23, pp. 1137-1144.
[2] Yu, C.W., and Webb, R.L., 2001, “Thermal Design of a desktop Computer Using CFD analysis,” Proceedings of the 17th SEMI-THERM. 20-22 March 2001, USA, 18-26.
[3] Dirker, J, Van Wyk, J.D., and Meyer, J.P., 2006, “Cooling Power Electronics by Embedded Solids,” Transactions of the ASME, J. of Electronic Packaging, Vol. 128, pp. 388-397.

[4] Amon, C.H., 1992, "Heat Transfer Enhancement by Flow Destabilization in Electronic Chip Configurations," Transactions of ASME, J. Electronics Packaging, Vol. 114, pp. 35-40.
[5] Choi, S.K., Kim, E.K., and Kim, S.O., 2004, "Computation of turbulent Natural Convection in a Rectangular Cavity with the $k - \varepsilon - v2 - f$ Model," *Numerical Heat Transfer,* Part B, vol. 45, pp. 159-179.
[6] Ozturk, E., and Tari, I., 2007, "CFD Modeling of Forced Cooling of Computer Chassis," Engineering Applications of Computational Fluid Mechanics, vol. 1, no. 4, pp. 304-313.
[7] 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.
[8] 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.
[9] 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.
[10] 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.