Numerical Investigation of Conjugate Heat Transfer in a Computer Chassis

Konstantinos-Stefanos P. NIKAS* and Andreas D. PANAGIOTOU

Received 4 October 2012; Revised 16 November 2012; Published online 13 December 2012

© The author(s) 2012. Published with open access at www.uscip.org

Abstract

The present work investigates a numerical simulation of conjugate heat transfer effects inside a computer chassis which is cooled by fans forcing heat convection. The cooling schemes selected for the current study were three, based on the available commercial applications. The temperature distribution on the heat sources and the air pathlines were investigated in order to get a clear picture about which case fully satisfies the operating conditions. The simulation was conducted by the commercial CFD software (FLUENT). The solution is based on the Navier-Stokes equations applying into a three dimensional model. The turbulence model used in this paper was well validated by corresponding experimental data consisting mainly of velocity profiles and temperature distribution provided in the literature review. The study introduced ways that could be beneficial, when cooling of electronic components is needed. In addition, through CFD process, the problematic areas are revealed in order to consider improvements.

Keywords: Simulation; Conjugate heat transfer; CFD; Chassis; Cooling

1. Introduction

The modern world consists of electronic devices and especially microelectronic applications. Almost every device has been made in favour of the actual human user. Airplanes, industrial machinery and even domestic devices contain chips and electronic circuits, in order to perform simple or even complex functions. Therefore, scientists try to upgrade every system in order to provide more power and allow more tasks to be performed at the same time. The major disadvantage of electronic systems is the temperature rise, when certain voltage passes through their circuits. Every system has a threshold temperature and it is very crucial to avoid reaching this level. When passing the manufacturer’s threshold temperature, the system starts to fail or even make the device to work off. As a result, the heat transfer in the electronics must be examined before designing the application.

*Corresponding e-mail: ksnikas@teipir.gr
Technological Education Institute of Piraeus, Mechanical Engineering Department, Fluid Mechanics Laboratory, 250 Thivon & P. Ralli ave., Aigaleo 12244, Greece
Several studies have been performed worldwide regarding the passive cooling of the electronics (Sultan, 2000; Banerjee et al., 2008; Geisler, 2009), but as the systems are improved, passive cooling is not the best option to be considered. The performance of electronic devices has been improving along with the rapid technology development. Cooling of electronic systems is consequently essential in controlling the component temperature and avoiding any hot spot. There are several methods to cool down the electronics components such as the pin-fin heat sink, confined jet impingement, heat pipe, micro heat sink and so on. However, for active and passive cooling methods, the pressure drop could extremely rise when the heat transfer rate is increased (Gururatana, 2012). At this point, many researchers, located all over the world, indicated that heat sinks must be placed in the electronic applications in order to achieve an increased heat flux between the heated blocks (El-Sayed et al., 2002; Yazawa et al., 2006; Fan et al., 2012). Heat sinks may vary significantly as far as the design, material or the fins are concerned. However, they are chosen in order to suit the usage of each system. A lot of systems use a forced flow regime due to the fact that quite fast cooling of the electronic devices is needed. Impingement experiments regarding the behaviour of plate fin heat sinks were investigated by Duan and Muzychka (2006, 2007). They focused on the plate fin geometry and assumed one dimensional heat transfer in the radial direction. They tried different fin spacing and heights, in order to develop a heat transfer model to predict the behavior of the fin heat sinks based on each individual use. The model that they tried to develop was supposed to examine the pressure drop as well during laminar flow. The importance of this paper stems from the fact that they performed several measurements for four heat sinks. They concluded that the pressure difference increases when there is a decrease at the impingement inlet width and that the fin spacing plays a significant role over the thermal resistance, which also was evaluated. Furthermore, El-Sayed et al. (2004) experimented in turbulent environment using rectangular fin heat sinks, by searching for the best orientation in order to have the optimum heat transfer effect for each condition they used. The knowledge of this issue is essential, since every device differs substantially.

Nowadays, many researchers try to cope with the challenges of the heat transfer effects around the electronic applications by using simulation processes. Computational Fluid Dynamics (CFD) softwares solve the transport governing equations of fluid and mass flow in order to solve each case individually. Rodgers and Eveloy (2003) indicated that computational fluid dynamics software can be a great tool to substantially predict the temperature distribution, the pressure drop, fluid velocity and various other characteristics which are related to conjugate heat transfer problem around electronics. A CFD simulation was also performed by Leon et al. (2004), who solved the numerical equations with the implementation of ANSYS FLUENT program. Their study focused on the fins of the heat sink. They believed that if they applied staggered formation between the spacing of the fins, they would achieve improved cooling and thermal efficiency. They also suggested implying rounded fins instead of the most common geometry, which is the rectangular type.

Another successful computational case study was performed by Chen and Wang (1998). They examined the flow over two objects using the naphthalene sublimation technique, in order to obtain heat transfer analysis. A wide range of data occurred from this experiment, while the computational analysis graphically illustrated the heat transfer effects and the velocity of the fluid. The turbulent data showed good agreement with the experimental measurements; however, there were deviations when low Reynolds number was considered. Last but not least, the spacing was examined in reference to the heat transfer proving that the Nusselt number was substantially
increased. It is well noticeable that CFD applications become more and more popular thanks to the constant computer’s growth, improvement of the existing fluid flow models and low cost of experiments.

Finally, it has to be mentioned that there are a lot of research works examining, in general, the ventilation in bigger and different volumes than a computer chassis such as buildings, which use the mechanisms of ventilation for other purposes. Both experimental (Karava et al., 2007, 2011) and numerical studies (Nikas et al., 2010; Larsen et al, 2011; Nikolopoulos et al., 2012) have been reported throughout the literature, showing quite well agreement between them.

2. Problem Description

This paper investigates a well-known commercial application solved by CFD technique by modeling the cooling of a complete computer chassis. An investigation will be made regarding the air flow circulation and the temperature distribution profiles occurred when certain thermal considerations are applied. Position, geometrical characteristics and the number of fans will be altered to study the thermal effects. The numerical simulation of the computer chassis investigates a few cases of active and passive cooling scheme in a standard geometry layout. Active cooling case is the condition of the computer chassis that includes a fan on top of the processor’s heat sink. The inlet fans and their geometrical characteristics will be altered in order to identify which is the best case that produced the maximum heat transfer results. A general picture of the problem with one inlet and one outlet fan is shown in Fig. 1.

![Fig 1. Side view of ATX case with inlet fan and outlet fan](image)

3. Numerical methodology

From the available programs of ANSYS commercial package, FLUENT was the one that was used for validation against the experimental data, through the solution of the main governing equations
around the thermo fluid effects (Fluent, 2005). The current application uses the finite volume method and the main equations solved are the continuity equation for mass transfer, the momentum equation, the turbulence-modeling equations as well as the energy conservation equation. The Reynolds-Averaged Navier-Stokes (RANS) technique is the most suitable for the present scale model due to its capability to solve turbulence environments and the compatibility with FLUENT. RANS technique uses two-equation models such as the standard, RNG and Realizable version of k-ε. In addition, during the validation process, the k-ω turbulence model was tested among the rest turbulence models. Upon the validation of the model, the same three-dimensional complex geometry of a computer chassis was simulated. The model, which produced the best approach as far as the comparison between computational and experimental data, was the standard k-ε turbulence model, as described in the following paragraphs. For all the models used, enhanced wall functions were enabled in order to achieve more accurate results at the near wall section of the computational domain.

Finally, hereby it should be noted that the present study performed under steady state conditions and incompressible air flow, while the QUICK discretization scheme was used for the momentum, turbulence and turbulence equation. The energy equation was also discretized with a second-order scheme. The equations used for the k-ε models are presented in great detail both in Fluent’s User’s Guide (2005) and international bibliography, such as Launder and Spalding (1972, 1974).

4. Validation Process

4.1 Numerical Details and Grid independency

At this study, a representative heat sink was chosen from the available geometries indicated in the experiment of El-Sayed et al. (2004) as shown in Fig. 2. The designing of the aforementioned heat sink was crucial in order to evaluate the validation of the CFD model. The validation of the model was confirmed by the corresponding comparison of the available experimental data presented on
the experimental study of El-Sayed et al. (2004). The present calculations used the same boundary conditions provided by the experiment examining a horizontal square duct with dimensions 440 x 440 mm with fixed inlet velocity (Re = 24761) and temperature (T_{in} = 30°C). The cooling fluid is air and it flows through a blower to the duct. The heat sink model is attached to the side wall section and consists of 10 rectangular fins with fin height of 46 mm, fin spacing of 10 mm and fin thickness of 6 mm. Air flows in a parallel formation of the streamwise direction, while measurements carried out in the interfing region and in different heights/distances. Below the heat sink model, two heat sources exist with total heat load of 1573 W.

In order for the simulation to be as reliable as possible, the computational domain was tested in different mesh sizes in the range of 2.88M up to 5.48M cells. The cells were denser in the region of the fins and especially in the interfing region where the experimental data were drawn, including almost the 50% of the whole mesh. The fluid’s cell zone regions consisted of tetrahedral element type cells, where in the solid regions hex element type cells were used. Finally, the denser grid was used since it produced independent results (see Fig. 3).

**Fig 3.** Computational mesh with the corresponding details of the geometry used under the validation process

### 4.2 Fluid flow results

The work of El-Sayed et al. (2004) included velocity measurements at different distances from the interfing region entrance. Fig. 4 shows that there were 2 sub models from the k-ε and k-ω turbulence models which were considered during the fluid flow calculations. After the examination of the standard model, the rest models were simulated and found to have underestimated the width of the
boundary layer concerning the hydrodynamic characteristics, compare with the corresponding experimental data (see Fig. 4). Most turbulence evaluated models couldn’t even reach the highest velocity terms, thus by using the k-ε standard model and comparing the computational and experimental results shown in Fig. 4, one should conclude that the error is quite reasonable for the velocity profiles. The results of the velocity profiles at the positions of x = 75 and 135 mm match with the experimental data up to the fin’s height of 44 mm. Thereafter, a small deviation is observed but again the error is not high enough to disturb the results. On the other hand, the velocity profile of distance x = 135 mm matches better as the fin’s height increases, but there is a small error as the boundary layer is formed. A closer view at the computational results reveal that the simulation with the use of k-ε standard turbulence model with enhanced wall treatment and second-order discretization scheme is the most suitable and matches the corresponding experimental velocity data quite well. The chosen model was preferred because the behaviour of the velocity boundary layer is simulated better, compared with two other models k-ε RNG and k-ω which produced the velocity profile through the interfin region either too early (see Fig. 4) or they overestimated the velocity much more/less than it was needed over the heat sink channels. As a result the heat transfer could not be produced in a satisfactory manner.

Fig 4. Comparison of most common models used for the experimental data and computational results in the inter-fin region at distances x = 75 mm and x = 135 mm
4.3 Heat transfer results

Upon the completion of the fluid flow results, the energy model was added in order to process the heat transfer results. Two sources along with insulation and a wooden box were added on the heat sink in order to create heat fluxes based on the experiment of El-Sayed et al. (2004). As previously mentioned, the whole model was placed on the side wall of the duct and the fluid flow was fixed at the $Re = 24761$. Through thermocouples at the fins surface, thermal data were measured in order to gain appropriate temperature data. Fig. 5 illustrates the corresponding comparison of heat transfer data that were simulated at the FLUENT software. At this stage, one may notice that although most computational measurements have an agreement with the experimental data, deviations occur at the height of 11 and 44 mms respectively. This small error exists due to the fact that fluid flow results affect the thermal behaviour of the heat sink and especially at the boundary layer/maximum fin’s height. Through the post processing phase of the computational programs, there was the possibility of examining more deeply the validated case. Therefore apart from the numerical data, schematic presentations of the heat transfer coefficient, deviation of Nusselt number, or surface temperature could be created in order to examine the heat sink deeper. The new data has shown how the heat is dissipated on the base plate of the heat sink. Since the air is coming through the front side of the heat sink model, a large amount of heat is absorbed and the fins are sufficiently cooled instead of the other side (behind) which has the highest base plate temperature. In addition, the convection process is quite high at the inlet of the heat sink channels but very small as the velocity decreases. The need of further cooling techniques is needed if we use this heat sink for further research in order to increase the heat transfer coefficient.

**Fig 5.** Comparison between experimental data and computational results in the inter-fin region for temperature distributions
4.4 Results and Discussion

The validation process between experimental and computational work revealed certain conclusions for the current case study. The main aspects of the study such as the conditions of the experiment, details of the experimental rig were presented and analyzed based on the literature review. The process of the grid independence was achieved based on both fluid and heat transfer experimental data. The CFD process matched the experimental data and the whole CFD case seemed that the calculation using the QUICK scheme along with the k-epsilon standard turbulence model is ideal for heat sinks simulation. The QUICK scheme is used for hybrid meshes with quadrilateral and hexahedral cells, such as the mesh used in the present study, while the second-order discretization is better applied to all other types of cells. However, in some cases, problems revealed at the boundary layer and at the maximum velocity of the fins height. The conclusion of the fluid flow and heat transfer results is that the velocity has the greatest value when the fluid enters the interfin region and slowly decreasing as the hydrodynamic boundary layer is developed. This causes the wall temperature to be low at the inlet of the heat sink and high at the end distance of the leading edge in the streamwise direction.

5. Simulation of the Computer Chassis

In order to produce useful conclusions concerning complex geometries involving conjugate heat transfer, a desktop computer case including most of the electronic devices will be investigated. Intel manufacturer will be the main vendor of the parts that will be studied along with other secondary items such as the hard disk and optical drive and this is chosen for convenience only and not an endorsement of any particular vendor. As far as the computer chassis is concerned, there is a limitation for the current case. BTX and ATX chassis types can be used from the available commercial market. The most common choice of the consumers is the ATX case since this type of the computer chassis is extensively used the past decade. However, Intel as a product manufacturing company does not produce chassis, thus a universal publication organization is in place to publish standards which can be globally used for a wide variety of computer parts. The aforementioned site is an electronic library wide open to all users and vendors in order to publish the latest specifications for the corresponding products. The site also has a registration form in order to welcome new vendors at their list. The cooperation with thermal and mechanical testing labs offers capabilities of improving and ensuring reliability of the existing products (Intel, 2009).

In order to understand the features of an ATX chassis, Fig. 6 illustrates an overview as recommended in the Form Factors. Each of the electronic parts used in this study were designed based on INTEL’s technical manuals regarding the properties of the LGA 775 motherboard type. The processor under investigation is a quad processor Q9550S. The maximum surface temperature of the processor is 76.3°C in order to operate without problems (Panagiotou & Nikas, 2011). The major simulated parts are the processor, chipset, bus control hub and memory DIMMS. Secondary devices such as the hard drive and DVD rom were also simulated in order the problem to be more realistic when conjugate heat transfer effects will apply (Fig. 7). However, this study will also investigate what the best arrangement of the fans is in order the chassis to be sufficiently cooled.
5.1 Numerical Details

The current numerical process depends on the solution of the transport governing equations (Fluent, 2005). They define the problem under simulation and provide accuracy of the simulated data. The equations represent the flow which is formed by the motion of the air inside the computer chassis and especially around the heat sources. In the present study, there will be an extensive use of the mass, momentum, energy and turbulence equations. More specifically, the energy equation is defined as follows:

\[
\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\bar{u} (\rho E + p)) = \nabla \cdot \left( k_{\text{eff}} \nabla T - \sum_j h_j \bar{J}_j \right) + S_b
\]  

(1)
where \( k_{\text{eff}} \) is the effective conductivity (defined according to the turbulence model being used), and \( J_j \) is the diffusion flux of species \( j \). The first three terms on the right-hand side of Eqn. 1 represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. The term \( S_i \) includes the heat of chemical reaction, and any other volumetric heat sources that have been defined. The computational domain the computer’s chassis used have dimensions of 460 x 210 x 440 mm (height x width x depth). The mesh for all parametric cases consists of about 4.8x10^6 mixed tetrahedral and hexahedral computational cells. Inside the interfin regions of the heat sinks, special attention was addressed in order to fill with an appropriate amount of hex type cells, approximately the 2.2x10^6 cells, as well as for the solid material, in order to predict the thermal and fluid characteristics of the conjugate heat transfer phenomenon in the best way. The remaining computational area was meshed by tetrahedral cells, since the geometry examined was not appropriate for hex cells. The whole problem has no time-dependent factors therefore it is solved with steady state conditions. The working fluid was supposed to be incompressible air and the turbulence model applied during the simulation process was the k-\( \varepsilon \) standard with enhanced wall treatments and QUICK discretization scheme.

5.2 Boundary Conditions
The inlet fans that cool the chassis are of two types, one with 80 mm diameter and one with 120 mm diameter. As per manufacturer specifications it was calculated that the inlet velocity of the 80 mm and 120 mm fan is 2.85 m/s and 2 m/s, respectively. Those fans have specific installation positions at the chassis as per ATX specifications (Intel, 2009). They can be used at the front, side and back face. During the simulation, the chassis will be simulated with no wall thickness but only the wall-faces to form the boundary conditions of the problem. The velocity and temperature inlet in considered uniform based on the fact that the computer operates inside a building with temperature of 299 degrees K. The power supply unit’s fan blows the air inside the chassis with temperature of 304 degrees K. Heat term sources were in the range of 3W up to 150 W. For instance, the major heat source under consideration (CPU) had 65 W thermal design power. The thermal conductivity of all materials was acquired as well in order to input all parameters to the CFD software.

5.3 Results and Discussion
The numerical results of the six simulated cases (See Table 1) offer the possibility to identify the optimum fan orientation in which the processor and all the secondary devices managed to achieve operating conditions based on the manufacturer recommendations. In addition, there will be a comparison between the cases that included a side inlet fan. Through the simulation process it is rational to find when the side fan has greater performance. It can be set to blow air directly to the processor from the environment or to suck air from the chassis. The types of figures created from the FLUENT’s post-processing phase were mainly pathlines and contours filled by temperature distribution. Both figures addressed the problem well and revealed the weaknesses of each case.
Table 1 Breakdown analysis of simulated cases

<table>
<thead>
<tr>
<th>Cases Simulated</th>
<th>Description of each case</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Passive cooling with one inlet fan of 120mm diameter</td>
</tr>
<tr>
<td>B</td>
<td>Active cooling with one inlet fan of 120mm diameter</td>
</tr>
<tr>
<td>C</td>
<td>Passive cooling with two inlet fans of 80mm diameter</td>
</tr>
<tr>
<td>D</td>
<td>Active cooling with two inlet fans of 80mm diameter</td>
</tr>
<tr>
<td>E</td>
<td>Active cooling with two inlet fans of 80mm diameter and one discharge side fan of 120mm diameter</td>
</tr>
<tr>
<td>F</td>
<td>Active cooling with two inlet fans of 80mm diameter and one suction side fan of 120mm diameter</td>
</tr>
</tbody>
</table>

Case A: Passive cooling with one inlet fan of 120mm diameter
The first case refers to a passive cooling scheme. In general, a PC chassis has another two fans. One is located at the back chassis face as fluid’s outlet and another one which is used by the power supply unit. The temperature distribution of the whole model under the passive cooling case can be illustrated in Fig. 8. The simulation showed that the maximum temperature is up to 500K and it is concentrated over the processor area due to the lack of air flow directly at the processor. The lowest temperature can be seen at the secondary sources such as the DVD and the hard disk. A closer view at the motherboard and the processor surfaces indicate that the south side of the motherboard has lower temperature than the north area. Typically, this is justified by the cool air from the front fan which washes the south area and then it exits from the outlet fan. The processor maximum temperature is at the range of 483K which affects significantly the functionality of the device. In addition, it can be noticed that the socket of the processor maintains the temperature of the motherboard surface below 400 K. This is quite important because heat should be dissipated upwards where the air flows through the heat sink and the processors side walls. The rest of the motherboard sources maintain their temperature below the threshold one. The reason is obviously stemmed from the small heat flux of the chipset the thermal design power and secondly from the available air flow around the device. It has to be mentioned that the temperatures of hard disk and the DVD drive exceed the manufacturer’s critical limit. The reason of the heat transfer failure is shown by the pathlines. The chassis faces lack of air circulation which increases the parts inside sources’ temperature and therefore performance is decreased.

Referring to the power supply unit fan, it was discovered that the air exits directly from the outlet fan. The advantage of this result is that hot air from the power supply unit is not circulated in the chassis. As far as the temperature of the chassis is concerned, it can be seen that the case temperature is at very good levels. The temperature ranges between 299K and 319K, which is quite normal for electronic parts cooling. Intel as a global manufacturer recommends that a temperature of 40°C (313K) is supposed to be the maximum environmental temperature (Panagiotou, 2011).

Case B: Active cooling with one inlet fan of 120mm diameter
The next case refers to an active cooling scheme chassis with the same fan parameters as case A. Upon completion of the simulation process, the contours (see Fig. 9) may show the temperature distribution over the whole model, which seem quite satisfactory. The maximum model’s temperature is close to 340K. The part with the maximum temperature is not the processor as it was initially expected, but the hard drive. The average temperature of the motherboard is 317K and
most of the devices do not exceed the value of 328K. The processor's temperature distribution is satisfied in the current case and it can be noticed that the processor's fan is capable of maintaining the temperature at the range of 325K. The fins dissipate quite well the heat yield from the processor and generally all the sources on the motherboard behave pretty well. The chipset fins show a variation in the temperatures from left to the right side and this is caused by the air circulation inside the chassis. On the other hand, one should, nevertheless consider the creation of velocity pathlines in order to check the air movement inside the chassis. Fig. 9 illustrates the velocity pathlines. The self explanatory detail view of the processor's fan clearly indicates how the air is discharged on the fins. Due to the air turbulence and the location of the rest parts of the pc, the air flows centrifugally. Temperature at the incoming air is well maintained below 313K, therefore the recommended conditions are verified.

![Temperature distribution contours (a) and air pathlines (b) for CASE A](image)

**Fig 8.** Temperature distribution contours (a) and air pathlines (b) for CASE A

![Temperature distribution contours (a) and air pathlines (b) for CASE B](image)

**Fig 9.** Temperature distribution contours (a) and air pathlines (b) for CASE B

Specifically, the air has an inlet temperature of 299K, but while it travels through the computer case, it reaches the temperature of 313K in the motherboard area. As far as the air movement is
concerned, it can be noticed that the south side of the chassis has pretty low air circulation. In contrary with the northwest side, the air's velocity is high in some local areas due to the sudden flow of air over the heat sink. The turbulence forces the air to escape from the rest sides of the heat sink, thus rest electronic parts are cooled as well. The velocity contours also indicate a few black zones regarding the flow of air. The air is forced to wash better the DVD-rom and not the hard disk device, thus high temperature is experienced in the east area of the chassis.

Case C: Passive cooling with two inlet fans of 80mm diameter
In order to evaluate if the fan's number and its power is crucial for the chassis cooling, the inlet conditions of the model will be modified so that two fans of 80mm diameter are positioned almost in the same location as in case B. Although the fan's area is smaller, the velocity of the passing air is higher due to the fact that the flow rate is almost doubles than a fan of 120mm diameter. The present case involves passive cooling of the computer's chassis. Fig. 10 indicates that the passive cooling scheme of the current case is not better than case A. The difference of 10K at the maximum temperature of the chassis confirms the previous statement. A better view on the motherboard geometry shows that the processor holds the maximum temperature of the chassis, while the rest electronic parts are ranged below 380K. The processor's maximum temperature cannot be cooled and its operation is critical. On the other hand the chipset's temperature is far below than the processor temperature, which ranges between 342K and 350 K. Again, chipset's and bus controller hub temperatures satisfy the operating conditions of the manufacturer. Apart from the temperature contours, pathlines could highlight the reason of such high temperatures inside the chassis. Fig. 10(b) derives the same conclusion like in case A. The air inlet from the power supply unit exits directly from the back fan without entering at the chassis environment, causing high temperatures due to the lack of air circulation. Maximum temperature of the air stands for 305 - 312K. Pathlines at the top of the chassis may report that the temperature of air at the upper side of the chassis is pretty high since no air is circulating at these areas. In any case the present cooling scheme is not in position to be used for the cooling of the mentioned electronic parts.

**Fig 10.** Temperature distribution contours (a) and air pathlines (b) for CASE C
Case D: Active cooling with two inlet fans of 80mm diameter

Case D refers to the active cooling scheme of the previous mentioned model. The same technique for the fan processor will apply as in case B. The range of temperatures occurred during the simulation on the essential parts is between 300K and 334K. The general model geometry filled by contours of temperature distribution is depicted by Fig. 11(a). One should note that the temperature on the secondary devices is pretty low in contrary with the previous cases. Not only is the temperature below 316K, but the general view does not indicate any important areas of high temperature. A closer look at the motherboard geometry, one may observe the uniformity of the temperatures on the heat sources. The temperature of the chassis is low at the southeast side where there is air blowing by the fan and as it flows upwards, the temperature is increased. The processor temperature and generally all the heat sources satisfy the operating conditions of the computer. A better view to understand why the current case is better than the passive cooling scheme, pathlines will be created. Fig. 11(b) illustrates pathlines of the inlet and processor's fans colored by temperature profiles. When the inlet air from the front ATX fans joins with the air coming by the main processor's fan, a centrifugal turbulence scheme is created. The low temperatures inside the chassis are justified by the coming air which is forced to move at areas that previously could not be reached. The air's general temperature fluctuates close to 309K at the north side of the computer case and at the south side of the chassis it ranges at 304K. Another proof of the fact that this case is the best so far is the layers of pathlines around the secondary sources. They seem to be denser as air is forced to pass from the gap between the hard disk and the DVD rom.

![Fig 11. Temperature distribution contours (a) and air pathlines (b) for CASE D](image)

Case E: Active cooling with two inlet fans of 80mm diameter and one discharge side fan of 120mm diameter

Apart from the front side fans, another inlet fan case will be investigated. A side fan of 120mm diameter will be settled above the position of the processor. This option will ensure additional flow of air to be driven directly to the processor's fan and therefore to the processor. Furthermore, the position of the side fan will also need to be investigated in order to understand whether or not the side fan plays an important role at the rest chassis air flow. The case will be studied as active
cooling scheme, thus fan processor terms are inserted. The maximum temperature of the chassis’ parts can be seen at the hard drive source. Although this electronic device cannot function properly at this temperature limit, the rest equipment seems to be at a good agreement. According to Fig. 12(a), the maximum temperature is up to 330K and is located at the bus control hub device. The processor as the main heat source of the simulation introduces temperatures of up to 320K. In contrary with the previous cases B and D, one may notice that the side fan cools the processor sufficiently. As far as the bus controller hub is concerned, it can be seen that temperatures are on the range of case B. However, according to Fig. 12, fins are not in direct contact with the air since no large deviations exist at the recorded temperatures. The average temperature at the heat sink surface is about 325K. The corresponding pathlines of the current case are illustrated in Fig. 12(b) and show that although the side fan discharges the air directly to the processor area, the turbulence existed in case D is not present. The trend of the air is to circulate more in the lower layers of the ATX chassis than the main and upper ones, which causes high temperatures over the secondary sources as well. Fig. 12(b) concludes that the side fan inlet is not of any further effect on the areas where the whole chassis needs to be cooled. The chipset, bus controller hub and memory DIMMs seem to have a lack of air circulation. Local temperature near the hard disk is also at high levels affecting the functionality of the device.

![Fig 12. Temperature distribution contours (a) and air pathlines (b) for CASE E](image)

Case F: Active cooling with two inlet fans of 80mm diameter and one suction side fan of 120mm diameter

The last case completes the analysis of the simulations. The model for evaluation will be exactly the same as case E, but with different side fan terms. In many forums, there is a debate about the chassis cooling method characteristics. Therefore the setting of the side fan to function either as a discharge or a suction fan has to be considered and studied. Case E evaluated the side fan as a discharger. The contours of temperature distribution over the general model geometry will be introduced to examine where the maximum temperatures are located (Fig. 13). A quick view on the simulation model indicates that the highest temperature exists again on the hard disk surface. The rest parts of the chassis seem to have reasonable temperatures. The processor’s maximum
temperature is observed at 323K, while at the previous cases D and E behaved much better at the cooling of the processor and case B had more or less the same values. A general view of the pathlines filled by temperature profile will complete this case as per aforementioned simulations A to E. The pathlines show that air exits from both side and back outlet. As a result, air does not circulate sufficiently as if it does in case E therefore, reduction in the cooling performance is observed. Again the hard drive surface presents high local temperatures due to the lack of air circulation. However, it is concluded that the chassis temperature requirements are satisfied.

6. Conclusions

Based on the useful results of the validation process, the six cases simulated lead to the following conclusions:

- The passive cooling techniques are not able to cool neither the chassis, nor the processor as they produce high temperatures which affect the functionality of the hardware operation.
- Larger fans with fixed flow rate behave better than two small fans used in the passive cooling schemes.
- The best examined case for the optimum processor thermal attitude was the one with the side fan blowing air directly to the processor.
- Secondary devices on the motherboard are cooled without any problems in all active cooling cases. However, most active cooling cases revealed problems at the cooling of the hard disk source since it exceeded the manufacturer's operating conditions. The installation of the hard drive in a fin-type case would give a better chance to overcome this malfunction.
- The side-fan section whether it should be fan of air inlet or outlet seems to be in favour of the fan inlet. This option yields the best temperature conditions for the processor.
From the above arguments, it is clear that the flow field inside a computer chassis is very complex and is affected by the processor fan and the way that it spreads the air at the rest areas. In combination with the inlet fans, turbulence may occur so that the air could flow in all zones and cool the parts effectively.

It is obvious that there are wide opportunities for future scientific research on this very interesting scientific field, especially in the cooling optimization process by choosing the correct accessories as well as their optimised position in such a complex environment as a computer chassis. Moreover, there is enough space for research in more complicated cases for a computer chassis, such as positioning the power supply at its lower part or putting wires and changing the flow field inside it. A more thorough CFD parametric analysis, including time-dependent parameters on the turbulence and heat transfer simulation could be carried out, in order to gain more knowledge and taking measurements in full scale would also be very interesting, which could be difficult to monitor all the parameters affecting the phenomenon in reality.

References


