Imminent Needs in Future Development of Air Cooled Microprocessors Heat Sinks

Ganescu, V., PhD, Harrisburg Area Community College, Harrisburg, PA – U.S.A., Technology Division, vcganesc@hacc.edu
Pascu, A., PhD, University "Politehnica" Bucharest, Romania, Department of Mechanical Engineering, apascu@ing.pub.ro

Abstract

This study reiterates the fact that revolutionary heat sink geometries, materials and overall exponentially higher performing alternatives are continuously and highly needed as applied to the air cooling of a typical computer system microprocessor. Attention was focused on forced convection regimes of operation and from a system level approach. Minor improvements in the performance of air cooled microprocessor heat sinks via typical small design improvements are discussed. Laminar convection and constant heat dissipation were looked at. The CFD simulations exemplified were completed for several power levels and ambient air characterized by a Pr = 0.71. The numerical results presented coincided in large with the experimentally derived documented data. In conclusion, the authors stress the fact that leading-edge alternatives in air-cooled heat removal of such applications are imperiously necessary.

Keywords: heat sink, convection, CFD, microprocessor, thermal resistance, numerical model

1. Introduction

In every computing device, the central processor unit must be cooled to limit its temperature to the manufacturer's maximum allowed value. Current typical limiting CPU die surface (or CPU case) temperatures are in the range of 60^0C to 90^0C depending on different CPU packages. In most instances the CPU is still cooled simply using an attached heat sink with the heat transferred to the air supplied by a fan. Several methods have been used to supply air to the heat sink. One is an "active" heat sink, for which a small fan is directly mounted on the heat sink. Although an active heat sink has excellent cooling capability, the active heat sink fan adds to system noise, has questionable reliability for long-term operation [2], and is known to increase EMI emitted from the CPU. Another approach is to install a fan in the rear wall of the case and blow air into the case. A problem with the case fan is that most of the air delivered by the case fan bypasses the heat sink.

Cooling current alternatives for PC systems also involve "ducts" aimed to direct the air flow in the vicinity of the heat sink, providing this way a relatively high heat transfer coefficient on the heat sink fins ([1]). The cost of such ducts is small compared to that of the heat sink. Further, the cost of the ducted heat sink is approximately 50% lower that that of an active fan - heat sink.

Efforts to simulate 3D computational modeling of air-cooled desktop computers can be found in several publications. Linton [3] and Linton and Agonafer [4] simulated an entire desktop PC with one fan using Phoenics code. Lee and Mahalingam [5] used the Flowtherm code to simulate detailed flow and temperature fields within a computer chassis having 2 fans. They also measured the temperature data at selected locations to verify the computational results. Similar work was reported by Wong and Lee in [2]. They analyzed passive heat sink solutions for a multiprocessor system. Since the analyzed system had complex chassis structures, they used three successive modeling stages (system level, board level, and package level) to expedite their numerical analysis procedure with limited computing resources. Bash and Patel [6] used Icepak to analyze a "mock-up" chassis of a quad processor server with four fans. In their study they demonstrated a "thermo-volume resistance" technique, where a computer component (mainly heat sinks) was represented as a lumped volume, which was characterized by the experimentally generated thermal and hydraulic resistance curves versus volume flow rate through the component. Chang, Yu, and Webb [1] also analyzed a ducted design employed to direct the air flow over the CPU and then to the inlet air vents of the power supply.

The present article compiles results of several numerical studies (and the corresponding lab setting validations) addressing a small form factor chassis, the Micro-ATX, emphasizing current limitations of “standard” cooling solutions as well as the need for novel future cooling solutions of such devices.

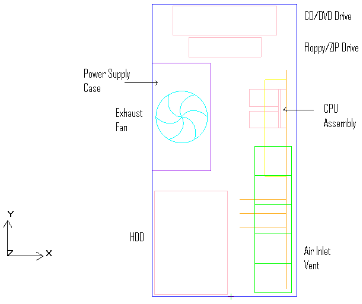


Fig. 1 – Frontal view of chassis layout identifying the major components of the system

The computational fluid dynamics software package Icepak (by Fluent) was used to simulate the air flow and temperatures through the chassis.

2. CFD Modeling Methodology

The CFD model layout of the present desktop computer is shown in Fig. 1. The chassis components shown in the figure are simplified representations of their physical counterparts.
Definition of a sufficiently fine mesh to describe details of every component in the chassis was not necessary, because it would have contributed little to the final computational results. The modeling procedure for each component is discussed below.

1.Heat Sink and Processor Representation

The geometry of the subject heat sink is the typical vertical fin heat sink.

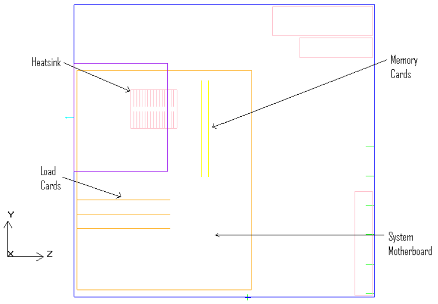


Fig. 2 – Side view of the chassis

Note that the base plate thickness of the heat sink is considerably greater than the fin thickness. The thick base plate serves as a spreader to conduct heat from the CPU die to the full base area of the finned heat sink.

The numerical analyses were performed for a constant heat flux over the CPU solid block case. Zero heat loss was assumed from the bottom side of the PGA package.

The results showed that the highest case temperature occurs near the top central area of the CPU case because of the thermal spreading resistance through the heat sink base plate thickness.

In actual systems, heat sinks are normally attached to the CPU case via a steel spring clip. However, the clip was not included in the numerical model, because it was assumed to have negligible contribution to the computational results.

Radiation heat transfer from the CPU heat sink was not considered in the present numerical model. Because radiation was not included, the present study is relatively conservative.

2. Power Supply Representation

The power supply is a very complex geometry within the case, which includes numerous electronic components, wiring, and heat sinks. It was not deemed practical to numerically model this complex geometry. Rather, the power supply was modeled as a volume hydraulic resistance. Here, the effect of any resistance is to be modeled as a pressure drop through its volume.

The pressure drop resulting from a resistance is to be calculated either by the approach-velocity method or by the device-velocity method.

Because the resistance to the fluid flow due to a volumetric resistance may be different in each of the three coordinate directions, one must provide the loss coefficient and the method to calculate the pressure drop in each direction. At any rate, the difference between the approach-velocity and device-velocity methods is in the velocity used to compute the pressure drop. The device velocity is related to the approach velocity by:

(1) $v_{dev}=v_{app}/A$

where, A is the free area ratio (ratio of the area through which the fluid can flow unobstructed to the total planar area of the obstruction). Also, must mention the fact that the loss coefficients may be obtained either from experimental measurements, computational measurements or from published data available for many grill and vent configurations.

Due to the fact that the power supply is located downstream from the CPU, heat dissipation within the power supply case can only have a minute effect on the maximum CPU case temperature. Therefore, in the present work, heat dissipation within the power supply case was not taken into consideration.

3. Power Supply Fan

The exhaust model of a fan was implied for this part of the power supply. The flow rate through the fan is governed by its characteristic curve and was specified by a piecewise-linear (volume flow – pressure points) curve. The linear characteristic curve does not always adequately approximate the true fan characteristic curve over its entire operational range, so it is best to specify the actual fan curve, if possible.

In our simulations the (exhaust) fan was defined by its characteristic curve; the built-in text editor feature was used to specify the volume flow / pressure pairs of this piecewise-linear curve.

Fan static pressure is computed by:

(2) $p_{fs}=p_{discharge}-p_{intake}$

where, p_{intake} is the pressure averaged over the face of the intake side of the fan, and $p_{discharge}$ is the pressure over the face of the discharge side of the fan. For an exhaust fan, $p_{discharge}$ is the ambient pressure, and p_{intake} is computed by the CFD software (the default ambient pressure is zero and has been proven to be satisfactory in almost all situations).

In actual system operation, the fan speed (note that a fixed diameter fan model was implemented) is controlled by a temperature sensor in the power supply case. This reduces air flow and fan noise when the system is not fully loaded.

4. Other Conditions

Based on data available in literature, it was safe to assume that heat was also dissipated from components (e.g.: chipset) installed on the motherboard surface. Since the motherboard, DIMM cards, and the plug-in load cards were modeled as plates in Icepak (rather than PCBs), our numerical models implied that the heat mentioned above was dissipated evenly from the motherboard surface.

By definition, plates are solid, flat, rectangular objects that are impervious to fluid flow and able to conduct heat in either direction (they posses a thickness, as well). From a different perspective, we have to point out the fact that all surfaces of the plates were considered hydrodynamically smooth and the standard no-slip boundary conditions were applied.
The conducting thin model was considered for our plates (in order to keep a realistic, the plates were assigned an effective thickness, though).

The CPU assembly and the mass storage systems (HDD, floppy/ZIP, CD/DVD) were modeled as solid blocks, and the no-slip condition for fluid velocity applies to all blocks' surfaces. Once again, since the storage devices are not located in the proximity of the CPU assembly or upstream from the CPU, heat dissipation within these objects (hollow blocks, where only side characteristics are relevant to the fluid flow) has minimal effect on the CPU case temperature.

A hydraulic resistance concept was used in the numerical model to account for flow losses across the air vents. This loss coefficient was set based on data from Idelchik [12], which characterizes the pressure drop versus inlet air velocity by an equation of the form $\Delta P=K(\rho v^2/2)$, where ρ is inlet air density, and v is inlet air velocity. By default, the external fluid entering the computational domain is at the ambient temperature specified in the initial problem setup.

3. Numerical Model

Hexahedral meshing schemes were called for. Depending on the restrictions imposed, the solutions were found to be converging anywhere between 175 iterations (for standard vent placement and no deflectors) and 550 iterations. The Reynolds, and Peclet numbers characterizing the flow were found to be in the vicinity of 19000 and 13500 respectively.

First order discretization schemes were selected for the characteristic equations of the flow. In order to jump-start the calculations, a small initial velocity was imposed for the Oz axis. The “zero equation” turbulence model was considered.

All numerical simulations were done for 35^{0}C inlet air. The gravitational field was considered for the Oy axis; air buoyancy effects in the y direction were accounted for using the Boussinesq approximation, which incorporates the effects of variable air density into the momentum equation.

The top, bottom, and sides of the chassis were represented as adiabatic boundaries.

4. Results and Discussion

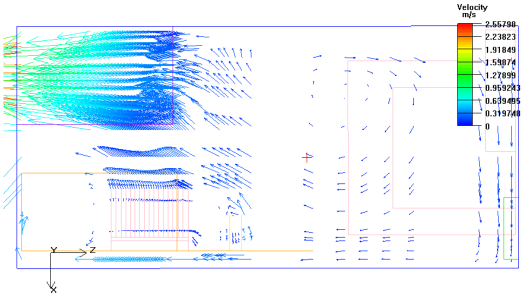


Fig. 3 - Velocity vector profiles

Limitation 1: Figure 3 shows velocity plots at a cross section of the chassis in the X-Z plane taken through the center of the heat sink. It can be seen that most of the air bypasses the heat sink and goes directly to the power supply vent holes (this aspect is also visible by analyzing the dye-trace represented in fig. 4). Due to the *significant air bypass around the heat sink*, the heated air streams from the load cards (when loaded, of course) would have very only a small impact on the CPU heat sink temperature.

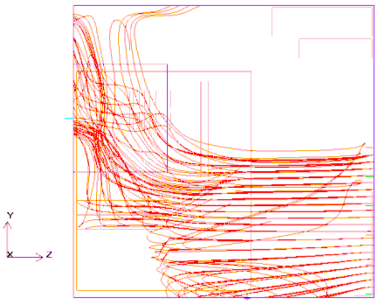


Fig. 4 - Particle trace throughout the computational domain

Limitation 2: Also, a re-circulation flow pattern is clearly observed at the right upper corner of the chassis represented in figure 4. It was believed that by forcing some of this re-circulated volume of air back towards the CPU / heat sink, the maximum temperature of the CPU assembly may be reduced. The research conducted herein showed that potential deflectors (fig. 5) had some impact on the CPU’s maximum temperature; unfortunately, due to the relative low instantaneous velocity characterizing the re-circulated flow in the area, *the effect of this deflectors design on the cooling process was limited* (maximum CPU assembly temperature dropped to only 74.5^{0}C).

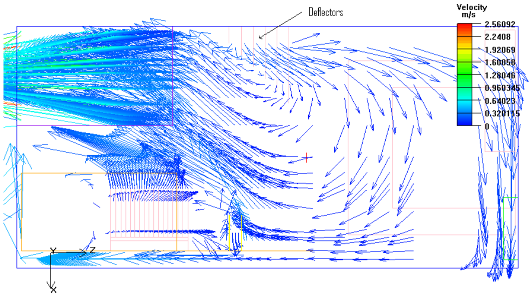


Fig. 5 – Impact of deflectors on flow re-circulation area

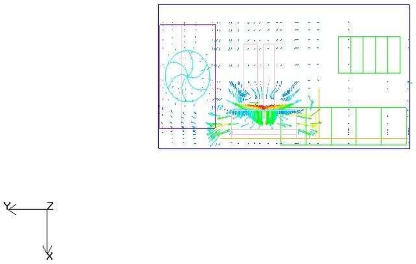
Other designs for the potential deflectors have also been considered but their impact on the CPU’s maximum temperature value was negligible.

Limitation 3: On the other hand, it was observed that a *30% obstruction (e.g.: user’s negligence, accumulation of dust, etc.) of this single air inlet vent system caused a minimal increase in the maximum temperature value of the CPU assembly* (76.9^{0}C). From a different perspective, the search for an alternative position and/or size of the air inlet vent that may improve the cooling characteristics of this single fan - single heat sink design did not return any positive results.

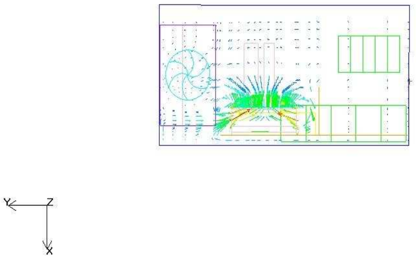
Limitation 4: There has been some controversy in the literature about the "correct" direction of airflow over a CPU's heat sink. This refers to the fan installation. Should the fan be "blowing" or should it be "sucking". Documented arguments make good cases for either path, and many have had good experiences with both configurations.

It seems that *there isn't a "right" or "wrong" way to have the fan installed.* The question of fan airflow is one that can only be answered by the system designer or the person performing an upgrade. The engineer has to consider the system in which the subject fan-heat sink cooling assembly will be placed and examine the other factors that can help and/or hurt the cooling abilities.

When comparing our microprocessor cooler (blowing fan configuration) with a sucking fan / heat sink cooling system a difference of approximately 2.4802^{0}C (revealed by the numerical simulation), 2.4^{0}C (as provided by the heat sink sensor) respectively, was obtained in the microprocessor's temperature value, meaning that a sucking fan/heat sink cooling assembly would work better in this particular PC configuration.



(a)



(b)

Fig. 6: Velocity Profiles for:
(a) blowing fan configuration, (b) sucking fan configuration

Possible explanation of the results involving a sucking fan – heat sink cooling could be drawn by analyzing Figure 6, which compares the velocity profiles of the air flow in the vicinity of the CPU cooling system. When a blowing fan configuration was used (a), the direction of the air flow targeting the cooling of the CPU was opposing the flow sucked in by the power supply exhaust fan, this causing a poorer local ventilation.

The results computed in the current research paper revealed that for a blowing fan, slightly higher temperature values were obtained (for the loaded system case) at the level of the power supply and disk drives, as well.

Limitation 5: *If Copper (mainly because of the higher thermal conductivity value) is to be used for the manufacture of the heat sink, a decrease of about 2% in the value of the characteristic thermal resistance* was observed for these forced convection cases, fact that represents a better heat transfer capacity of the heat sink.

Limitation 6: On the other hand, by keeping the same geometry and physical dimensions but slightly modifying the manufacturing process (according to a different industry standard – where mounting of the heat sink will be done via a standard spring clip) as depicted *in fig. 7, the values of the thermal resistance were found to be decreasing (with an average of 1.5% for the power range studied in the natural convection heat transfer)*, as well.

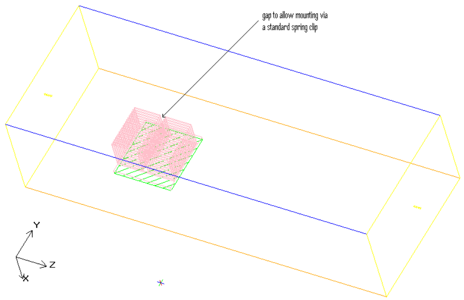


Fig. 7 – Different Pin Design

5. Conclusions

Results of practically validated and documented numerical research were here presented and aimed at emphasizing the current limitations of present cooling solutions implemented in the cooling of desktop computer' MPUs.

The limitations listed herein are just a glimpse of the situation/s at and hand (and by no means comprehensive), stressing yet again the fact that novel, future cooling solutions for such applications are much needed.

Presently, leading-edge alternatives in air-cooled heat removal are increasing. Sintered heat sinks, metal injection molding, 5X performance fans, aluminum extrusions at 25:1 ratio and self-fanning piezoelectric fin heat sinks are visions that are yet to come feasible reality. While many companies are working to develop these next generation products, the authors intended to re-iterate that most enhancements presently implemented in the area of thermal management of microelectronics have become highly limited.

6. Nomenclature

A = free area ratio
c_p = specific heat of air, J/kgK
n = fan speed, rpm/min
p = pressure, N/m^2
v = air velocity, m/s
V = air flow rate, m^3/s
Δp= pressure difference, N/m^2
ρ = air density, kg/m^3

7. References

[1] Chang, J.,Yu, C., Webb, R., 2001, "Identification of Minimum Air Flow Design for a Desktop Computer Using CFD Modeling", Journal of Electronic Packaging, Vol. 123, pp. 225 – 231
[2] Wong, H., Lee, T., 1996, "Thermal Evaluation of a PowerPC 620 Microprocessor in a Multiprocessors Computer", IEEE Transaction - Components, Packaging Manufacturing Technology, Part A, 19, No. 4, pp. 469 – 477
[3] Linton, R., 1991, "CFD Modeling of Electronic Enclosures", ASME - Heat Transfer in Electronic Equipment, HTD – Vol. 171, pp. 95 – 100
[4] Linton, R., Agonafer, D., 1994, "Thermal Model of a PC", ASME – Journal of Electronic Packaging, Vol. 116, pp. 134 – 137
[5] Lee, T., Mahalingam, M., 1994, "Application of a CFD Tool for System Level Thermal Simulation", IEEE Transaction – Components, Packaging Manufacturing Technology, Part A, 17, No. 4, pp. 564 – 571
[6] Bash, C., Patel, C., 1999, "Modeling and Metrology for Expedient Analysis and Design of Computer Systems", Proceedings of 1999 International Systems Packaging Symposium, San Diego – CA, pp. 64 – 70
[7] Intel Application Notes, 1998, "Micro – ATX Chassis Design Suggestions", Version 1.0
[8] Intel Datasheets, 1999, "Intel Celeron at 266 MHz, 300 MHz, 333 MHz, 366 MHz, 400 MHz, 433 MHz, 466 MHz and 500 MHz", Order Number: 243658 – 009
[9] Chu, H., Belady, C., Patel, C., 1999, "A Survey of High Performance, High Aspect Ratio Air Cooled Heat Sinks", Proceedings of 1999 International Systems Packaging Symposium, San Diego – CA, pp. 71 – 76
[10] Incropera, F., DeWitt, D., 1996, "Fundamental of Heat and Mass Transfer", 4th Edition, Wiley – NY
[11] Icepak 3.01, User's Guide
[12] Idelchik, I., 1995, "Handbook of Hydraulic Resistance", Boca Raton, FL – USA, CRC Press