CFD ANALYSIS OF CPU FOR COOLING OF DESKTOP COMPUTERS

Mayank Srivastava[1], Pardeep Bishnoi[2], Dr.M.K.Sinha[3]

[1]Phd Scholar, Mechanical Department, Nit Jamshedpur, Jharkhand, (India)

[2]Associate Professor, Mechanical Department, Nit Jamshedpur, Jharkhand, (India)

ABSTRACT

The life of the electronic components strongly depends upon the heat transfer which is generated within that component or the cooling of that device. For this reason the need for forced air cooling is the main factor that should be consider at the starting phase of electronic system design. The one of the critical parts of any electronic device is heat sink, which dissipates heat from electronic device to surrounding. Thus the choice of an optimal heat sink depends upon the number of geometric parameters like fin height, fin length, fin thickness, number of fins, fin materials etc. With the help of commercially available computational fluid dynamics software's Icepak and Fluent, we can easily analyse the cooling effect of CPU.

Keywords: Cfd Analysis, Cpu, Temperature Contour, Velocity Contour, Fluent

I. INTRODUCTION

All electronic hardware depends on the stream and control of electrical current to perform a mixture of capacities. At whatever point electrical current courses through a resistive component, heat is created. With respect to suitable operation of the hardware, heat dispersal is a standout amongst the most discriminating angles to be considered when planning an electronic gadget. Heat must be uprooted with a specific end goal to keep up the nonstop operation. With different degrees of affectability, the unwavering quality and the execution of every single electronic gadget are temperature subordinate. For the most part they bring down the temperature and the change of temperature as for time, the better they are. Unadulterated conduction, regular convection or radiation cool the segments to some amplify though today's electronic gadgets require all the more capable and muddled frameworks to adapt to heat. Thusly new heat sinks with bigger broadened surfaces, exceedingly conductive materials and more coolant stream are keys to diminish the heat.

M.Saini et al. [1] performed a parametric study and determine heat rejection limit of air-cooled plain fin heat sinks for computer cooling. They obtained 103.4W as a limit for plane fin heat sink. In R.Arularasan et al.[2] research work, optimal design of the heat sink have been carried out on a parallel plate heat sink taking geometric parameters like fin height, fin thickness, base height and fin pitch with a constant length and width of a heat sink using CFD study. They also validate their model with experimental results and both results were compared. R.Mohan et al. [3] investigated fin materials, number of fins, their distribution and base plate thickness for enhancing the heat dissipation rate from CPU. C-W. Yu et al.[4] Performed a CFD analysis to

define chassis and CPU cooling requirement for a chassis which dissipates 313 W, with an 80W CPU. The cooling for AGP & PCI cards is improved with PCI side vent and baffle. T.Tirupati et al. [5] analysed the effectiveness of tangential convective cooling for the protection of structure from intense, localized thermal loads. A.Bar-Cohen et al. [6] proposed a new concept for cooling the electronic component using the copper-base heat sink. The thermal performance and temperature distribution for the heat sink were analysed and present a procedure for optimizing the geometrical design parameter. To investigate flow and conjugate heat transfer in the copper-based heat sink , a three dimensional model is developed. Ilker Tari et al. [7] analysed a notebook computer management system using CFD package. The active and passive paths which are used for heat dissipation are examined for different steady state operating condition. Ilker Tari et al. [8] investigated CPU cooling in a complete computer chassis with three different heat sink and performances were compared. The effect of mesh resolution, turbulence model selection, convergence criteria and discretization schemes were investigated to find the best result at least computational cost. The heat sink temperature difference results and specific thermal resistances were compared with the available experimental results. R.Biswas et al.[9] used computational software Icepak to study the airflow in a compact electronic enclosure. The aim of their study was to investigate the pressure loss due to the presence of the inlet and outlet grilles. C.W.Argento et al. [10] studied system level electronic packaging thermal design computationally and compared the result with experimental data. After that they worked on redesign of an inlet plenum. R.L.Linton et al. [11] compared the result of complete CFD modelling of a heat sink with the experimental data. They present a technique for representing the heat sink in a coarse manner for less time consuming solutions. Their coarse model agrees well with the detailed model without losing the characteristics of the heat sink. P.Sathyamurthy et al. [12] computationally studied planner and staggered heat sink performance with Fluent. The computational and experimental results were quite satisfied. They found that the thermal performance of staggered fin configuration is better than planner fin configuration. However the pressure drop requirements for the staggered fin heat sink was greater than those for the planar case.

In this study we are analyzing the cooling effects of the CPU by modelling the geometry numerically by commercially available computational fluid dynamics software's Icepak and Fluent. The main aim of this paper is to identify a cooling solution for the desktop computer which uses 80 W CPU and find the way to increase the cooling rate. The design is based on a total power dissipation of 252 W. The modelling is carried out by solving the governing equations for a flow within a closed domain. The turbulent flow is considered and finite volume method is used to discretize the governing equations. Radiation heat transfer is neglected, the flow is considered as steady state, viscous work is neglected and boussineq approximation is valid within the domain. The temperature and velocity contour is then plot in post processing stage and is found that the maximum temperature and velocity inside the CPU chassis falls within the fixed limit.

II. MATHEMATICAL FORMULATION

2.1 Assumptions

- Radiation heat transfer is neglected,
- The flow is considered as steady state,

- Viscous work is neglected,

- Boussineq approximation is valid within the domain.

Governing Equations Time-independent flow equations with turbulence are solved. The viscous dissipation term is omitted. Therefore, the governing equations for the fluid flow and heat transfer are the following form of the incompressible continuity equations, Navier – stokes equations x-y and z direction momentum, and energy equations together with the equation of state.

Continuity equation:

$$\frac{\partial(u)}{\partial x} + \frac{\partial(v)}{\partial y} + \frac{\partial(w)}{\partial z} = 0 \qquad \qquad \text{.... (1)}$$

X direction Momentum:

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \gamma\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right] \qquad \text{.... (2)}$$

Y direction Momentum:

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \gamma\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right] - \rho g \qquad \text{.... (3)}$$

Z direction Momentum:

$$u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + \gamma\left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right] \qquad \text{.... (4)}$$

Energy equation:

$$\rho C\left[u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}\right] = k\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right] + \dot{q} \qquad \text{.... (5)}$$

Equation of state:

$$\rho = \rho_m\left[1 - \beta(T - T_m)\right] \qquad \qquad \text{.... (6)}$$

Where,

$$T_m = \frac{T_h + T_c}{2} \qquad \qquad \text{.... (7)}$$

III. NUMERICAL SIMULATION

3.1 CFD Calculation

In CFD calculations, there are three main steps.

- Pre-Processing

- Solver Execution

- Post-Processing

Pre-Processing is the step where the modelling goals are determined and computational grid is created. In the second step numerical models and boundary conditions are set to start up the solver. Solver runs until the convergence is reached. When solver is terminated, the results are examined which is the post processing part.

3.2 Computational Domain

The computer chassis is the computational domain as shown in Fig. 1. It is a 3-D model of a common PC chassis in dimensions of H x W x D = 444 x 424 x 187 mm. Since the aim of this analysis is to investigate the temperature distribution on CPU heat sink. There are so many components attached on the computer chassis. The complete geometry of cabinet included all components created in Ansys-Icepak software. The heat dissipation rates, material of components are listed in Table 1. The detailed model of geometry is shown in Fig.1. Since the aim of this analysis is to investigate the temperature and velocity distribution on CPU heat sink.

Table 1 Materials Thermal Property

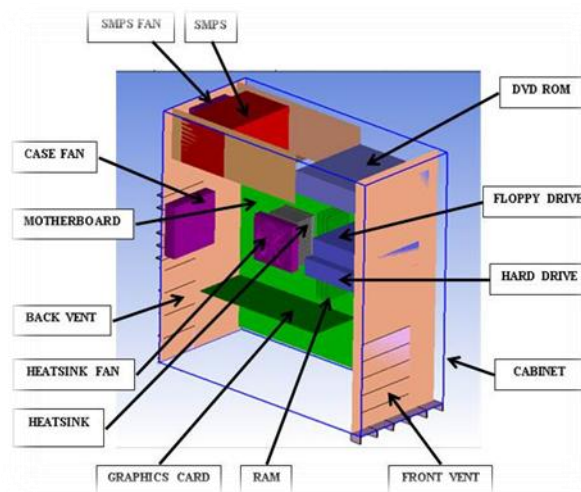| Material	ρ (Kg/m^3)	C_p (J/Kg-K)	k (W/m-K)
Copper	8979	381	387.6
Aluminium	2719	871	202.4
Fr-4	1250	1300	0.35

Figure 1: Detailed Computational Domain

3.3 Mesh Generation

The second part of pre-processing is the mesh/grid generation as shown in Fig. 2. Mesh is the main part to get the exact or high quality solution. In Icepak, there are three types of meshing algorithms available. These are Hexahedral Cartesian, Hexahedral Unstructured and Tetrahedral mesh. In this analysis Hexahedral Unstructured mesh is used. All details are given in Table-2.
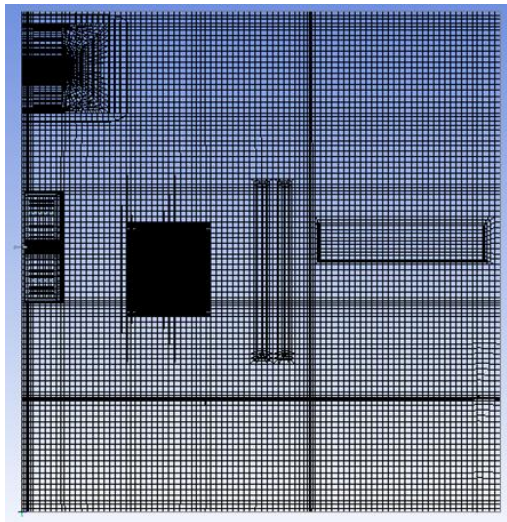
Figure 2: Meshing of Computational Domain

3.4 Initial Conditions

Initial velocities and temperatures are defined. Initial pressure is taken to be the ambient pressure. The initialization of the model is important for convergence. If the initial conditions are poor, then it takes longer to converge or it may even result in divergence.

Table 2: Meshing Detail

Mesh Type	Mesh Unit	Mesh parameters
Hexa-unstructured	Millimetres	Medium
Number of Elements		12486
Number of Nodes		18896

Fluent 6.1 uses the finite volume method. Interpolations have to be done for discretization. There are numerous schemes, and the easiest one is the first-order upwind. The advantage of this scheme is that it converges easily. In our model we use first order and second order schemes for pressure, momentum, temperature and RNG k-ε turbulence model is used for turbulence kinetic energy and turbulence dissipation rate.

Table 3: Basic Parameters

Variable Solved	Radiation	Flow Regime	Ambient Conditions	
Flow (Velocity & Pressure	Off	Turbulent (RNG k-ϵ model)	Temperature	35 ∘C
Temperature			Default Fluid	Air
Basic Settings		**Advanced Settings**		
Iterations	1000	Discretization Scheme		
Convergence Criteria		Pressure	Standard	
Flow	0.001	Momentum	First	
Energy	1e-7	Temperature	First	
Joule heating	1e-7	Precision	Double	

IV. RESULT AND DISCUSSION
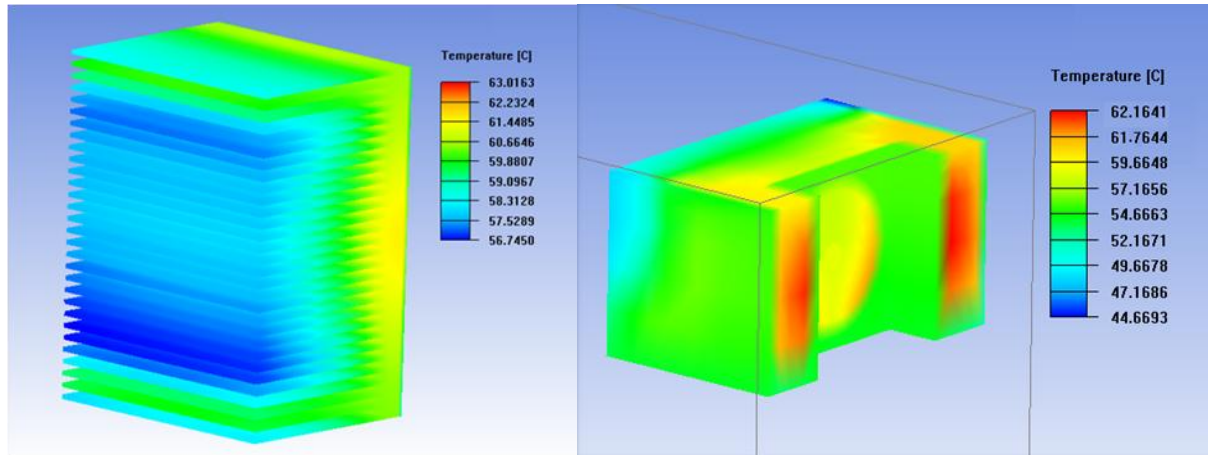
4.1 Temperature Contour

Figure 3: **Temperature contour of Heat sink and SMPS**

Fig. 3 shows the temperature variation within the heat sink. The processor maximum temperature is at the range of 63° C which affects significantly the functionality of the device. The fins dissipate heat quite well, which is generated by CPU. Referring to the power supply unit, it is found 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 SMPS temperature is at very good level.
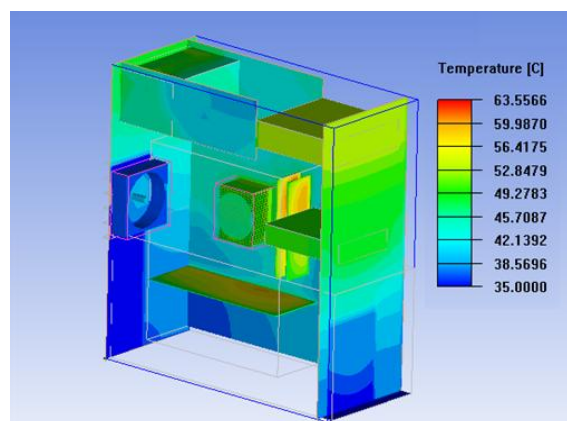

Figure 4 Temperature Contour of desktop chassis and its components

As shown in Fig. 4, Temperature of lower part of chassis is minimum because fresh air is coming in from the lower vent .When the air absorb heat from lower components its density reduces and that hot air directly exit from outlet fan .
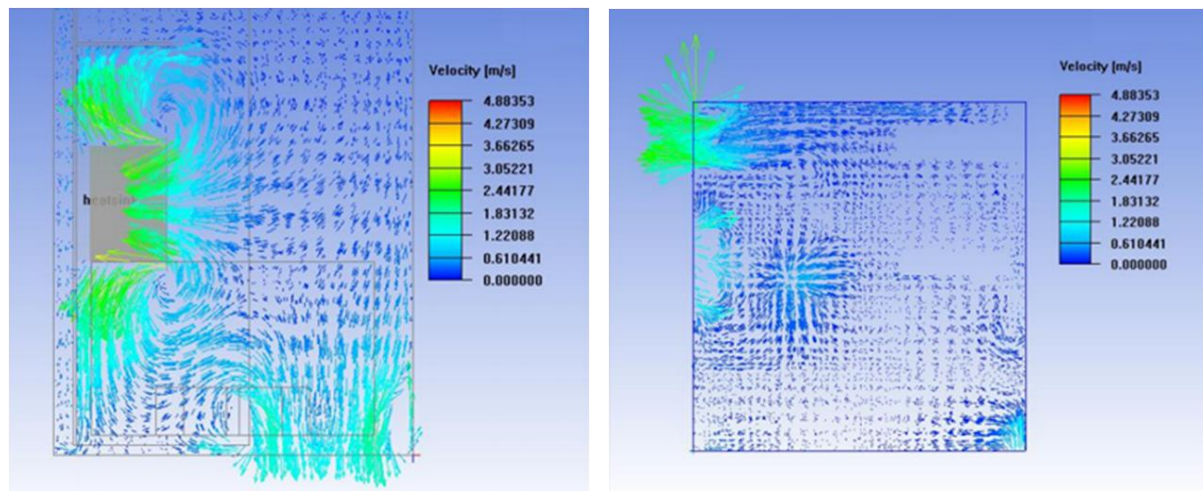
4.2 Velocity Contour

Figure 5: **Velocity contour inside computational domain**

As we can see in Fig. 5 air circulates around the heat sink. The heat produced by the heat source is dissipated by the fin by conduction. When air gets contact with the fin surface, it goes up and a fresh stream of cool air circulates near the heat sink. The space available for air circulation inside the PC chassis is limited which decreases the heat dissipation rate therefore performance is decreased.

V. CONCLUSION

Too many fins on heat sink base is not a solution for dissipating the heat by the heat sink since they may prevent the passage of air coming from the fan to the hottest centre parts of the heat sink. The heat transfer rate is strongly depends upon the placement of heat sink on heat source and the fan poisoning inside the chassis. The space inside the CPU should be sufficient to circulate the hot air which will increase the heat transfer rate. First order discretization scheme is enough if the grid is dominated by hexahedral cells. Convergence is faster with this scheme. RNG k-ε turbulence model is suitable for analyzing fluid flow and heat transfer in a computer chassis. Radiation effects are ignored due to the domination of forced convection and relatively low temperature differences inside the chassis. Convergence must be assured by checking the residuals and also the temperature monitors. It is necessary to let the residuals to drop more than three orders of magnitude.

In this analysis we have considered a three dimensional case with a single heat sink placed on the CPU, which is the main heat dissipation source. There exists much more scope to extend this work. For example, the analysis can be carried out with multi heat sink, which can be mounted on some other heat source. The heat generation rates from other components are very less as compared to the CPU but near in future as the size of chassis will decrease & the load, working hours of electronic gadgets will increase, components dissipate more heat. Improvements on heat sink designs are possible via CFD.

REFERENCES

[1] M.Saini and R.L,Webb, Heat rejection limits of air cooled plane fin heat sinks for computer cooling, IEEE Transaction Component Package Technology, 26(1), 2003, 71-79.

[2] R.Arularasan and R.Velraj, CFD analysis in heat exchanger for cooling of CPU, International Journal of the computer, the internet and management, 16(3), 2008, 1-11.

[3] R.Mohan and P.Govindarajan, Thermal analysis of CPU with variable heat sink plate thickness using CFD, International journal of the computer, the internet and management, 18(1), 2010, 27-39.

[4] C-W. Yu and R. L. Webb, Thermal Design of a Desktop Computer System Using CFD Analysis, 17 IEEE SEMI-THERM Symposiums.

[5] T. Tirupati and A. G. Sarwade "CFD Analysis of Convective Cooling for a Heated Skin." Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, DECEMBER-2010.

[6] A.Bar-Cohen and M.Iyengar, " Least energy optimization of air cooled heat sinks for sustainable development ", IEEE Transactions on Components, Packaging, and Manufacturing Technology, Vol. 26, no.1, pp. 16-25, March 2003.

[7] Ilker Tari and Fidan Seza Yalcin " CFD Analysis of a Notebook Computer Thermal Management System and a Proposed Passive Cooling Alternative.", IEEE Transactions on Components and Packaging Technologies Vol.33, No.2, JUNE-2010

[8] Emre Ozturk and Ilker Tari "Forced Air Cooling of CPU's with Heat Sinks: A Numerical Study." IEEE Transactions on components and packaging technologies, Vol. 31, No. 3, SEPTEMBER 2008.

[9] Rebecca Biswas, Raghu B. Agarwal, Avijit Goswami, Vivek Mansingh, "Evaluation of airflow prediction methods in compact electronic enclosures," Fifteenth IEEE Semi-Therm Symposium, pp. 48-53,November 1999.

[10] Christopher W. Argento, Yogendra K. Joshi, Micheal D. Osterman, "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, Vol.19, No 2, pp. 248-257, 1996.

[11] Ronald L. Linton, Dereje Agonefer, "Coarse and detailed CFD modelling of a finned heat sink," IEEE Transactions on Components, Packaging, and Manufacturing Technology-Part A, Vol.18, pp. 517-520, No 3, 1995.

[12] P. Sathyamurthy, P.W. Runstadler, "Numerical and Experimental Evaluation of Planar and staggered heat sinks," IEEE Inter Society Conference on Thermal Phenomena, pp.132-139, 1996.