© 2014 IJEDR | Volume 2, Issue 2 | ISSN: 2321-9939

CFD Analysis & Simulation of Pin Fin for Optimum Cooling of MotherBoard

[1]Deepak Gupta,[2]Momin Nausheen, [3]A.D. Dhale

[1,2]Final year students of mechanical Engineering, [3]Associate Professor
Shivajirao S. Jondhale College of Engineering, Dombivli (E),Thane-421 204,India.
[1]dpak8879@gmail.com, [2]momin.naush786@gmail.com, [3]atuldhale32000@yahoo.com

Abstract - Since the development of the first electronic digital computers in the 1940s, the effective removal of heat has played a key role in ensuring the reliable operation of successive generations of computers. As day by day our work load is increasing and hence our use of computers are increasing which leads to increase in heat and hence consequence is sabotage of computer with our work. So it's a great challenge to packaging engineers to remove the heat generated by the chip efficiently. Many researches are going on in this direction for the past few decades. In the last decade or so CFD simulations have become more and more widely used in studies of electronic cooling. Validation of these simulations has been considered to be very important. The computational fluid dynamics is concentrated on the natural air cooling of the CPU using a heat sink. This paper utilizes CFD (using FLUENT) to identify a cooling solution for a desktop computer, which uses an 5 W CPU. The design is able to cool the chassis with heat sink attached to the CPU is adequate to cool the whole system . This paper considers the optimal rectangular fin heat sink design and cylindrical fin heat sink design with aluminium base plate and the control of CPU heat sink processes.

I. INTRODUCTION

TODAY'S rapid IT development likeinternet PC is capable of processing moredata at a tremendous speed. This leads tohigher heat density and increased heatdissipation, making CPU temperature riseand causing the shortened life, malfunctionand failure of CPU. The failure rate ofelectronic components grows as anexponential function with their risingtemperature. Power dissipation would be amajor bottleneck to development of themicro electronic industry in the next 5 to 10years. The performance level of electronicsystems such as computers are increasingrapidly, while keeping the temperatures ofheat sources under control has been achallenge.

In recent years, as the heat loads haveincreased, better heat conductors such ascopper plates are used to improve thespreading of heat from heat sources into theheat sinks. To meet the next generation, CPUneeds the thermal requirements with a lowprofile heat sink. Therefore new heat sinkswith larger extended surfaces, highlyconductive materials and more coolant floware keys to reduce the hot spots. To meetthese constraints, CFD is a good approach toexplore various design alternatives quicklywith reasonable accuracy.

For high volumemanufacture, the heat sinks should beinexpensive, reliable and fit to otherconstraints in the manufacturing process.The modified fin geometry with air coolingis more effective and economic, since thewater cooling requires water pump, aseparate cooling system for coolant and aseparate flow study.

II. LITERATURE SURVEY

[1] R.Mohan and Dr.P.Govindarajan. "Thermal analysis of CPU with composite pin fin heat sinks",International Journal of Engineering Science and Technology,Vol. 2(9), 2010, 4051-4062. The computational fluid dynamics is concentrated on the forced air cooling of the CPU using a heat sink. This paper utilizes CFD to identify a cooling solution for a desktop computer, which uses an 80 W CPU. In this study a complete computer chassis with different heat sinks are investigated and the performances of the heat sinks are compared.

[2]Konstantinos-Stefanos P. NIKAS* and Andreas D.PANAGIOTOU."Numerical Investigation of Conjugate Heat Transfer in a Computer Chassis" Columbia International Publishing Journal of Advanced Mechanical Engineering (2013) 1: 40-57 doi:10.7726/jame.2013.1004.

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 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).

III. PROBLEM DESCRIPTION

This paper investigates a well-known commercial application solved by CFD technique by modelling 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 considerationsare applied. Position, geometrical characteristics and the number of fins will be altered to study thethermal effects. The numerical simulation of the computer chassis investigates a few cases of activeand passive cooling scheme in a standard geometry layout. The number of fins and theirgeometrical characteristics will be altered in order to identify which is the best case that producedthe maximum heat transfer results.

IV. COMPUTER CHASSIS MODEL

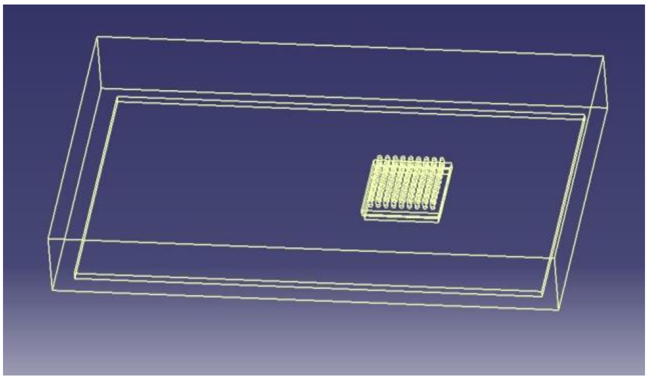

Fig 1 CPU Chassis Model

The model studied consider of following object

4.1 Computer Chassis

The computer chassisconsists of inlet which allow air for cooling and discharges hot air through outlet.It also consists of printed circuit board(PCB), heat source, and heat sink with base plate.

4.2 PCB

A **printed circuit board (PCB)** mechanically supports and electrically connects electronic components. PCBs require the additional design effort to lay out the circuit but manufacturing and assembly can be automated. Manufacturing circuits with PCBs is cheaper and faster than with other wiring methods as component are mounted and wired with one single part. Furthermore, operator wiring errors are eliminated.

The PCB is made of material "Board"
Density = 1250kg/m3; cp = 1300 J/kg K; Thermal conductivity 0.35 W/m K

4.3 Heat source

We need to set both the material, AND the thermal power (5W) dissipated by this component .The heat source volume is $4.332x10^{-6}$ m^3. Hence, the volumetric source is 1154201 W/m3 .Create 1 constant energy source with the above value.

The Heat source is made of material "Component"
Density = 1250kg/m3; cp = 1300 J/kg K; Thermal conductivity 0.35 W/m K

4.4 CPU heat sink

The most important object in the computer chassis is the heat sink various model have been created for the investigation of cooling characteristics of different heat sink .in every model, only the heat sink geometry is changed . All the other objects remained unchanged .This is most complicated object of all model. Therefore it takes more time to create this geometry since there is no CAD geometry available ,some model are created by measuring the dimension of the actual heat sink and or by using the dimension given by the manufactures. **Heat sink** is a passive heat exchanger that cools a device by dissipating heat into the surrounding medium. In computers, heat sinks are used to cool central processing units or graphics processors. A heat sink transfers thermal energy from a higher temperature device to a lower temperature fluid medium. The fluid medium is air.
The Heat sink is made of material "Aluminium"
Density = 1250kg/m3; cp = 1300 J/kg K; Thermal conductivity 0.35 W/m K

V. OBJECTIVES

5.1 To increase the convective heat transfer rate from the solid surface to the surrounding thus cooling the system.
5.2 To compare the total heat transfer rate of a computer chassis with no fins, 45 fins and with 81 fins of cylindrical crossection.
5.3 To compare the total heat transfer rate of cylindrical pin fin and rectangular pin fin.

VI. METHODOLOGY

In CFD calculations, there are three main steps: Pre-Processing, Solver Execution, Post-Processing. Pre-Processing is the step where the modeling goals are determined andcomputational grid is created. In the second step numerical models and boundaryconditions are set to start up the solver.Solver runs until the convergence is reached. When solver is terminated, the results areexamined which is the post processing part.

6.1 Governing Equations

Time-independent flow equations withturbulence are solved. The viscousdissipation term is omitted. Therefore, thegoverning equations for the fluid flow andheat transfer are the following form of the incompressible continuity equations; Navier– stokes equations x-y and z directionmomentum, and energy equations togetherwith the equation of state.

The continuity Equation

$$\nabla(\rho \, \forall) = 0$$

The X, Y, Z Momentum Equations

$$\nabla(\rho \, u \, \forall) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \beta_x$$

$$\nabla(\rho \, v \, \forall) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \beta_y$$

$$\nabla(\rho \, w \, \forall) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \beta_z$$

The Energy Equation

$$\nabla(\rho \, h \, \forall) = -p\nabla\forall + \nabla(k\nabla T) + \emptyset + S_h$$

Equation of state

$$P = \rho \, R \, T$$

Where ρ is the density, u, v and w arevelocity components, ∀is the velocity vector, P is the pressure, β terms are thebody forces, h is the total enthalpy and τterms are the viscous stress components.

6.2 Selection Of Fin

We have selected two types of fin cylindrical and rectangular fin.And changed the number of fin for both the cases for the purpose to CPU.

6.3 Calculation

Using above momentum equation, continuity equation, energy equation all the three cases are simulated with the help of all three equations and applying particular boundary conditions to their corresponding parameters we have calculated all the three cases. With the help of 150 iteration we got approximation results. Following figures shows a 150 iteration graph that display a converging of results which shows a perfectness of solution.

6.4 Case Study

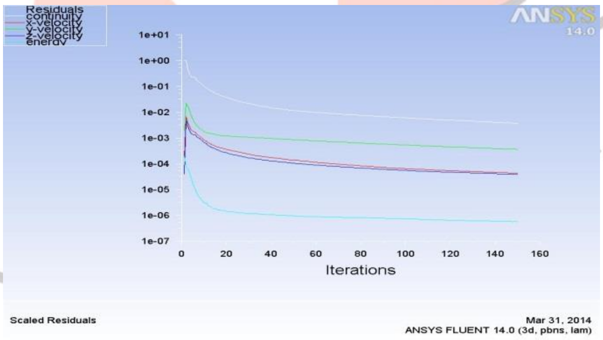

Fig 3 a convergence curve

Table 1

Object name	Material	Net Heat Transfer Rate
Without Fin		0.924
Cylindrical pin Fins(45 fin)	Al	4.398
Cylindrical pin Fins(81 fin)	Al	5.623
Rectangular Fin(81 fin)	Al	4.632

VII. Results and PostProcessing

Eventually fig7.1 shoes a resultsof all cases using column chart here red colour show the heat transfer rate and blue colour show the number of fins.In first case we do not use fins and hence heat transfer rate in case of no heat sink is 0.923.When we used 45 cylindrical fins then heat transfer rate is 4.398W. When we use 81 cylindrical pin fins then heat transfer rate is 5.32W.When we use 81 rectangular pin fins then heat transfer rate is 4.632W.In third and forth case we use same no of fins but in third case we used cylindrical fins and in forth case we used rectangular fins.We can see for rectangular fins heat transfer rate is decreased.Therefore we can see cylindrical fin having more heat transfer rate as compare to rectangular fins.

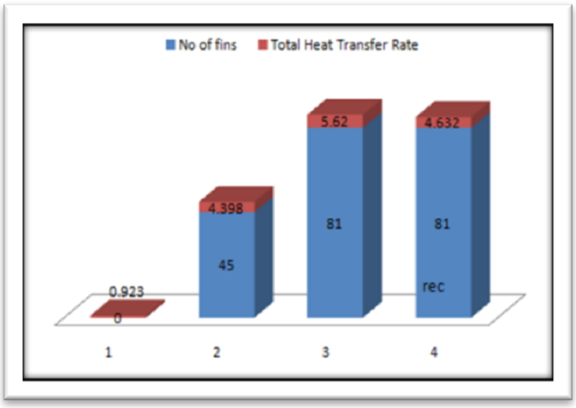

Fig 4 Results of cases using column chart

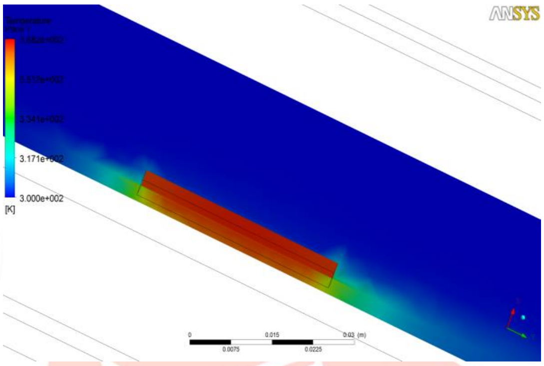

Fig 5 Temperature distribution without heatsink

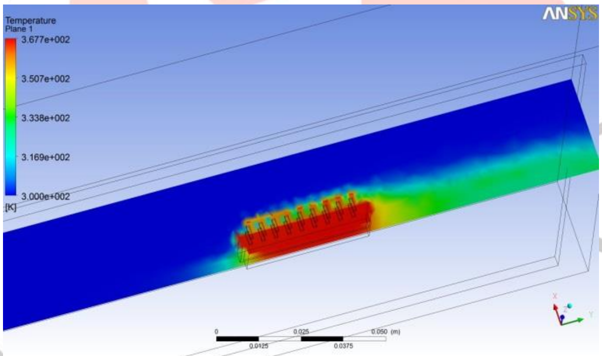

Fig 6 Temperature distribution with 81 cylindrical pin fins

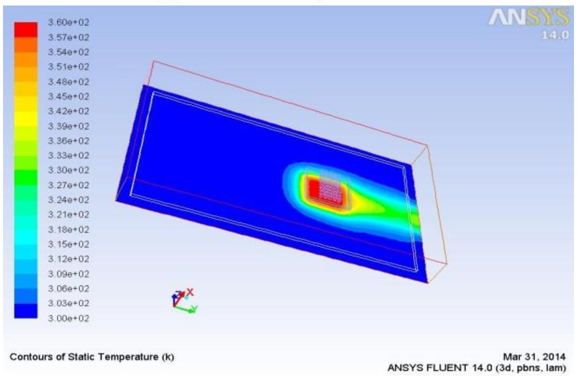

Fig 7 Temperature distribution with 45 cylindrical pin fins

Fig 5 and Fig 6 shows a post processing results of cases without pin fins and with 81 cylindrical pin fins.

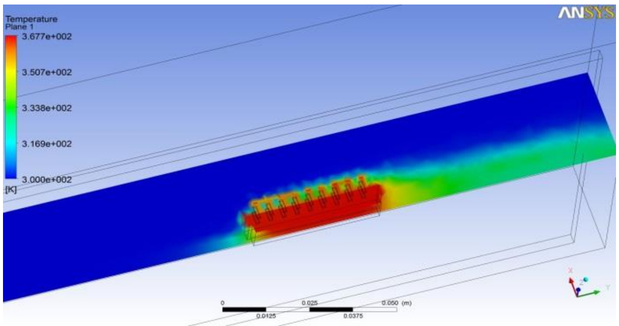

Fig 8 Temperature distribution with 81 Rectangular pin fins

Fig 7 and Fig 8 shows a postprocessing results of cases with 45 cylindrical pin fins and with 81 rectangular pin fins.

VIII. Conclusion

8.1 As the number of fins were increased, the total heat transfer rate also increases i.e the number of fins is directly proportional to the Total heat transfer rate.

8.2The total heat transfer rate of 81 cylindrical pin fins is greater than 81 fins of rectangular cross section.

References

[1] R.Mohan and Dr.P.Govindarajan. "Thermal analysis of CPU with composite pin fin heat sinks",International Journal of Engineering Science and Technology,Vol. 2(9), 2010, 4051-4062.

[2] Konstantinos-Stefanos P. NIKAS* and Andreas D.Panagiotou."Numerical Investigation of Conjugate Heat Transfer in a Computer Chassis" Columbia International Publishing Journal of Advanced Mechanical Engineering (2013) 1: 40-57 doi:10.7726/jame.2013.1004.

[3] Y. Wang and K. Vafai, An experimental investigation of the thermal performance of an asymmetrical flat plate heat pipe, *Int. J. Heat Mass Transfer*, 43 (2000) 2657-2668

[4] D. Kim et al, Thermal optimization of microchannel heatsink with pin fin structures, *Proceedings of the IMECE'03*,IMECE2003-42180 (2003).

[5] H. Jonsson and B. Moshfegh, Modeling of the thermal and hydraulic performance of plate fin, strip fin, and pin fin heat sinks-influence of flow bypass, *IEEE Trans. Compon.Packag. Technol*., 24 (2000) 117-123.

[6] Y. Chung and K. Luo, Unsteady heat transfer analysis of an impinging jet, *J. Heat Transfer*, 124 (2002) 1039-48.

[7] D. Lober, Optimizing the integration of an electronics system into an existing enclosure using CFD modeling techniques, *International journal of microcircuits and electronicpackaging*, 22 (1999) 146-151.