Analysis Of Heat Transfer Enhancement Of Electronic Chip Using CFD

Sanjeev Kumar Gupta, Manish Kumar Rawat, Nitin Kukreja

Abstract: As the technology is advancing day by day, experiments and researches are being carried out to make the cooling of electronic systems such as PCB and projector's internal electronic components more efficient. The motive of this paper is to endow the better cooling environment to the electronic devices. The location of a fan or the blower installed in them is varied to enhance the heat transfer through forced convection phenomenon. If the cooling system of an electronic device is designed effectively the high heat generated at the maximum load can be easily cast off, and hence better reliability can be achieved. Turbulence model is applied to visualize the flow behavior. This analysis is done on the software ANSYS 15.0 (modeling in ICEM CFD, and flow analysis on FLUENT). This analysis will help to predict the better conditions for the manufacturing of cooling models for electronic systems.

Index Terms: Computational Fluid Dynamics (CFD), ANSYS, Forced Convection, Chip Cooling, Heat Transfer

1. INTRODUCTION

Since the birth of the first generation computers in the 1940s, the effective cooling of the system in terms of enhanced heat transfer rate has played a main role to ensure the credibility of the system. The principle of forced convection has become a boon to attain the higher cooling rate. We are seeing that with the changing world the size of the electronic instruments is decreasing drastically day by day. In this scenario, the chips are getting smaller in size but becoming multitasking with a huge no. of functions. In the modern era, the development in IT is very rapid and use of electronic devices is also increasing in the same, so heating of component is a big problem nowadays. CFD is famous especially as a decision support tool for the design of different cooling systems for electronic components. In this paper, the analysis is done for the different locations of cooling fan so that it can be analyzed that what is the effect of changing its location. Many researchers are doing great work in this direction for the past few decades. Saroj Kumar Patra analyzed the flow through a channel via an obstruction for laminar, transient flow and plotted different contours for pressure, temperature, velocity and Nusselt number [1]. Kevin R. Anderson, Matthew Devost, Watit Pakdee, and Niveditha Krishnamoorthy presented CFD simulations of heat and fluid flow behavior in a moderate-sized package of electronics undergoing very large power dissipation [2]. Deepak Gupta, Vignesh Venkataraman, and Rakesh Nimje analyzed the greater heat transfer rate in rectangular plate fins rather than cylindrical pin fins with the same dimension and boundary conditions [3]. M.A.I. Rashid, M.F Ismail, and M. Mahbub found that circular pin fin carbon nano-tube based micro-channel heat sink shows better thermal performance than the rectangular pin fins [4]. Randeep Singh, Aliakbar Akbarzadeh, and Masataka Mochizuki calculated the various design parameters for the design of future laptops based on the miniature loop heat pipe (mLHP) [5].

2 Governing Equations

2.1 Continuity Equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \quad (1)$$

2.2 Momentum Equation

$$\frac{Du}{Dt} = X - \frac{1}{\rho}\frac{\partial p}{\partial x} + \frac{\mu}{\rho}\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right] \quad (2)$$

$$\frac{Dv}{Dt} = Y - \frac{1}{\rho}\frac{\partial p}{\partial y} + \frac{\mu}{\rho}\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right] \quad (3)$$

2.3 Energy Equation

$$\rho c_p\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\left[\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right] \quad (4)$$

2.4 K- ε Equation

$$\frac{\partial \rho \kappa}{\partial t} + div(\rho u \kappa) = div\left[\left(\mu_t + \frac{\rho \mu_t}{\sigma_\kappa}\right) grad\kappa\right] + \rho \mu_t G - \rho \varepsilon \quad (5)$$

$$\frac{\partial \rho \varepsilon}{\partial t} + div(\rho u \varepsilon) = div\left[\left(\mu_t + \frac{\rho \mu_t}{\sigma_\varepsilon}\right) grad\varepsilon\right] + C_{1\varepsilon}\rho\mu_t\left(\frac{\varepsilon}{\kappa}\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{\kappa} \quad (6)$$

.

- Sanjeev Kumar Gupta , Assistant Professor, Department of Mechanical Engineering, IET, GLA University Mathura, India, PH-9568188288, E-mail: sanjeev.gupta@gla.ac.in
- Manish Kumar Rawat , Assistant Professor, Department of Mechanical Engineering, IET, GLA University Mathura, India, PH-8938017159, E-mail: manish.rawat@gla.ac.in
- Nitin Kukreja , Assistant Professor, Department of Mechanical Engineering, IET, GLA University Mathura, India, PH-7409186806, E-mail: nitin.kukreja@gla.ac.in

3 METHODOLOGY

The computational methodology is followed to analyze this study of cooling of the electronic chip with the help of CFD. In CFD there are three main steps to solve a problem namely: Pre-Processing, Solver-Execution, and Post-Processing. In the first step i.e. Pre-Processing we deal with the geometry and modeling of the problem and after that, the whole computational domain is divided into no. of small parts to capture the flow behavior in full domain. This process is called the meshing in CFD and after it, the mesh file is generated. In the second step of Solver-Execution, the mesh file is read in the FLUENT, after this the general setup is done and also the boundary conditions are provided according to the problem. After it the setup is run until the convergence is achieved. In Post-Processing step, all the results, contours and plots are studied.

3.1 Pre - Processing

The information about problem geometry and computational domain is described in this section. This study is done for three different cases. First is when the cooling fan is installed at side location to the circuit board on which electronic chips are mounted with a single outlet, the second case when there are two outlets and the third case is when the cooling fan is installed at the top of the hot chips with two side outlets. The 2D geometry is used for this analysis. After the drawing of the computational domain, it is split into no. of small elements by giving the no. of nodes on different edges. Certain mesh laws are used to create the mesh-like Exponential1, Exponential2, Bi-geometric, uniform etc.

Case: 1 when fan is installed on the left side with single outlet

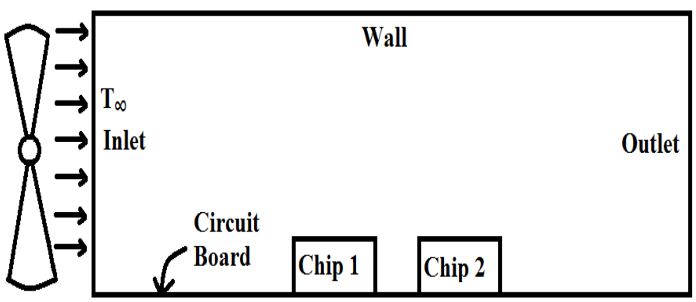

Fig – 1 *Computational domain for case 1*

Case: 2 when fan is installed on the left side with two outlets

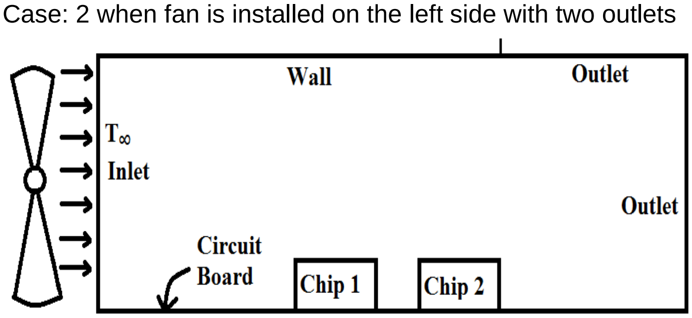

Fig – 2 *Computational domains for case 2*

Case: 3 when fan is installed on the top of the chip with two side outlets

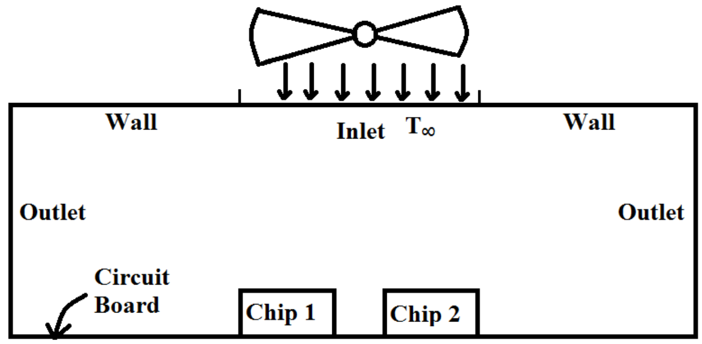

Fig – 3 *Computational Domain for case3*

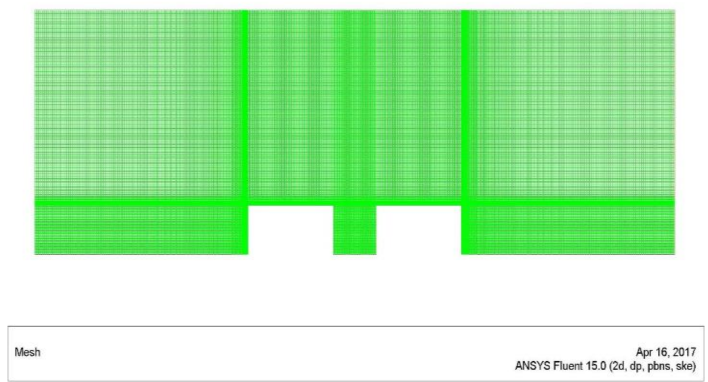

Fig – 4 *Mesh grid of the problem*

The mesh information is as follows:

Table – 1 *Mesh Information*

Quadrilateral Cells	122588, Zone 12
Interior Faces	244195, Zone 13
Velocity-inlet Faces	218, Zone 14
Pressure-outlet Faces	218, Zone 15
Wall Faces	237, Zone 16
Wall Faces	237, Zone 17
Wall Faces	358, Zone 18
Wall Faces	267, Zone 19
Nodes	123570
Min. Orthogonal Quality	1.00000e+00
Maximum Aspect Ratio	1.70209e+02

3.2 Solver Execution

Here all the boundary conditions are provided for the domain of interest where the flow is desired to be captured for the analysis. At the inlet, the ambient air is used to cool the chips. The information regarding the boundary conditions is as follows:

Table – 2 *Boundary Conditions for the domain*

Zone	Boundary Condition
Inlet	Velocity-inlet
Outlet	Pressure-outlet
Wall	No-slip condition with adiabatic wall
Chip 1	Heat Flux
Chip 2	Heat Flux

3.3 Solution Scheme

An implicit algebraic multigrid method of the solution along with second-order upwind scheme is used in the discretization to converge the results with higher accuracy. Pressure-velocity correlation is used to establish velocity-pressure coupling using Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm. Continuous residual monitoring is done to keep a check on all parameters for proper convergence rate.

4 RESULTS

The heat transfer coefficient plot for the chips is calculated with the help of software package. Temperature contours are drawn from it to see the cooling effect near the chips. The calculated numeric value of heat transfer coefficient by taking the average of it on both the chips. These are shown as follows:

Table – 3 *Results obtained for both the chips at 9 m/sec air velocity*

Parameter	Component					
	Case: 1		Case: 2		Case: 3	
	Chip 1	Chip 2	Chip 1	Chip 2	Chip 1	Chip 2
Heat Transfer Coefficient, h (W/m2-K)	104.14 46	121.45 44	103.18 89	119.66 52	233.22 12	224.35 49
Minimum Temperature (K)	3173.0 149	14448. 006	3172.9 958	1447.9 756	873.00 29	873.00 18
Maximum Temperature (K)	5185.5 254	6048.0 298	5185.5 159	4898.0 161	5185.5 254	4898.0 158

As we can see that through the results for all three cases for the air velocity at 9 m/sec. the heat transfer coefficient is greater than that for case 1 and case 2. The heat transfer coefficient in the third case gets approximate 2.24 times of the value in the first case.

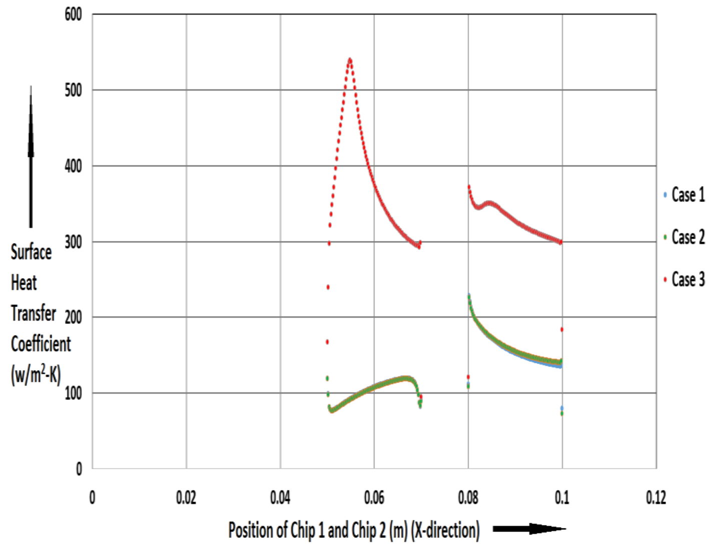

Fig.5. *Comparison of heat transfer rate through Chip 1 and Chip 2 for all three cases at 9 m/sec air velocity*

It is clear from the figure that surface heat transfer is maixumum is case 3 with chip 1 when fan is place at the top and sides are taken as outlet.

4.1 Velocity Contour

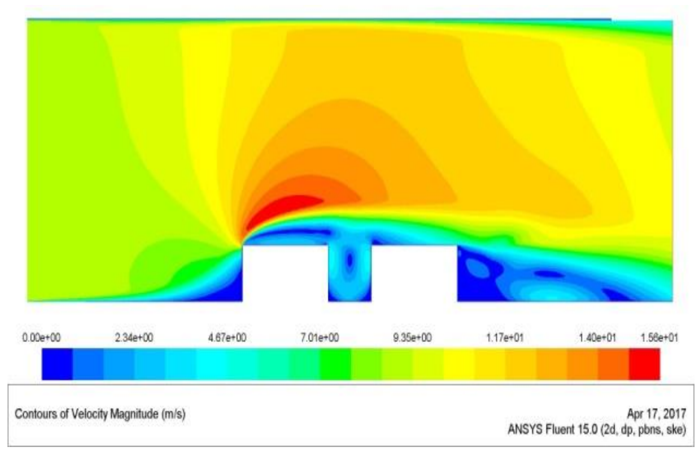

Fig.6. *Velocity contour for case 1*

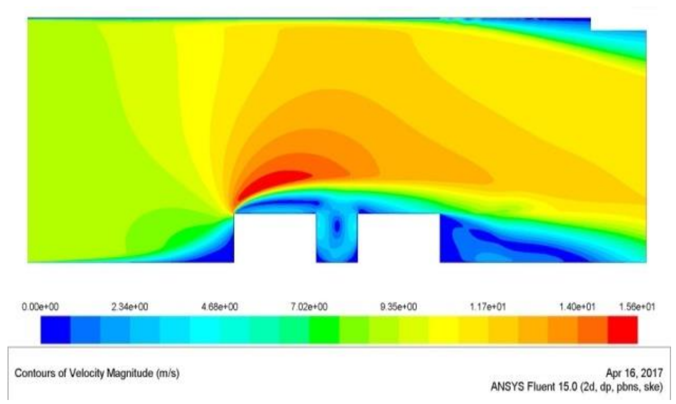

Fig.7. *Velocity contour for case 2*

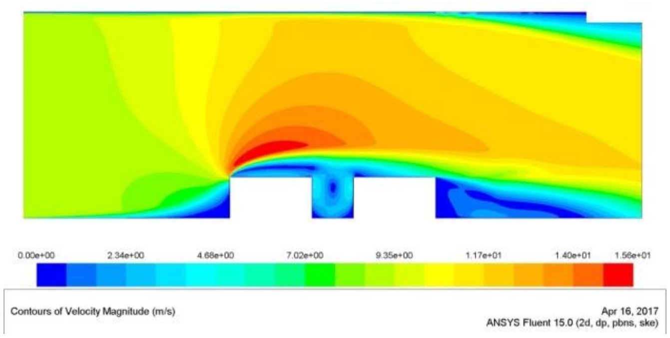

Fig.8*. Velocity contour for case 3*

4.2 Temperature Contour

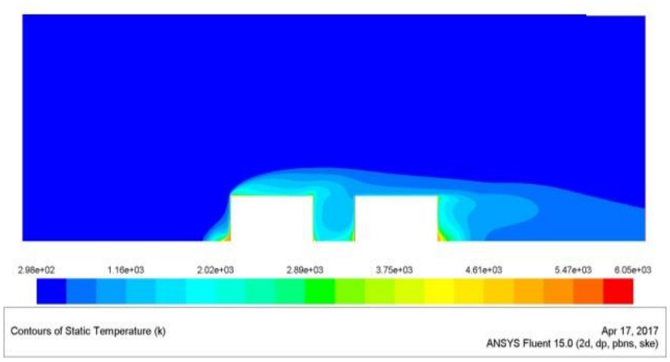

Fig.9*. Temperature contour for case 1*

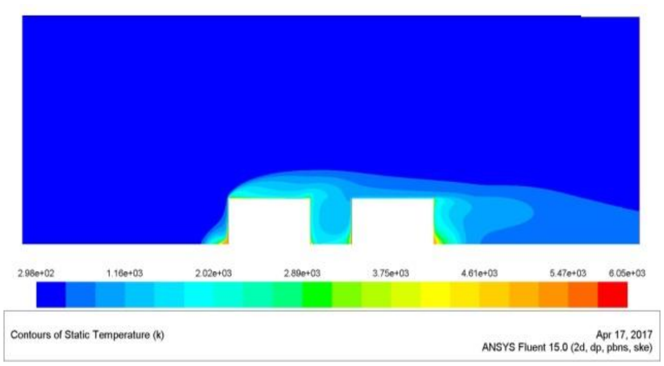

Fig.10*. Temperature contour for case 2*

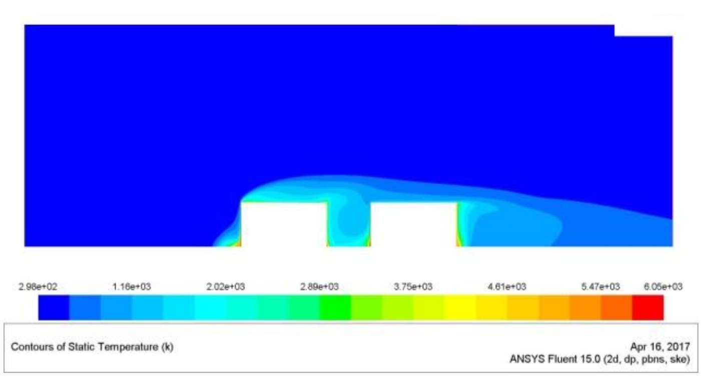

Fig.11. *Temperature contour for case 3*

5 CONCLUSION

The following conclusions can be made from this analysis:

The cooling rate is increased drastically by changing the location of the cooling fan i.e. at the top of the chips. There is an increment of about 124 % in the heat transfer coefficient for chip 1 and about 85% in the heat transfer coefficient for chip 2.

REFERENCES

[1] S. K. Patra, “CFD Analysis of Electronics Chip Cooling” A thesis of M. Tech., Department of Mechanical Engineering, National Institute of Technology, Rourkela, 2007.

[2] Kevin R. Anderson, Matthew Devost, WatitPakdee, and NivedithaKrishnamoorthy, “STAR CCM+ CFD Simulations of Enhanced Heat Transfer in High-Power Density Electronics Using Forced Air Heat Exchanger and Pumped Fluid Loop Cold Plate Fabricated from High Thermal Conductivity Materials” Journal of Electronics Cooling and Thermal Control,Vol. 3, pp-144-154, 2013

[3] D. Gupta, V. Venkataraman, R. Nimje, “ CFD& Thermal Analysis of Heat Sinkand its Application in CPU”, International Journal of Emerging Technology and Advanced Engineering, Vol. 4, No. 8, pp. 198-202, 2014.

[4] M.A.I. Rashid, M.F Ismail, and M. Mahbub, “CFD Analysis in a Liquid-Cooled Carbon Nanotube Based Micro-channel Heatsink for Electronic Cooling” IACSIT International Journal of Engineering and Technology, Volume 3, No. 5, Vol. 3, No. 5, pp. 553-559, 2011

[5] R. Singh, A. Akbarzadeh, M. Mochizuki, “Thermal Potential of Flat Evaporator Miniature Loop Heat Pipes for Notebook Cooling” IEEE Transactions on Components and Packaging Technologies, Vol. 33, No. 1, pp. 32 – 45, 2010.

[6] D. Gupta, M. Nausheen, A.D. Dhale, “CFD Analysis & Simulation of Pin Fin for Optimum Cooling of MotherBoard” International Journal of Engineering Development and Research, Vol. 2, No. 2, pp. 1381 – 1385, 2014.

NOMENCLATURE

ρ:	Density of fluid flowing (kg/m^3)
u:	Velocity of flow in x-direction (m/sec.)
v:	Velocity of flow in y-direction (m/sec.)
p:	Pressure in the direction of flow (N/m^2)
X:	Body force in x-direction (N)
Y:	Body force in y-direction (N)
T:	Temperature of Fluid (K)
μ_t:	Eddy Viscosity
κ:	Turbulent Kinetic Energy
ε:	Turbulent Dissipation Rate
G:	Turbulent Generation Rate
σ_κ:	Constant
σ_ε:	Constant
$C_{1\varepsilon}$:	Constant
$C_{2\varepsilon}$:	Constant
h:	Heat Transfer Coefficient (W/m^2-K)
T_∞:	Ambient air temperature (K)