IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 3 Issue 7, July 2016
ISSN (Online) 2348 – 7968 | Impact Factor (2015) - 4.332
www.ijiset.com

Numerical Investigation on Performance Heat Sink for Heat Source by Using CFD

Veeresha , Dr. M.C.Navindgi

M.Tech student (TPE) PDACE Kalaburgi,and Project guide
Associate Professor , Dept of Mechanical Engg,PDACE Kalaburgi
Email:veereshagudur@gmail.com, mcnavinfgi@gmail.com

Abstract

Heat indulgence techniques are the prime concern to remove the waste heat produced by Electronic Devices, to keep them within permitted operating temperature limits. Heat indulgence techniques include heat sinks, fans for air cooling, and other forms of cooling such as liquid cooling. Heat produced by electronic devices and circuitry must be self-indulgent to improve reliability and prevent premature failure. Integrated circuits such as CPUs, chipset, graphic cards, and hard disk drives are susceptible to temporary malfunction or permanent failure if overheated. As a result, efficient cooling of electronic devices remains a challenge in thermal engineering. Heat sinks are commonly used for cooling of electronic devicesHeat sinks, an array of heat fins, remove the heat from the surfaces of the chips by enhancing the heat Transfer rate through heat conduction process. Heat can also be removed from the chip surfaces through forced convection heat transfer. In this present work the CFD simulation of heat sink is carried out for two different velocity magnitudes. The heat sink is modelled as a combination of Copper and Aluminium materials. The base of the heat sink is taken as Copper and the fins are modelled as Aluminium. A conjugate heat transfer analysis is carried out for this fluid – structure case. Two cases are simulated with a velocity of flow as 4 m/s and 10 m/s. The base of the heat sink is given with heat flux boundary. The results of both the scenarios are showing good coherence with the physical phenomenon. As velocity increases the temperature of the heat sink decreases, so the heat generated in the heat sink will become less. The rate of heat transfer will be more.

Keywords

Computational Fluid Dynamics (CFD), CPU, Heat sink, Heat flux, Aluminium.

1. INTRODUCTION

1.1 Introduction to CFD:

Computational Fluid Dynamics (CFD) provides numerical approximation to the equations that govern fluid motion. CFD is the art of replacing PDE systems by a set of algebraic equations which can be solved using digital computers.Application of the CFD to analyze a fluid problem requires the following steps.

- First, the mathematical equations describing the fluid flow are written.
- These are usually a set of partial differential equations.

- These equations are then discretized to produce a numerical analogue of the equations.
- The domain is then divided into small grids or elements.
- Finally, the initial conditions and the boundary conditions of the specific problem are used to solve these equations.

In addition, certain control parameters are used to control the convergence, stability, and accuracy of the method.

All CFD codes contain three main elements:

1) A pre-processor, which is used to input the problem geometry, generate the grid, define the flow parameter and the boundary conditions to the code. It consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver.
2) A flow solver, which is used to solve the governing equations of the flow subject to the conditions provided.
   a) There are four different methods used as a flow solver:
   b) Finite difference method;
   c) Finite element method,
   d) Finite volume method, and
   e) Spectral method.

3) A post-processor, which is used to massage the data and show the results in graphical and easy to read format.After geometry has been prepared and checked, the boundary conditions have been applied, and model has been solved, it is the time to investigate the results of the analysis. This activity is known as post-processing. Post processor facilitates the analyst to check the validity of the solution, examines the values of the primary quantities, derives and examines additional quantities.

CFD has shown itself to be a powerful tool in the nuclear, aeronautical and electronic industries for over two decades; its reputation has been built on extensive work specific to those fields. Itprovides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modeling (partial differential equations), numerical methods (discretization and solution techniques) and software tools (solvers, pre- and post processing utilities).It enables scientists and engineers to perform ‘numerical experiments’ (i.e. computer simulations) in a ‘virtual flow laboratory.CFD is a highly interdisciplinary research area which lies at the interface of physics, applied mathematics, and computer science.

There exists significant work carried out in the thermal analysis of heat sink design.C.L Chapman *et al* [1] are analyzed the detailed comparison of thermal performances of different fin geometries by experimentally and theoretically. In this they investigated cross cut pin fin, straight or parallel plate fins and they compare with elliptical pin fins in their work. And they conclude Extruded straight design performed significantly better than either of the other two designs over the flow range examined. DenpangeSoodphadkee*et al* [2] analyzed the performances of Round, Elliptical and Plate fins Staggered and in-line configuration. In this they compare different fin geometries and they were simplifying by assuming periodically developed 2D flow and isothermalheat transfer surface. In general it is found that rounded geometries outperform similarly sharp edged fin shapes. By comparing all finally they concluded staggered shows better performance than in-line. AmbeprasadKushwal*et al* [3]are study the comparison of the different shaped finned heat sink by numerical investigation and obtaining the thermal performance as thermal resistance, pressure drop, heat transfer rate, heat transfer coefficient, surface nusselt number. By comparing concluded trapezoidal performs better. EmreÖztürk [4]. He investigated the Heat sink effectiveness, effect of turbulence models, effect of radiation heat transfer and different heat sink geometries were numerically analyzed by commercially available computational fluid dynamics software Fluent for forced cooling. The numerical results are compared with the experimental data. Conjugate heat transfer is simulated for all the electronic cards and packages by solving Navier-Stokes equations. Qu and Mudawar [5] have performed experimental and numerical investigations of pressure dropand heat transfer characteristics of single-phase laminar flow in 231μm by 713 μm channels.

2. Numerical methods

The equations governing fluid motion are based on the fundamental physical principles. They are the mathematical statements upon which all of fluid dynamics is based. These equations speak physics.

1. Mass is conserved; change of mass=0
2. F= ma (Newton's second law)
3. Energy is conserved.

In fluid flow these are usually express as rate equations.

2.1 Governing equations:

The equations governing the fluid motion are the three fundamental principles of mass,momentum, and energy conservation.

Continuity:$\frac{\partial\rho}{\partial t}+\nabla.(\rho V)=0$ ……………………………………………………..1

Momentum:$\rho\frac{DV}{Dt}=\nabla.\tau_{ij}-\nabla p+\rho F$…………..…..2

Energy:$\rho\frac{De}{Dt}+p(\nabla.V)=\frac{\partial Q}{\partial t}-\nabla.q+\emptyset$…………………………………...3

where ρ is the fluid density, *V*is the fluid velocity vector, $\tau_{ij}$is the viscous stress tensor,p is pressure, $F$is the body forces, e is the internal energy, $Q$is the heat source term, $t$is time, $\emptyset$is the dissipation term, and $\nabla.q$is the heat loss by conduction. Fourier's law forheat transfer by conduction can be used to describe $q$as:

q=-$k\nabla$T……………………………………………..………...4

Where$k$ is the coefficient of thermal conductivity, and T is the temperature. Depending onthe nature of physics governing the fluid motion one or more terms might be negligible.

For example, if the fluid is incompressible and the coefficient of viscosity of the fluid, μ,as well as, coefficient of thermal conductivity are constant; the continuity, momentum,and energy equations reduce to the following equations.

$$\nabla.V=0$$

$$\rho\frac{DV}{Dt}=\mu\nabla^2 V - \nabla p + \rho F$$

$$\rho\frac{De}{Dt} = \frac{\partial Q}{\partial t} + k\nabla^2 T + F$$

Presence of each term and their combinations determines the appropriate solutionalgorithm and the numerical procedure.

2.2 Navier stokes equations-conservation form:

The viscous flow is one where the transport phenomenon of friction, thermal conduction and mass diffusion are included. These transport phenomenon are dissipative they always increases the entropy of the flow. The mass diffusion occurs when there are concentration gradients of different chemical species in flow.

Continuity equation:

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho V) = 0$$

Momentum equation:

X- Component:

$$\frac{\partial(\rho u)}{\partial t}+\nabla.(\rho uV) = -\frac{\partial p}{\partial x} + \frac{\partial\tau_{xx}}{\partial x} + \frac{\partial\tau_{yx}}{\partial y} + \frac{\partial\tau_{zx}}{\partial z} + \rho f_x$$

Y- Component:

$$\frac{\partial(\rho v)}{\partial t}+\nabla.(\rho vV) = -\frac{\partial p}{\partial y} + \frac{\partial\tau_{xy}}{\partial x} + \frac{\partial\tau_{yy}}{\partial y} + \frac{\partial\tau_{zy}}{\partial z} + \rho f_y$$

Z- Component:

$$\frac{\partial(\rho w)}{\partial t}+\nabla.(\rho wV) = -\frac{\partial p}{\partial z} + \frac{\partial\tau_{xz}}{\partial x} + \frac{\partial\tau_{yz}}{\partial y} + \frac{\partial\tau_{zz}}{\partial z} + \rho f_z$$

Energy equation:

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla.\left[\rho\left(e+\frac{V^2}{2}\right)V\right] = \rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}$$
$$-\frac{\partial(wp)}{\partial z}+\frac{\partial(u\tau_{xx})}{\partial x}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}$$
$$+\frac{\partial(v\tau_{zy})}{\partial z}+\frac{\partial(w\tau_{zy})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}+\rho f.V$$

2.3 Boundary conditions:

All CFD problems are defined in terms of initial and boundary conditions. It is important that the user specifies these correctly and understand their role in the numerical algorithm. In transient problems the initial values of all flow variables need to be specified at all solution points in the flow domain. The boundary conditions used for the simulation of flow around the NACA 0012 air foil are follows.

a) Wall boundary condition
b) Pressure far field

2.3.1 Wall boundary condition:

- Used to bound fluid and solid regions.
- In viscous flows, no-slip condition enforced at walls.
  – Tangential fluid velocity equal to wall velocity.
  – Normal velocity component is set to be zero.
- Alternatively one can specify the shear stress.

- Thermal boundary condition.
  – Several types available.
  – Wall material and thickness can be defined for 1-D or in-plane thinPlate heat transfer calculations.
- Wall roughness can be defined for turbulent flows.
  – Wall shear stress and heat transfer based on local flow field.

2.3.2 Pressure Far-Field Boundary Conditions:

Pressure far-field conditions are used to analyze the model of free-stream condition at infinity, with free-stream Mach number. The pressure far-field boundary condition is often called a characteristic boundary condition, since it uses characteristic information to determine the flow variables at the boundaries.

- This boundary condition is applicable only when the density is calculated using the ideal-gas law.
- It cannot be applied to flows that employ constant density, the real gas model, and the wet steam model, which are available in the density-based solver.

2.3.3 Inlet Boundary Conditions:

The assessment of all the flow variables required to be specified at inlet boundaries. The direction of the flow is imagined to be mainly from left to right in the drawings. At an inlet, fluid comes in the region and hence, its velocity or pressure or the mass flow rate may be identified and still the fluid may have some characteristics, like the turbulence indicates which is required to be recognized.

3. CFD analysis of heat sink

3.1 Introduction to Heat Sink:

Heat sinks, used in electronic devices, usually consist of arrays of fins arranged in an in-line manner as shown in Fig 3.1. The fins are attached to a common base and the geometry of the array is determined by the fin dimensions, number of fin arrangement.

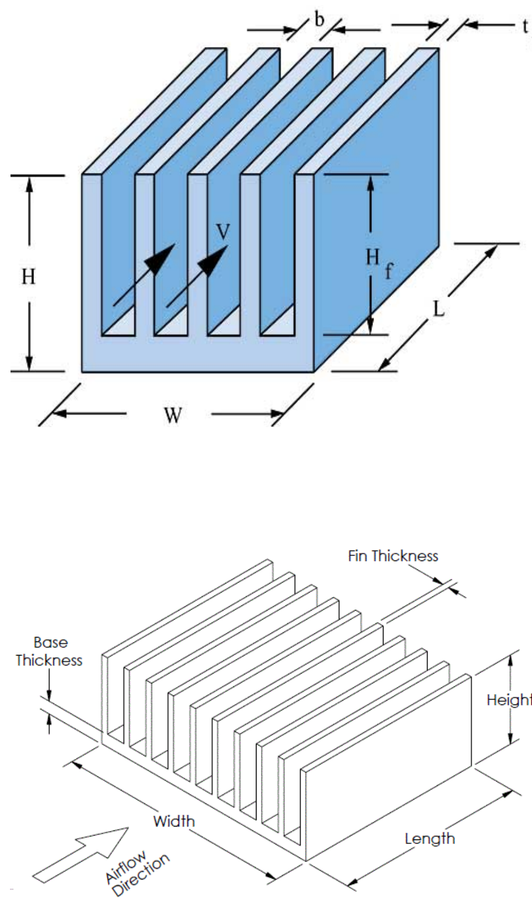


Fig3.1: Heat Sink

The geometry of an in-line continuous rectangular fin heat sink is shown in Fig 1.The dimensions of the base plate are L × W × H, where L is the length in the stream wise direction, W is the width, and H is the height. The approach velocity of the air is U. The direction of

IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 3 Issue 7, July 2016
ISSN (Online) 2348 – 7968 | Impact Factor (2015) - 4.332
www.ijiset.com

the flow is parallel to the x-axis. The base plate is kept at constant heat flux and the top surface of the fins is adiabatic. The heat source is idealized as a constant heat flux boundary condition at the bottom surface of the base plate. It is assumed that the heat sink is fully shrouded and the heat source is situated at the centre of the base plate. It is assumed that the fluid temperature is averaged over the height of the heat sink, so the fluid temperature T is the bulk meanfluid temperature. Fully developed heat and fluid flow are assumed in the analysis, and the thermo physical properties are taken to be temperature independent.

3.2

3.3 Geometry:

The design of heat Sink with rectangular fins is done in GAMBIT in IGES format. A flat platformof 150 X 150 X 2.5 mm [9] used. The extended domain considered is having 50mm*50mm*50mm dimensions in the all the directions. The geometry considered for the analysis is shown in the following figures.

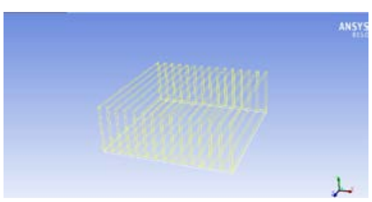


Fig 3.2.2: Geometry of Fin

3.4 Mesh:

In order to analyze fluid flows, flow domains are split into smaller domains. The sub domains are often called elements or cells, and the collection of all elements or cells is called a mesh or grid.

I have used GAMBIT as a pre-processor for generating mesh. The details of the mesh are given in below table.

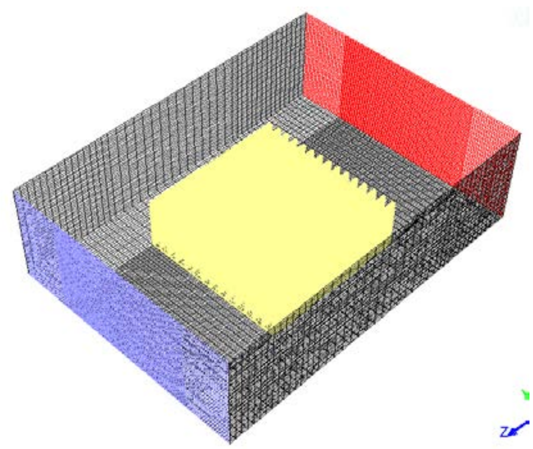
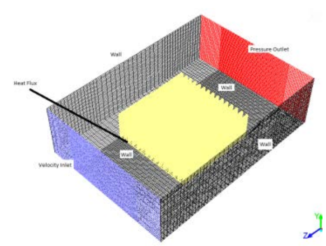


Fig3.3.2: Mesh of Fins

Fig3.4: Boundary Conditions Used for the Simulation



Heat Flux: No Slip Wall with Heat Flux=10000 W/m2

Velocity Inlet: Velocity inflow with velocity = 4 & 10 m/s

Pressure Outlet: Pressure = 0 pa

Wall: No slip isothermal condition

Inlet air temperature : 27degre (assumed)

3.5 Solver parameters:

- Pressure based coupled solver is used for the solution
- K- € turbulence model is used
- Second order up wind discretization method is used for density, momentum and energy in the solution.
- The relaxation factor used for momentum and pressure is 0.5.

3.6 Initialization:

As the domain is combination of both solid and fluid, hybrid initialization is used for better initialization.

3.7Case Studied:

- **Case1:**The bottom of the fin is given with a heat flux of 10000W/m2 & velocity of air is 4 m/s.
- **Case2:**The bottom of the fin is given with a heat flux of 10000W/m2 & velocity of air is 10 m/s.

4. Results and discussion

The cases discussed in chapter 3 are simulated using fluent software and the results of all the cases are discused in this chapter. The contours for static pressure, Temperature and velocity vector are plotted and discussed.

4.1 Different Plane

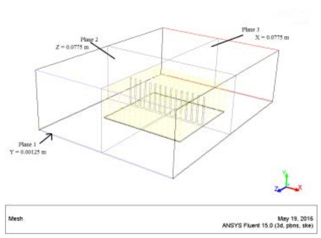


Fig 4.1.1: Planes at different Locations

4.2 Plane 1

4.2.1 Static Pressure

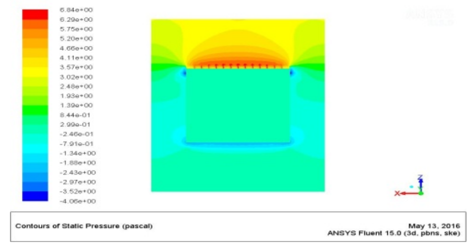
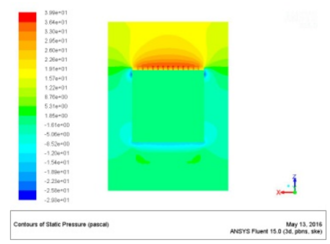


Fig 4.2.1: Static Pressure contour of case 1



.

Fig 4.2.2: Static Pressure contour of case 2

4.2.2 Static Temperature

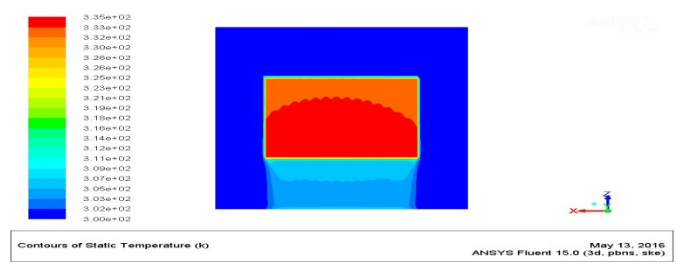


Fig 4.2.3: Static Temperature contour of case 1.

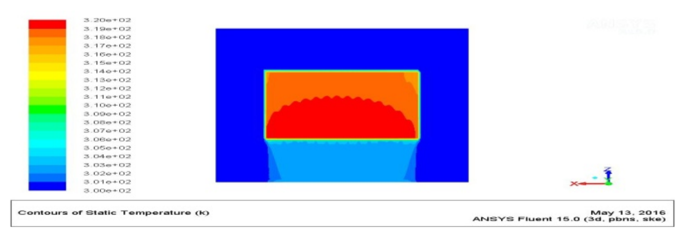


Fig 4.2.4: Static TemperatureContour forcase 2

4.2.3 Velocity Magnitude

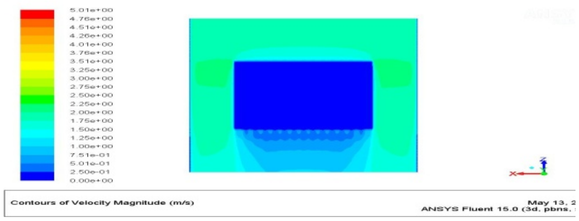


Fig 4.2.5: Velocity Magnitude contour of case 1.

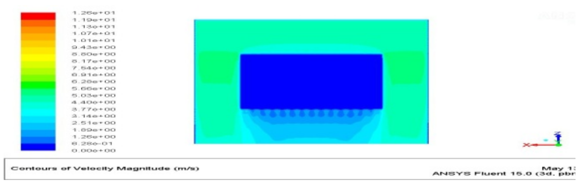


Fig 4.2.6: Velocity MagnitudeContour forcase 2.

4.5 Comparision of Results:

Table 1 gives the comparion between Case1 and Case2. The Case1 (4 m/s) is showing higher temperature compared to Case2 (10 m/s) because of the lower velocity. The pressure is high in Case2 because of the higher velocity compared to Case1.

CASE	Velcoity m/s	Static Pressure in Pa	Temperature In Kelvin	Velocity in m/s
1	4	6.84	335	5.01
2	10	39.9	320	12.6

6.Conclusion

In this work the numerical simulations carried out for a heat sink with two different velocity magnitudes (4 m/s & 10 m/s). The results obtained for both the simulations are in good coherence with physical phenomena. The heat sink is taken as solid with two different materials. The base is assigned with copper material properties and fins are assigned with aluminium material properties. The fluid is assigned with air properties. The velocity 4 m/s case gives lower pressure and higher temperature. But the velocity 10 m/s case gives more pressure and lower temperature. It is observed that increasing the velocity of the flow increases the pressure and decreases the temperature.

Further the numerical simulation can be carried out by taking different material properties of solid. The simulation can also be carried out by increasing the gap between the fins.

7.Reference

[1] C. L. Chapman, S. Lee, and B. L. Schmidt, "Thermal Performance of an Elliptical Pin Fin Heat sink," Proceedings of the Tenth IEEE Semiconductor Thermal Measurement and Management Symposium (Semi-Therm), San José, California, February 1-3, pp. 24-31, 1994.

[2] DenpongSoodpadhkee, MasudBehnia and Watabe, „A comparison of heat sinks in Laminar forced convection: part I- Round, Elliptical, and Plate Fins Staggered and Inline configuration‟. IJMP – Volume 24 NO-1, in 2001.

[3] AmbeprasadKushwaha and Prof. RavindraKirar, „Comparative study of Rectangular, Trepezoidal and Parabolic shaped Finned Heat sink‟. IOSR – 2278-1684 Volume 5 Issue 6 – Apr 2013.

[4] Ozturk‟ E (2004). CFD Analysis of heat sinks for CPU cooling with FLUENT. M.S thesis, Middle East

Technical University, Ankara, Turkey.

[5] Qu, W. and Mudawar, I. 2002. Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink.International Journal of Heat and Mass Transfer. 45, 2549 – 2565.

[6] Vinay Pal, "Modelling and thermal analysis of heat sink with scales on fins cooled by natural convection,'' International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308.