International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 11 | Nov 2019 www.irjet.net p-ISSN: 2395-0072

GEOMETRICAL OPTIMIZATION OF FINS FOR EFFECTIVE HEAT TRANSFER USING CFD ANALYSIS

G Satyanarayana[1], K. pavan kumar[2], G. Veeraiah[3], M Y Prasada Rao[4], Ch Varun[5], SS Naidu[6]

[1-6]*Department of Mechanical Engineering, MVGRCE (A), Vizianagaram, AP, India*

***

Abstract - *The role played by heat exchangers enhanced significantly in the recent evolutions of electronic and industrial components designs. This work has been undertaken to study the problem of geometric optimization for exchanger profiles of electronic devices with innovative shapes. In particular, it was analyzed for the Y-shape, keeping the dimensionless thermal conductance as reference parameter like in the thematic technical literature. The use of suitable geometrical constraints allows wide comparisons by using different kinds of geometries. It is seen that Y-shaped configurations are very effective, as they can allow more heat removal by very less space occupancy. The present work analyses the heat exchange behavior in systems characterized by Y-shaped fins and the temperature contours are plotted using ANSYS FLUENT 12.0. The geometries are obtained starting from T-shaped profiles and then to Y- shaped profiles by varying the angle between the arms. The analysis has been performed by superimposing some dimensional constraints to make it immediately comparable with the results obtained in the different configurations. In this work, the modeling of the fins has been done using Creo Parametric 2.0 Pro/ENGINEER) and the analysis was done using ANSYS FLUENT 12.0. The material of the fin is taken as aluminium and the fluid surrounding the fin is air. This result validates the new optimization criterion proposed.*

Key Words: **CFD ANALYSIS, T FINS, Y FINS, SHAPE OPTIMIZATION, HEAT EXCHANGERS**

1. INTRODUCTION

Heat transfer is of great importance to engineers because of its almost universal occurrence in many branches of science and engineering. Heat exchangers are widely used in various, transportation, industrial, or domestic applications such as thermal power plants, means of heating, transporting and air conditioning systems, electronic equipment and space vehicles. In all these applications improvement in the efficiency of the heat exchangers can lead to substantial cost, space and material savings. With the increase in heat dissipation from microelectronics devices and the reduction in overall form factors, thermal management has become an important element of electronic product design. Both the performance reliability and life expectancy of electronic equipment are inversely related to the component temperature of the equipment.

Heat flows from a hot body to a cold body. The transfer of heat energy due to temperature difference or gradient is called heat transfer. The basis of any heat transfer enhancement technique lies in the utilization of some external power in order to permit the mixing of working fluids, the rotation of heat transfer surfaces, the vibration of heat transfer surfaces or of the working fluids also the generation of electrostatic fields.

In the cooling enhancement of current electronic industry, heat sink is extensively used to provide cooling function for electronics components. Due to the circuit density and power dissipation of integrated circuit chips are increasing, the heat flux levels within these chips have increased. The accumulation of large amount of heat flux can create considerable quantities of heat stress on chips, substrate, and its package. Therefore, it is necessary for employing effective heat sink module to maintain the operating temperature of electronic components at a satisfactory level. If there is appropriate and effective heat sink design, it will critically affect the reliability and life span of chip function.

Among various cooling techniques for electronic chips and/or modules, forced convective cooling with air features advantages of convenience and low cost. The most common method for cooling electronic devices is by finned heat sinks made of aluminum. These heat sinks provide a large surface area for the dissipation of heat and effectively reduce the thermal resistance. In order to design an effective heat sink, some criterions such as a large heat transfer rate, a low pressure drop, an easier manufacturing, a simpler structure, a reasonable cost and so on should be considered. Unfortunately, heat sinks often take up much space and contribute to the weight and cost of the product.

The need for new design and more effective ways to dissipate this energy is becoming increasingly urgent. Because of compelling market requirements, design optimization of new electronic devices, is often found to be prohibitive. It is true that an undesirable phenomenon such as increasing in the pressure loss commonly takes place in the plate-fin type heat sinks in which fins are attached to the plate in order to enhance the heat transfer rate. Thus, high performance of heat sinks can be acquired through the design optimization which maximizes heat transfer and minimizes pressure drop. To achieve an optimum design of heat sink for an effective heat transfer, newer methodologies or techniques are to be identified.

1.1 EXTENDED SURFACES:

Fins, also referred to as extended surfaces, are thin strips of metal attached to a surface to enhance the rate of heat transfer dissipation from heated surfaces to air. Fins can be placed on plane surfaces, tubes, or other geometries. Fins are manufactured in different geometries-plain, wavy or interrupted—and can be attached to the inside, outside or to both sides of circular, flat or oval tubes or parting sheets depending on the practical applications. Fins are attached to the primary surface by brazing, soldering, welding, adhesive bonding or mechanical expansion, or extruded or integrally connected to tubes. They may be of uniform or variable cross-section.

Various types of fins used commonly are:

- Straight fins of uniform cross section
- Straight fins of non-uniform cross section
- Annular fins
- Cylindrical fins
- Pin fin

1.2 FIN EFFICIENCY:

Fin efficiency is the ratio of heat transfer from the actual fin to the heat transfer of an imaginary fin of the same geometry and same conditions but with an infinite conductivity (In other words, if the entire fin surface was in a temperature equal to that of the fin base).

$$\eta = \frac{Q_{fin}}{h\, A_s\, (T_b - T_\infty)}$$

This ratio will always be smaller than one be used.

1.3FIN EFFECTIVENESS:

Fin effectiveness is the ratio of heat transfer from the fin to the heat transfer if the fin wasn't existing. In other words, this quantity tells us how much extra heat is being transferred by the fin.

$$\epsilon = \frac{Q_{fin}}{hA_{fin\ base}\, (T_b - T_\infty)}$$

The desire is to have this ratio as large as possible while keeping the additional cost of adding the fins as low as possible.

Fin effectiveness can be enhanced by,

1. Choice of material of high thermal conductivity. Eg. Aluminium, Copper

2. Increasing ratio of area to the perimeter of the fins. The use of thin closely placed fins is more suitable than thick fins.

3. Low values of heat transfer coefficient (h).

2. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. CFD is part of computational mechanics, which in turn is part of simulation techniques. Simulation is used by engineers and physicists to forecast or reconstruct the behaviour of an engineering product or physical situation under assumed or measured boundary conditions (geometry, initial states, loads, etc.).

Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

- mathematical modelling (partial differential equations)
- numerical methods (discretization and solution techniques)
- software tools (solvers, pre- and post-processing utilities)

2.1 FLUID GOVERNING EQUATIONS:

The cornerstone of computational fluid dynamics is the governing equations of fluid dynamics- the continuity equation, momentum equation and energy equation. These equations speak physics. They are the mathematical statements of three fundamental physical principles upon which all of fluid dynamics is based:

(1)Conservation of mass

(2) Conservation of momentum

(3) Conservation of energy

2.2NAVIER-STOKES EQUATIONS

2.2.1 Conservation Law:

Navier-Stokes equations are the governing equations of Computational Fluid Dynamics. It is based on the conservation law of physical properties of fluid. The principle of conservational law is the change of properties, for example mass, energy, and momentum, in an object is decided by the input and output.

For example, the change of mass in the object is as follows

$$\frac{dM}{dt} = m_{in} - m_{out}$$

If $m_{in} - m_{out} = 0$, we have , $\frac{dM}{dt} = 0$

Where M is constant.

2.2.2 CONTINUITY EQUATION:

The continuity equation describes the conservation of mass. It is given by,

$$\frac{D\rho}{Dt} = \rho\frac{\partial U_i}{\partial x_i}$$

2.2.3 MOMENTUM EQUATION:

The momentum equation describes the law of conservation of momentum which states that the net force acting on a mass is equal to the change in momentum in that direction.

$$\partial\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + z\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) + \rho g_x$$

$$\partial\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + z\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) + \rho g_y$$

$$\partial\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + z\frac{\partial w}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) + \rho g_z$$

2.2.4 ENERGY EQUATION:

One of the most fundamental laws in nature, First Law of Thermadynamics, also known as, conservation of energy principle, provides a basis for studying the relations among energy and energy interactions. It states that "Energy can neither be created nor destroyed, it can only be changed from one form to another".

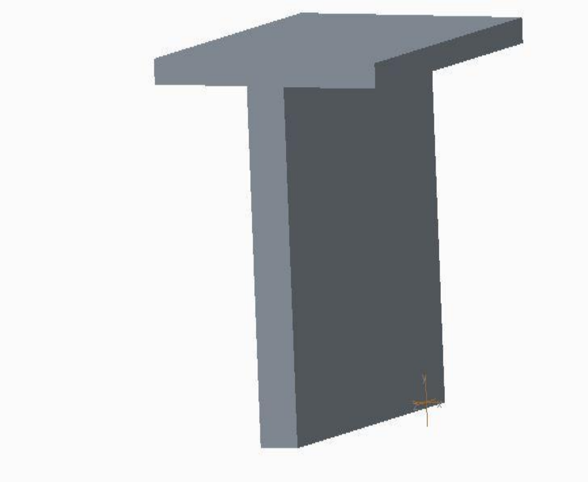
Conservation of energy principle for any system can simply be expressed as

$$\Delta E = E_{in} - E_{out}$$

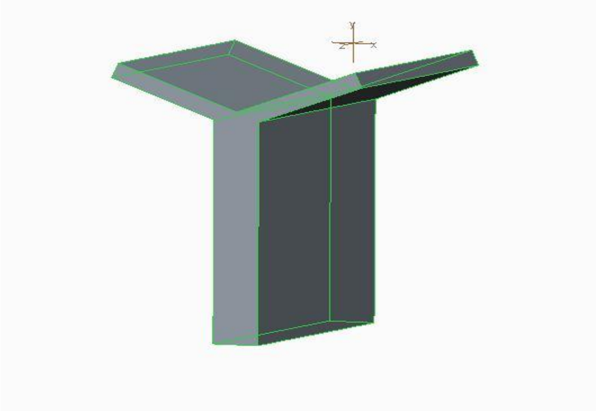
3. MODELING AND ANALYSIS

3.1 MODELING In Creo, the parametric part modeling process involves the following steps:

- Set up the units and basic datum geometry.
- Determine the type of base feature, the first solid feature, of the design.
- Create a rough two-dimensional sketch of the basic shape of the base feature of the design.
- Apply/modify constraints and dimensions to the two-dimensional sketch.
- Transform the two-dimensional sketch into a 3D feature.
- Add additional parametric features by identifying feature relations and complete the design.
- Perform analysis/simulations and refine the design as needed.
- Document the design by creating the desired 2D/3D drawings

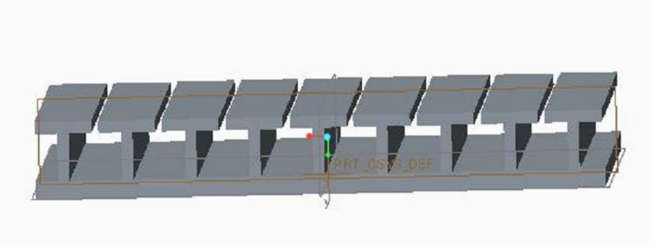


GEOMETRY OF T-SHAPED FIN

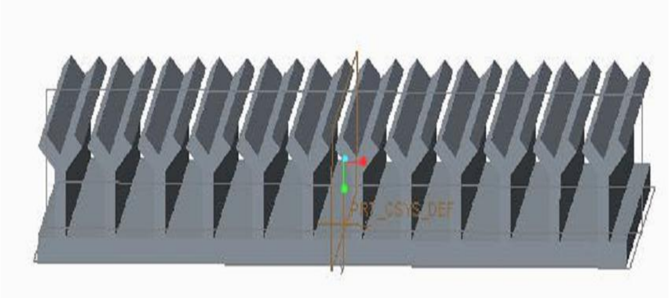


GEOMETRY OF Y-SHAPED FIN(∠135)

GEOMETRY OF Y-SHAPED FIN(∟90)



GEOMETRY OF 9 T-SHAPED FINS ON A FLAT PLATE



GEOMETRY OF 12 Y-SHAPED FINS ON A FLAT PLATE

3.2 ANALYSIS

The geometry modeling can be done within ANSYS workbench or a separate CAD system may be used. In this work, the modeling of the geometry is done using Creo parametric 2.0 and imported to ANSYS FLUENT and the meshing and solving is done.

STEPS IN SOLVING CFD PROBLEM

1. Define the modeling goals
2. Create the model geometry
3. Generate the mesh
4. Set up the solver and physical models
5. Compute and monitor the solution

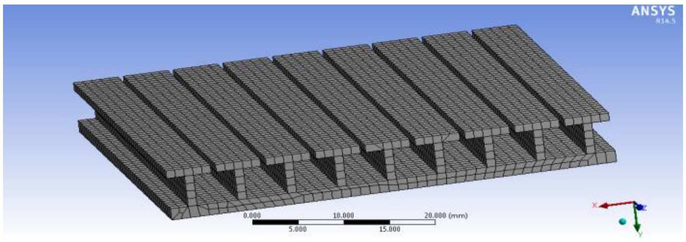
ANALYSIS OF T-SAHAPED AND Y-SHAPED FINS

The analysis of the t and y shaped fins is carried out using ANSYS FLUENT software. The meshing, solver, material selection, boundary conditions, etc are done step by step and the results are analyzed.

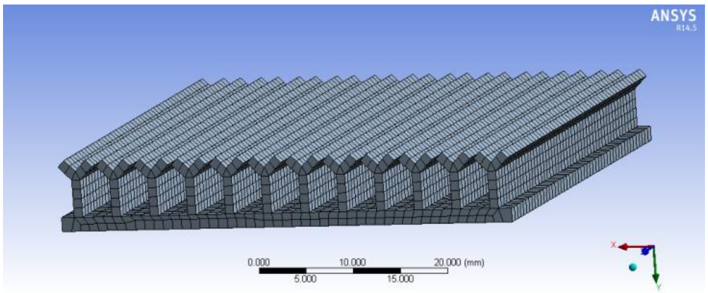
MESHING

ANSYS WORKBENCH is opened and FLUENT is selected from the toolbox. The geometry is imported from Creo and meshing is done with the following parameters as shown in the table below.

PHYSICAL PARAMETERS	VALUES
Use advance size function	On: Curvature
Relevance centre	Fine
Initial size seed	Active assembly
Smoothing	Medium
Transition	Slow
Span angle centre	Fine
INFLATION	
Use automatic inflation	Program controlled
Inflation option	Smooth transition



MESHING OF T-SHAPED FINS



MESHING OF Y-SHAPED FINS

SOLVER, MATERIAL SELECTION AND BOUNDARY CONDITIONS

PHYSICAL PARAMETERS	VALUES
Models	Energy: on K – Epsilon Viscous – Realizable K – Epsilon Standard wall fin
Materials	Fluid – Air Solid - Aluminium
BOUNDARY CONDITIONS	
Heat input	373.15 K
Ambient	293.15 K

SOLUTION METHODS

PHYSICAL PARAMETERS	VALUES
Pressure Velocity Coupling Scheme	Simple

SPACIAL DISCRETIZATION	
Gradient	Least squares cell based
Pressure	Presto
Momentum	2nd Order upwind
Turbulence KE	1stOrder upwind
Turbulence KE dissipation rate	1stOrder upwind

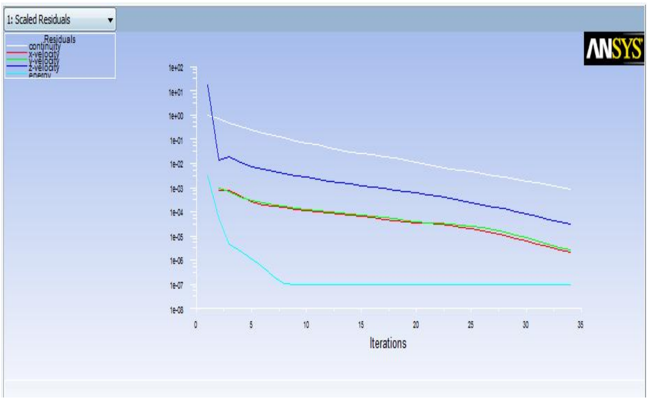
SOLUTION CONTROLS

PHYSICAL PARAMETERS	***VALUES***
Pressure	0.3
Momentum	0.7
Turbulent kinetic energy	0.8
Turbulent viscosity	1
Energy	1

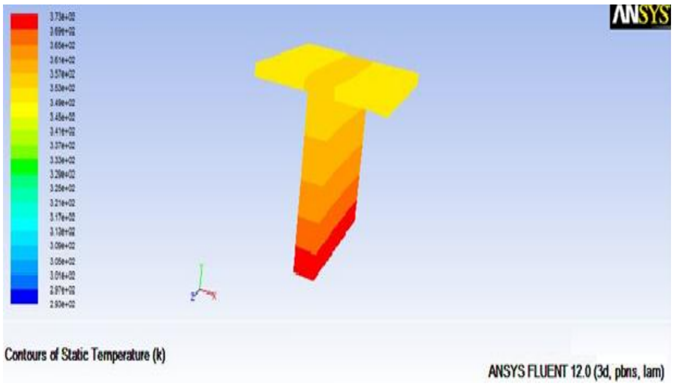
INITIALIZATION:

After defining all the parameters, the solution is initialized. Then the number of iterations to be performed is set up and the calculation starts.

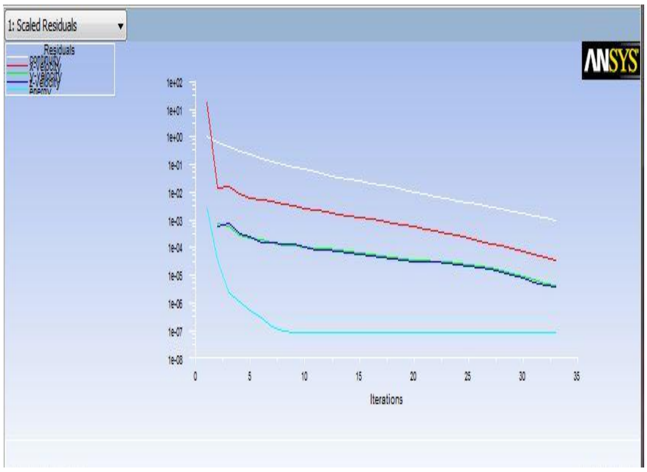
The iterations continue till the convergence is reached.



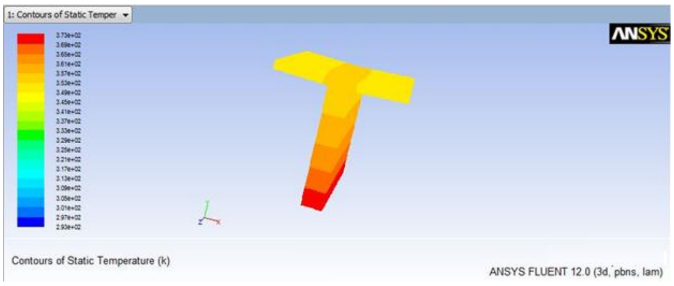
CONVERGENCE HISTORY FOR T (180) SHAPED FIN



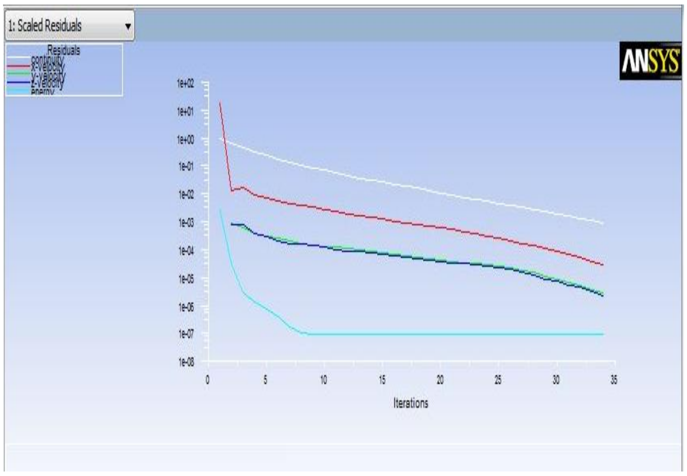
CONTOURS OF STATIC TEMPERATURE



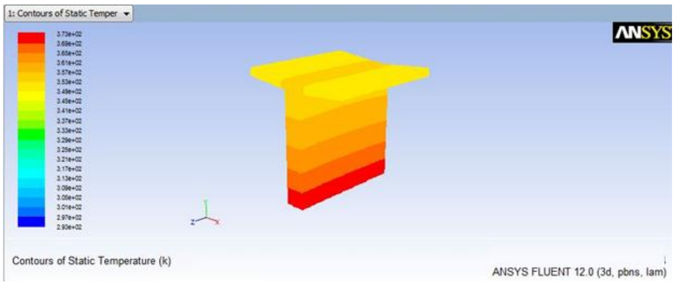
CONVERGENCE HISTORY FOR Y (175) SHAPED FIN



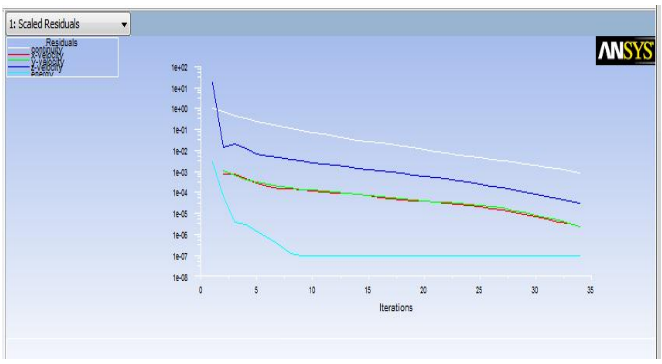
CONTOURS OF STATIC TEMPERATURE



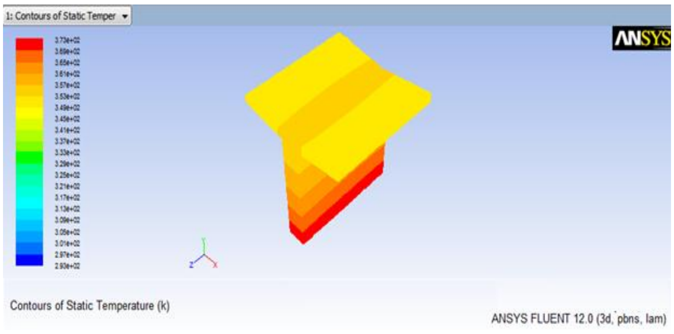
CONVERGENCE HISTORY FOR Y (170) SHAPED FIN



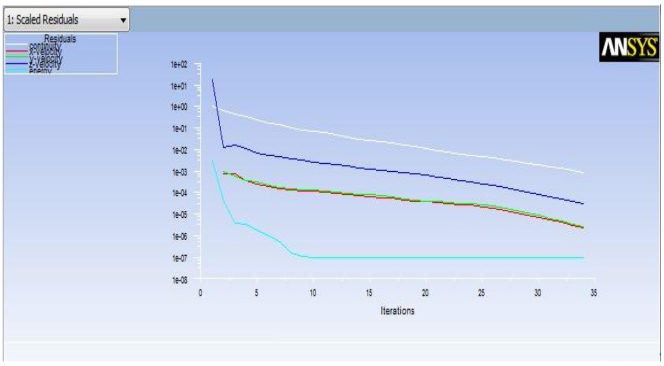
CONTOURS OF STATIC TEMPERATURE



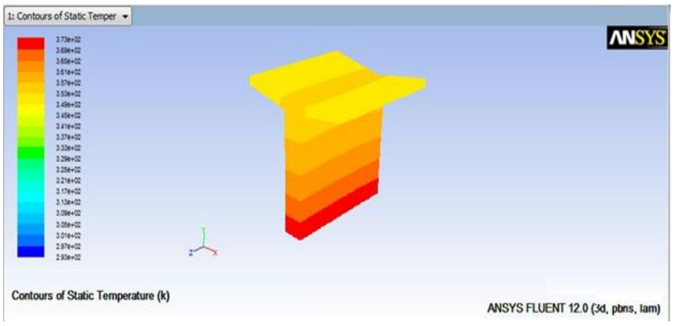
CONVERGENCE HISTORY FOR Y (165) SHAPED FIN



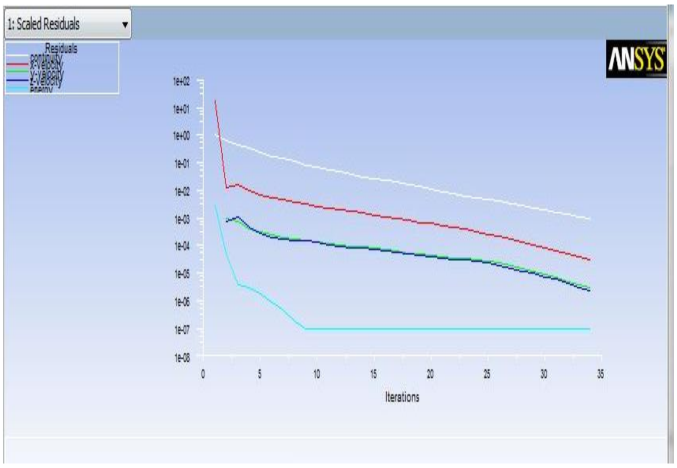
CONTOURS OF STATIC TEMPERATURE



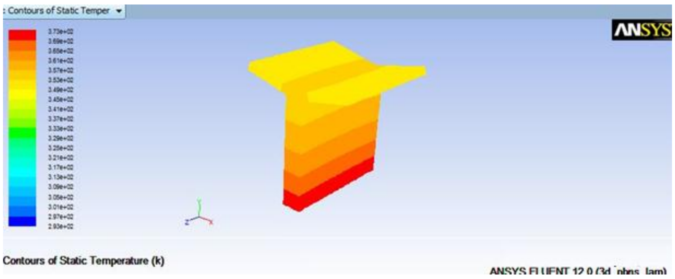
CONVERGENCE HISTORY FOR Y (160) SHAPED FIN



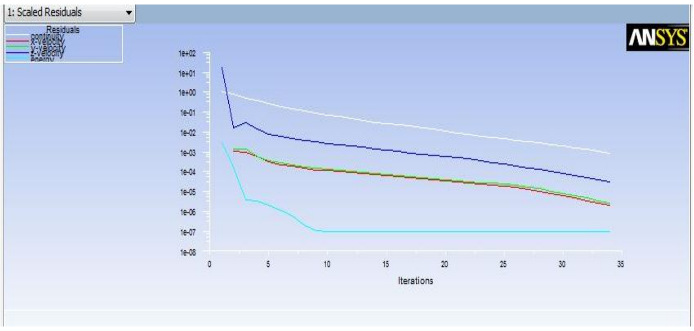
CONTOURS OF STATIC TEMPERATURE

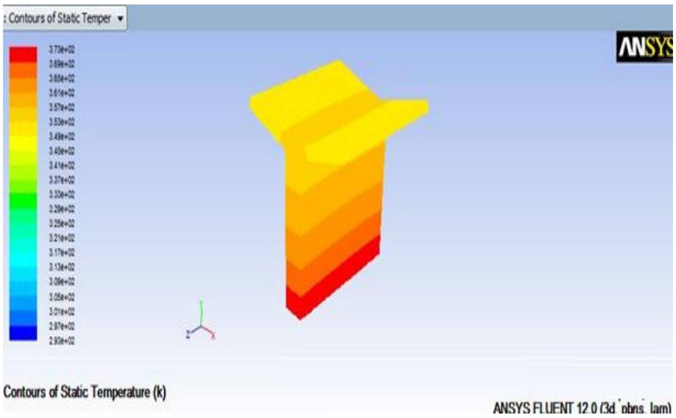


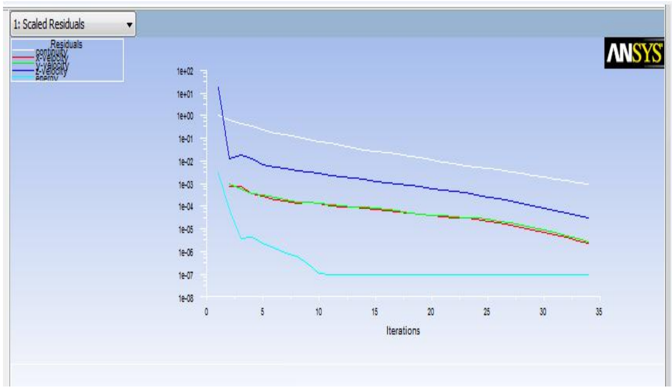
CONVERGENCE HISTORY FOR Y (155) SHAPED FIN



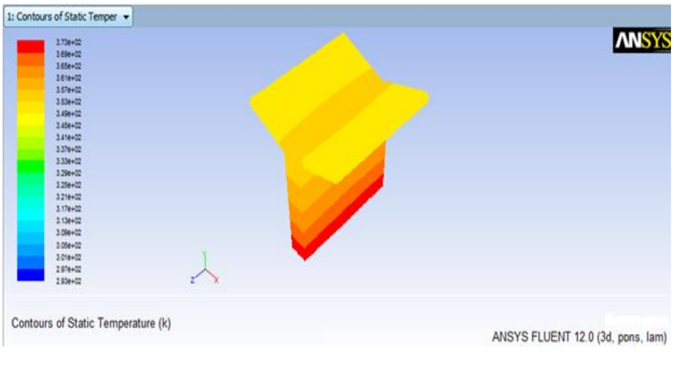
CONTOURS OF STATIC TEMPERATURE

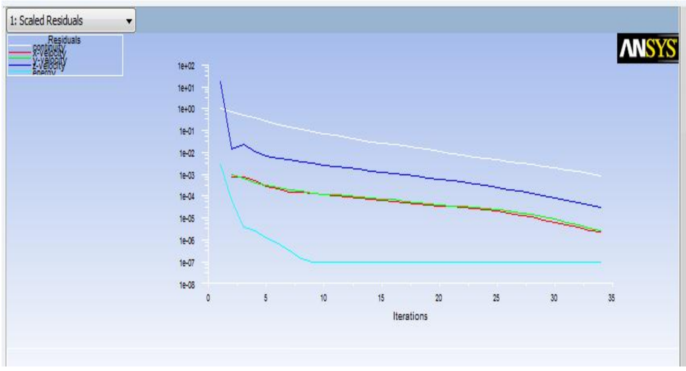




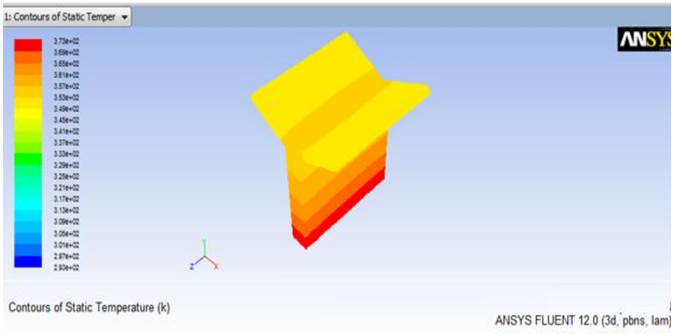


CONVERGENCE HISTORY FOR Y (140) SHAPED FIN

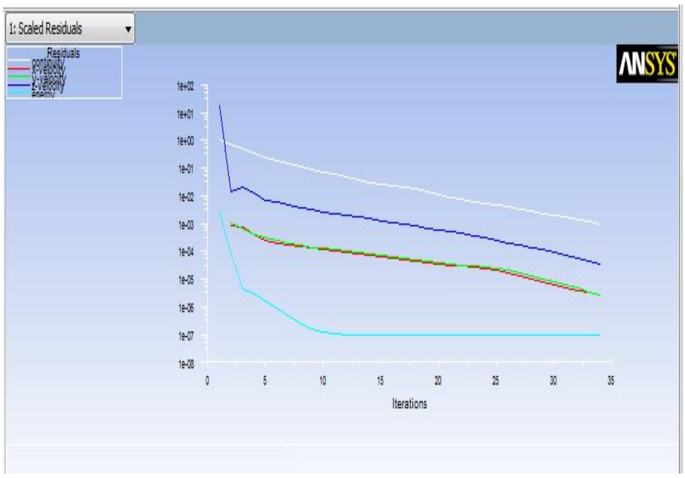




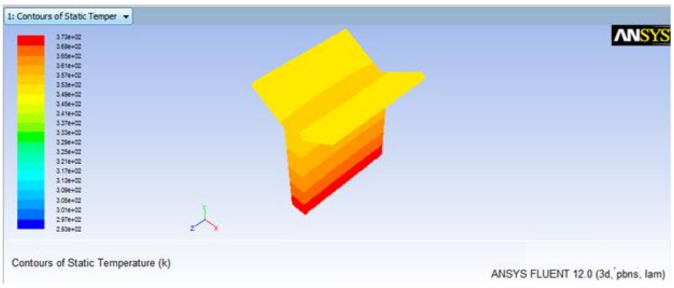
CONVERGENCE HISTORY FOR Y (130) SHAPED FIN



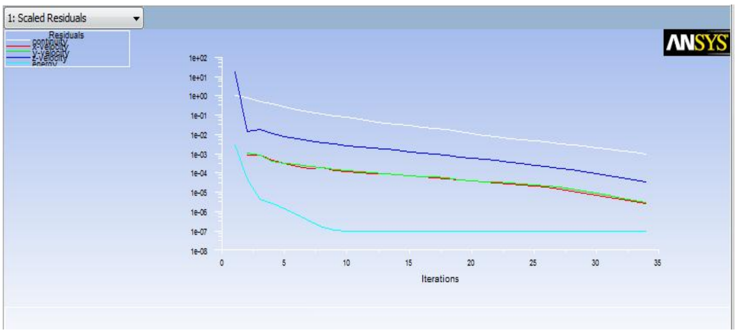
CONTOURS OF STATIC TEMPERATURE



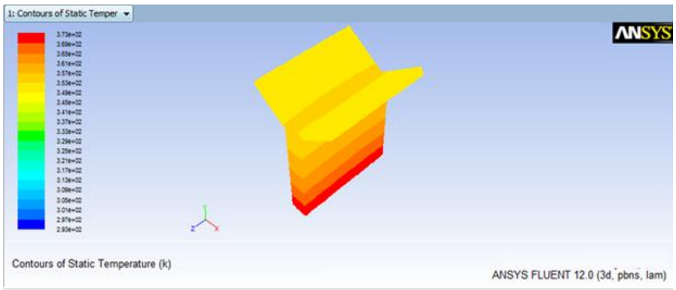
CONVERGENCE HISTORY FOR Y (120) SHAPED FIN



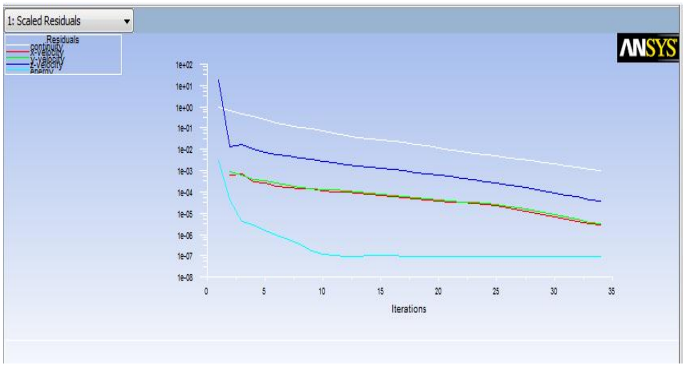
CONTOURS OF STATIC TEMPERATURE



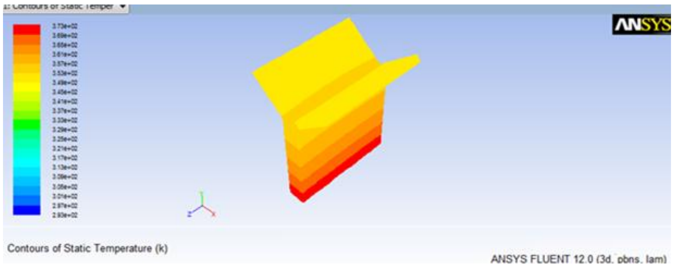
CONVERGENCE HISTORY FOR Y (110) SHAPED FIN



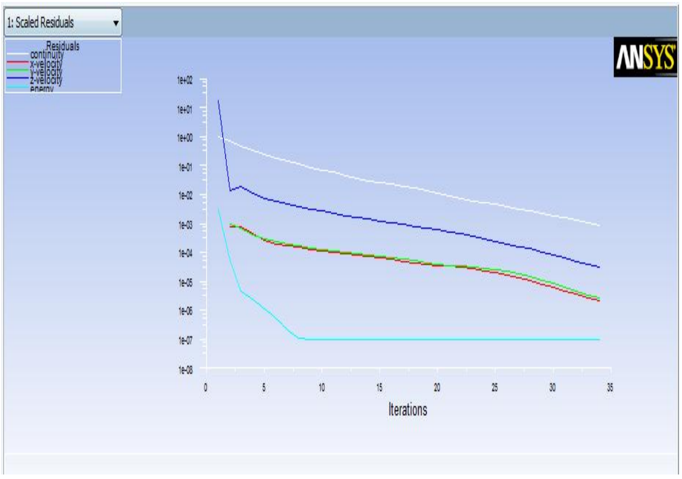
CONTOURS OF STATIC TEMPERATURE



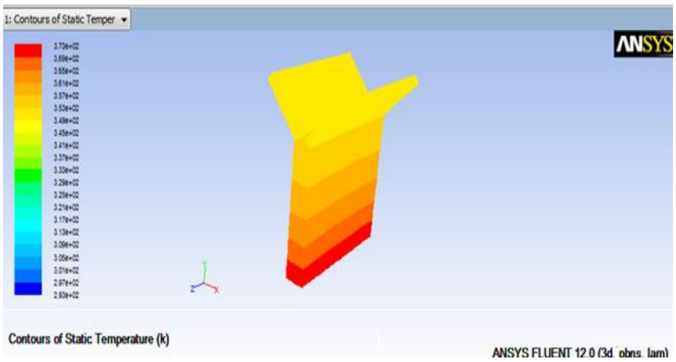
CONVERGENCE HISTORY FOR Y (100) SHAPED FIN



CONTOURS OF STATIC TEMPERATURE



CONVERGENCE HISTORY FOR Y (90) SHAPED FIN

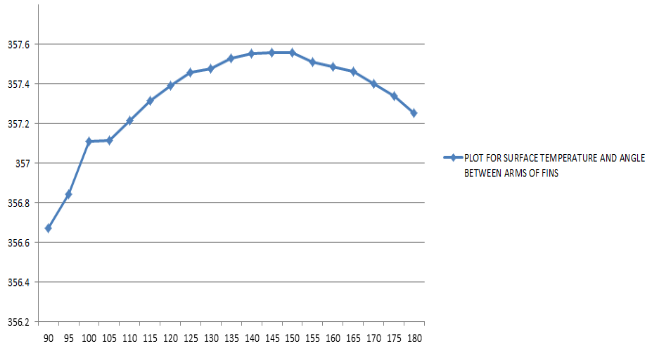


CONTOURS OF STATIC TEMPERATURE

RESULTS AND DISCUSSIONS

In this project, a study has been carried out to observe the heat dissipation in Electronic devices by using T- Shaped and Y – Shaped fins. Initially, the heat dissipation by using T – Shaped fins is calculated for a given area of the device. Later, for same dimensions of the T- shape, a Y-Shaped fin is used. It is seen that when the T- shape is changed to Y- shape, more number of fins can be accommodated within the same area of the electronic device. Thus, the space is optimized. Also, the heat dissipation is calculated for both types and the results and figures obtained are given below:

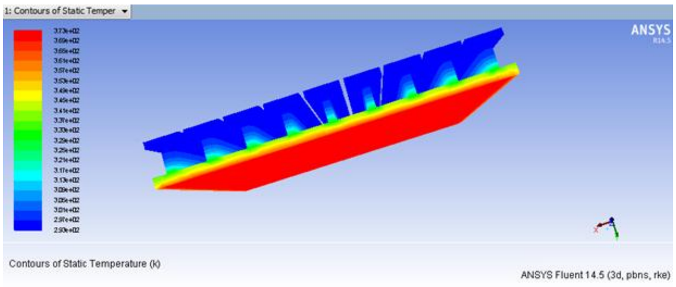
ANGLE BETWEEN ARMS OF FIN	HEAT INPUT in (K)	MINIMUM TEMPERATURE OBTAINED in(K)
180	373.15	357.254
175	373.15	357.342
170	373.15	357.402
165	373.15	357.464
160	373.15	357.488
155	373.15	357.511
150	373.15	357.560
145	373.15	357.561
140	373.15	357.553
135	373.15	357.531
130	373.15	357.477
125	373.15	357.460
120	373.15	357.390
115	373.15	357.315
110	373.15	357.214
105	373.15	357.114
100	373.15	357.112
95	373.15	356.845
90	373.15	356.673



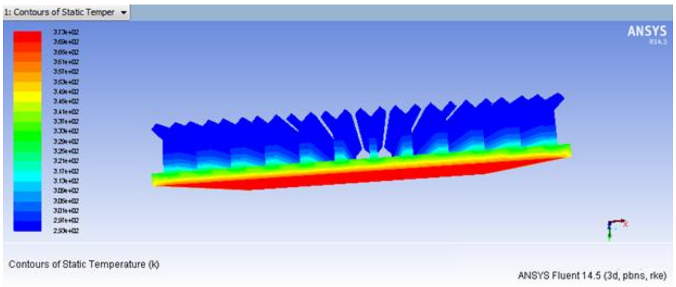
PLOT BETWEEN SURFACE TEMPERATURE AND ANGLE BETWEEN FIN ARMS

NET HEAT DISSIPATION OF T AND Y-SHAPED FINS

GEOMETRY	NET HEAT DISSIPATION
T-shape	320.81744
Y-shape	316.45139



TEMPERATURE CONTOURS OF 9 T-SHAPED FINS



TEMPERATURE CONTOURS OF 12 Y-SHAPED FINS

CONCLUSIONS

The present work is an approach to study the heat removal, using a finite element CFD code. The aim of the work is to identify a possible geometrical evolution for the optimization of T -shaped and Y- shaped fin profiles.

The results obtained, from the study of heat removal of T - shaped and Y- shaped fins configurations are presented in the previous chapter. From the results obtained the following conclusions are made:

- The heat dissipation can be increased by changing the T- shaped fin to Y – shaped.
- The maximum number of fins that can be accommodated on a flat plate can be increased from 09 (T- shaped) fins to 12 (Y – shaped) fins within the available area. Thus, space is optimized and the total heat dissipation is also increased.
- The base temperature given is 373.15 K and the net temperature obtained at the top surface of the T-shaped fin is 320.8174 K, whereas, the net temperature obtained at the top surface of the Y-shaped fin is 316.45139 K.
- The temperature difference obtained for 09 (T-shaped) fins and 12 (Y – shaped) fins within the same area is 4.36 K.

REFERENCES

A.Bejan et al.[1] presented an approach for the geometric optimization of T-shaped assemblies, in order to maximize the global thermal conductance of the assembly, subject to total volume and fin-material constraints. It is shown that every geometric feature of the assembly is delivered by the optimization principle and the constraints. These optimal features are reported in dimensionless terms for this entire class of fin assemblies. The results were developed for three assembly configurations i.e. T-shaped assembly, namely, the tau-shaped assembly where the free ends of the thinner fins are bent, the tau-shaped assembly that is narrower than the space allocated to it, and the umbrella-shaped construct containing cylindrical fins. The results show that some of the optimized geometrical features are relatively robust, i.e., insensitive to changes in some of the design parameters.

J.Marthinuss et al.[2] discussed about the air-cooled compact heatexchanger design using the book Kay's&London's Compact Heat Exchangers which contains measured heat transfer and pressure drop data on a variety of circular and rectangular passages including circular tubes, tube banks, straight fins, louvered fins, strip or lanced offset fins, wavy fins and pin fins. By this, optimized design on every possible heat exchanger design configuration was known. A comparison between fin performances was made and was found that, with avionics power densities increasing, it becomes even more critical to design an optimum compact heat exchanger to remove the heat load most efficiently.

C.L. Belay et al. [3] presented a system level thermal design methodology and applied to the design of a multi-processor enterprise server the RP8400. A systematic reduction of product risk through the careful application of available thermal design tools and techniques was the main purpose of this methodology. Numerical modeling and empirical results were presented and compared, followed by a discussion of methods for improving thermal design in future products.

B.A. Jabra et al. [4] investigated the effects of the size of modules, the presence of a cylindrical module and the missing module on the heat transfer coefficient and pressure drop characteristics of array configurations composed of individual rectangular modules for three different Reynolds numbers namely; 1690, 2250 and 2625. It was found that, using different sizes or shapes of modules in an array configuration tends to increase the Nussle number by as much as 40% for the rectangular module and 28% for the cylindrical module. The presence of a missing module in the array resulted in a 37% enhancement in the Nusselt number downstream the missing module. A drop in the pressure indicates that large size modules tend to enhance the pressure drop at their row locations by as much as 15%, while the cylindrical module tends to attenuate the pressure drop especially at low Reynolds number.

A. Bejan et al.[5] applied the method of thermodynamic optimization to several classes of simple flow systems consisting of T- and Y-shaped assemblies of ducts, channels and streams, in order to identify the geometric configuration that maximized performance subject to several global constraints.

Vivek Kumar et.al.[6] examined the effects of the configurations of the pin-fins design by a numerical physical insight into the flow and heat transfer characteristics. The governing equations are solved by adopting a control volume-based finite-difference method with a power-law scheme on an orthogonal non-uniform staggered grid. The coupling of the velocity and the pressure terms of momentum equations are solved by the computational fluid dynamics. The results show that the Elliptical Pin Fin Heat Sink has better unnaturally performance than the plate fin heat sink.

A.Horvat et al.[7] developed a fast running computational algorithm based on the volume averaging technique (VAT) to simulate conjugate heat transfer process in an electronic device heat sink. Finite volume method (FVM) was used to solve the equations. The VAT was tested and further applied to a simulation of airflow through an aluminum (Al) chip heat sink. The constructed computational algorithm enables prediction of cooling capabilities of the selected geometry. The numerical code yields sufficiently accurate results to be applicable in future optimization calculations for heat exchangers.

Sanjay Kumar Sharma et.al.[8] conducted a computational numerical analysis of air flow and heat transfer in a lightweight automobile engine, considering three different morphology pin fins. A numerical study using Ansys fluent (Version 6.3.26) was conducted to find the optimum pin shape based on minimum pressure drop and maximizing the heat transfer across the Automobile engine body. The results indicate that the drop shaped pin fins show improved results on the basis of heat transfer and pressure drop by comparing other fins. The reason behind the improvement in heat transfer by drop shape pin fin was increased wetted surface area and delay in thermal flow separation from drop shape pin fin.

S.K.Rout et.al. [9] carried out a numerical analysis on internally finned axi-symmetric tube heat exchanger using Finite Volume Method. A computational fluid dynamics (CFD) program named FLUENT was used to estimate the performance of the heat exchanger with different fin shapes, sizes and numbers. The three different shaped fins considered are rectangular, triangular and T-shaped fins and it is found that the wall temperature is least for triangular shaped fins, compared to rectangular and T-shaped fins. The optimum fin number for which the wall temperature is minimum and the heat transfer is maximum is found to be 10 and the optimum fin height is found to be 0.026m.

Qusay R.Al-Hagag et.al[10] presented a method for designing the optimum thermal performance of heat sinks with uniform cross section of plate by using the Heat Capacity for the material. The model was used to explore the optimal dimensions of heat sink that cooling natural convection. This heat sink optimization study allowed for a determination of the heat sink geometry, which produced a minimum thermal resistance, while producing the highest heat dissipation. It was found from the results that the best design was a (50 mm x 50 mm x 5 mm) aluminum heat sink with a temperature input of (50oC) and (7) fins with (0.5 mm) fin thickness. ANSYS 5.4 finite element code was used to analyze the thermal behavior of a heat sink for the given optimum dimensions.

Alurarasan R. et.al.[11] developed an optimal design of the heat sink on a parallel plate heat sink considering the geometric parameters such as fin height , fin thickness , base height and fin pitch with a constant length and width of a heat sink using computational fluid dynamics study. The simulation was carried out with commercial software provided by fluent Inc. Experimental studies were carried out with a parallel plate heat sink to validate the heat sink model. The results obtained in the experimental studies have been compared with the simulation results and found to be in good agreement. In their study, the geometric parameters fin height, fin thickness, base height and fin pitch were found to be optimal at 48 mm, 1.6 mm, 8 mm and 4mm respectively for an efficient heat sink design.