iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** IV **Month of publication:** April 2018

DOI: http://doi.org/10.22214/ijraset.2018.4471

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

Steady State Thermal Analysis of a Heat Sink with Rectangular Pin Fin

V. Himachandra Raju[1], Ch. Srinivasa Rao[2]
[1] M. Tech. Scholar, Department of Mechanical Engineering, Kallam Haranadhareddy Institute of Technology, Chowadavarm, Guntur, Andhra Pradesh, INDIA
[2] Professor, Department of Mechanical Engineering, Kallam Haranadhareddy Institute of Technology, Chowadavarm, Guntur, Andhra Pradesh, INDIA.

***Abstract:* Heat sink is a passive heat exchanger device or substance that absorbs excessive or unwanted heat generated by an electronic or a mechanical device and transfers it to a fluid medium, often air or a liquid coolant, where it is dissipated away from the device, thereby allowing regulation of the device's temperature at optimal levels. A heat sink is designed to maximize its surface area in contact with the cooling medium surrounding it, such as the air. Air velocity, choice of material, protrusion design and surface treatment are factors that affect the performance of a heat sink. A heat sink is usually made out of Copper or Aluminium. Aluminium heat sinks are used as a low-cost, lightweight alternative to copper heat sinks, and have a lower thermal conductivity than copper. Therefore, in the present work, Aluminium is chosen as the material for the heat sink.**

Computational Fluid Dynamics (CFD) codes are widely used as a tool of thermal analysis. With the advent of Computational Fluid Dynamics (CFD) in the recent years, flow and heat transfer computations have become quite readily possible. In particular, with the recent introduction of high power workstations and personal computers the cost of such computations has been drastically reduced and as a result many CFD codes have come into the market.

In the present investigation, CFD analysis of a heat sink has been carried with a rectangular pin fin for the study of fluid properties for steady state thermal analysis. The analysis involves simulation of Temperature distribution of the heat sink, Total heat flux distribution of heat sink and finding the dynamic pressure variation, turbulence kinetic variation, turbulence eddy dissipation and the surface Nusselt number at the tip of the pin fins for a velocity of 3 m/s and 4 m/s. The results are validated with theoretical results. There is a good agreement between the numerical and theoretical results.

***Keywords:* Steady State Thermal Analysi, *Computational Fluid Dynamics (Cfd)*, Heat Sink, Fluid Flow, Rectangular Pin Fin.**

I. INTRODUCTION

Heat sink research and development has had a long history but it is still continuing with efforts to improve design and performance by innovations in modelling and analytical techniques. Development of various heat sink designs along with various fin geometries has revolutionized the heat sink industry. Much work has been done in recent years to characterize and optimize the performance of fanned heat sinks amongst others. However, this work has focused on large scale applications, with no effort to date focused on scales appropriate to handheld electronic devices.

Computational Fluid Dynamics (CFD) codes are widely used as a tool of thermal analysis.CFD solutions of high spatial and temporal resolutions can be obtained on a desktop computer or even a laptop. However, CFD-based thermal analysis is not necessarily easy to perform where the object of analysis is geometrically complex. With the advent of Computational Fluid Dynamics (CFD) in the recent years, flow and heat transfer computations have become quite readily possible. In particular, with the recent introduction of high power workstations and personal computers the cost of such computations has been drastically reduced and as a result many CFD codes have come into the market.

In general, validation and benchmarking of CFD codes has been an on-going research area attracting a lot of attention from both users and code developers. Studies on micro channel flows in the past decade are categorized in to various topics.

II. STEADY STATE THERMAL ANALYSIS

Thermal and structural analysis of tree shaped fin array has been performed by considering the fins with slots and without slots [1]. The effect of material has been studied for the same geometries by taking aluminium alloy, structural steel and copper alloy as the materials. Thermal analysis of a heat sink has been carried for avionics applications with variable material [2]. Experimental, numerical, and analytical study of the optimal spacing between cylinders in cross-flow forced convection has been carried [3] and the optimal cylinder-to-cylinder spacing was determined by maximizing the overall thermal conductance between all the cylinders and the free stream. Fn-spacing effects in annular-finned tube heat exchangers have been carried by numerical study [4]. Experimental and CFD analysis of an elliptical pin fin heat sink has been carried using ANSYS Fluent 12.1 software varying the dimension of elliptical pin fin i.e., by varying the cross-section area. The results showed that for all the velocities 2mm minor axis elliptical pin fin had better thermal resistance and pressure drop [5].

CFD simulations for the Radial Heat Sink were successfully developed. There were a lot of requirements while simulating a case of Radial Heat sink. The result of the analysis showed that the pressure decreases along the fin length and maximum heat transfer takes place through the middle portion of the fin which is receiving maximum air flow [6]. The heat transfer correlations in the steady flow regime for the constant temperature and constant heat flux boundary conditions on the solid square cylinder in cross flow have been studies with cross flow placed symmetrically in a planar slit for a range of conditions [7]. CFD analysis has been performed by applying a load of 5 W to the heat sink and varied number of pin fins and found that on increasing total number of fins, the total heat transfer rate also increases [8]. Experimental and theoretical investigations have been carried for the study of the natural convection heat transfer from vertical rectangular fin arrays with and without notch at the center [9]. The notches of different geometrical shapes have also been analyzed for the purpose of comparison and optimization. Heat transfr by natural convection from triangular notched fin array have been observed for different notch geometries such as fin without notch, fin with 20% notch with area compensation and fin with 40% notch with area compensation with respect to various parameters such as height, length, notch dimension, fin spacing and fin thickness [10]. The studies showed that heat transfer coefficient is lower in notched fin as compared to without notch. And the heat transfer increases with increase in notch size with area compensation. Experimental analysis of incline narrow plate fins heat sink under natural convection have been carried with respect to aspect ratio and different heater input wattage [11]. The results showed that natural convection heat transfer increases with heat input and the convective heat transfer increases with aspect ratio. The conjugate conduction-convection heat transfer from a three-dimensional array of rectangular perforated fins with square windows arrangement on the lateral surface of fins and fluid flow has been numerically studied by using Navier-Stokes equations and RNG k-ε model [12]. Their results showed that the perforated fins have higher total heat transfer and considerable weight reduction compared to those with solid fins. The performance of a desktop heat sink under forced convective conditions of air cooling has been studied using ANSYS-FLOTRAN [13].

III. ANALYTICAL MODELLING

A. Heat Transfer Principle

Heat sink works on the principle of Fourier's law of heat conduction, simplified to a one-dimensional form in the *x*-direction, shows that when there is a temperature gradient in a body, heat will be transferred from the higher temperature region to the lower temperature region [5]. The rate at which heat is transferred by conduction q_k, is proportional to the product of the temperature

gradient and the cross-sectional area through which heat is transferred

$$q_k = -kA\frac{dT}{dx} \quad \text{……………......2.1}$$

Consider a heat sink in a duct, where air flows through the duct. It is assumed that the heat sink base is higher in temperature than the air. Applying the conservation of energy, for steady-state conditions, and Newton's law of cooling to the temperature nodes shown in the diagram gives the following set of equations:

$$Q = mc_{p,in}(T_{air,out} - T_{air,in}) \quad \text{……………2.2}$$

$$Q = \frac{T_{hs} - T_{air,av}}{R_{hs}} \quad \text{………………2.3}$$

Where

$$T_{air,av} = \frac{T_{air,in} + T_{air,out}}{2} \quad \text{…………2.4}$$

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

The heat sink thermal resistance model consists of two resistances, namely the resistance of the heat sink base, R_b, and the resistance of the fins R_f. The heat sink thermal resistance can be written as follows if the source is a uniformly applied to the heat sink base. If it is not then the base resistance is primarily spreading resistance:

$$R_b = \frac{t_b}{kA_b} \quad \text{.................2.5}$$

Where t_b is the heat sink base thickness, k is the heat sink material thermal conductivity and A_b is the area of the heat sink base.

The thermal resistance from the base of the fins to the air, R_f, can be calculated by using the following formulas:

$$R_f = \frac{1}{n h_f W_f (t_f + 2\eta_f L_f)} \quad \text{.........2.6}$$

Once the heat sink and fin resistances are known, and then the heat sink thermal resistance can be calculated as

$$R_{hs} = R_b + R_f \quad \text{...........2.7}$$

Using the mean air temperature is an assumption that is valid for relatively short heat sinks. When compact heat exchangers are calculated, the logarithmic mean air temperature is used and m is the air mass flow rate in kg/s.

The above equations show that
When the air flow through the heat sink decreases, this results in an increase in the average air temperature. This in turn increases the heat sink base temperature. And additionally, the thermal resistance of the heat sink will also increase. The net result is a higher heat sink base temperature. The inlet air temperature relates strongly with the heat sink base temperature. For example, if there is recirculation of air in a product, the inlet air temperature is not the ambient air temperature. The inlet air temperature of the heat sink is therefore higher, which also results in a higher heat sink base temperature. If there is no air flow around the heat sink, energy cannot be transferred.A heat sink is not a device with the “magical ability to absorb heat like a sponge and send it off to a parallel universe”. Natural convection requires free flow of air over the heat sink. If fins are not aligned vertically, or if fins are too close together to allow sufficient air flow between them, the efficiency of the heat sink will decline. The Sketch of a heat sink in a duct used to calculate the governing equations from conservation of energy and Newton’s law of cooling is as shown in figure-2.1

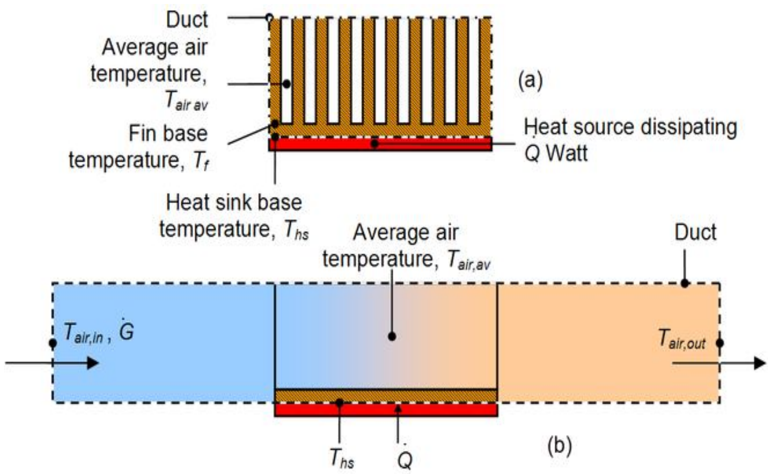

Figure- 2.1: Sketch of a heat sink in a duct used to calculate the governing equations from conservation of energy and Newton’s law of cooling [14]

B. Assumptions For Thermal Analysis Of Heat Sink

The performance of the heat sink is determined by the various parameters like the heat transfer coefficient, Fin efficiency, etc [2].
Heat Transfer Coefficient (h):
To define the heat transfer coefficient consider the Newton’s law of convection given by the equation

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

$$Q = hA_s(T_s - T_a) \qquad \therefore h = \frac{Q}{A_s(T_s - T_A)} \text{.............2.8}$$

Fin Efficiency

$$\eta = \frac{\tanh mL}{mL} \text{............2.9}$$

Where $m = \sqrt{\frac{hP}{k_f A}}$

h = heat transfer coefficient of fin, P = Perimeter of fin = 2(B+D), L = Length of pin fin.
K_f = thermal conductivity of Aluminium fin = 202.4 W/mK, A= Area of fin = B x D
Computational system gives a subjective or even quantitative forecast of fluid streams. Re-enactment can give forecast of stream wonders utilizing the ANSYS 15.0 workbench software for every single sought quantity, with high determination in space and time and basically any issue and practical working conditions. On the other hand, if discriminating, the outcomes should be worked.
The modelling of pin fin heat sinks is made by ANSYS Workbench software. This analysis is based on the following assumptions:

1) The fins are with adiabatic tip.
2) The fluid, air is assumed to be incompressible throughout the process. The airflow is normal to the fins.
3) Air properties are taken at film temperature.
4) There are no heat sources within the fin itself.
5) The radiation heat transfer is negligible.
6) The temperature at the base of the fin is uniform.

IV. FEM OF A HEAT SINK

In the present work, the software ANSYS 15.0 has been used to model and simulate Heat transfer and CFD analysis of heat sink with rectangular pin fin

A. Preparation of the Design model

The designs of heat Sink with rectangular pin fins with inline Staggered arrangements is done in ANSYS Workbench 15.0 in STEP format. A flat platform of 101.43 X 101.43 X 5 mm is common in all designs. Fin height for all models is 20 mm. There are a total number 81 pin fins in line arrangement with 9 pin fins in each row and 9 pin fins in each column with fin spacing of 6.76 mm between them.

B. Geometry

The top view and front views of the 2D geometry and 3D model are shown in Figures-3.1,3.2 and Figure-3.3 respectively.
The detailed specifications of the heat sink design are shown in the below table 3.1. The material properties for modelling of the heat sink with rectangular pin fin are shown in the Table- 3.2 and properties of air are tabulated and shown in Table-3.3

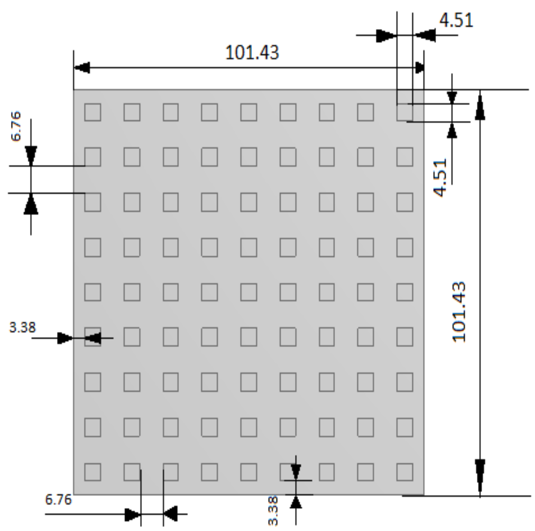

Figure-3.1: Top view of the model of the heat sink

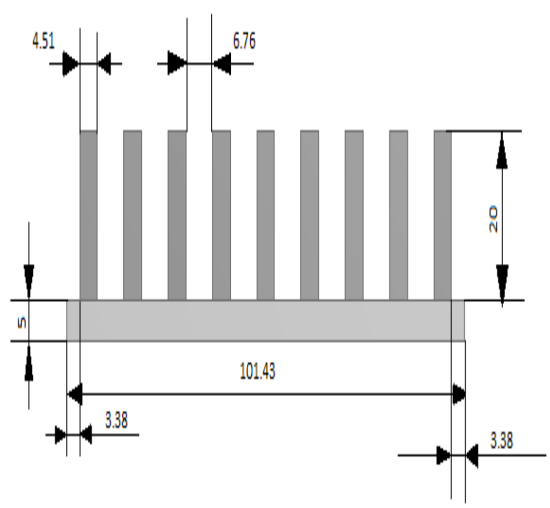

Figure-3.2: Front view of the model of the heat sink

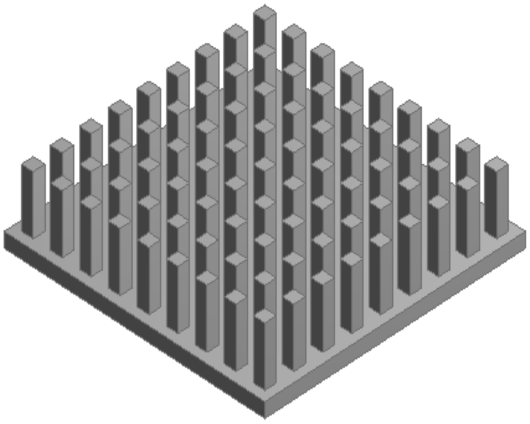

Figure-3.3: 3D model of a heat sink

Quantity	Dimensions	
Foot print (mm^2)	L x W	101.43x101.43
Base plate thickness(mm)	t_b	5
Overall height of fin (mm)	t_f	20
Fin transverse length (mm^2)	L	4.51
Fin longitudinal length (mm^2)	W	4.51
Horizontal Pitch (mm)	S_L	6.76
Vertical pitch (mm)	S_T	6.76

Table-3.1: Heat sink design specifications

S.No	Properties	Value
1.	Density (ρ)	2719 kg/m^3
2.	Specific heat (C_p)	871 J/KgK
3.	Thermal conductivity(k)	202.4 W/mK

Table-3.2: Properties of Aluminium

S.No	Properties	Value
1	Density (ρ)	1.225 kg/m^3
2	Specific heat (C_p)	1006.43 J/KgK
3	Thermal conductivity(k)	0.0242 W/mK
4	Viscosity (Kg/m-s)	1.7894e-05

Table-3.3: Properties of air

C. Mesh Generation & Simulation

The heat sink model is imported in to the work bench design modeler and meshed with a four node three dimensional tetrahedron element SOLID72the meshed model of the heat sink is as shown in the below figure-3.4 and the mechanical APDL (ANSYS Parametric Design Language) solver is used to mesh the heat sink.

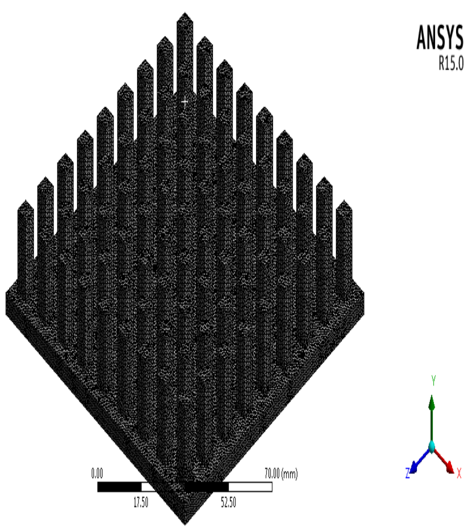

Figure-3.4: Meshing of the heat sink model

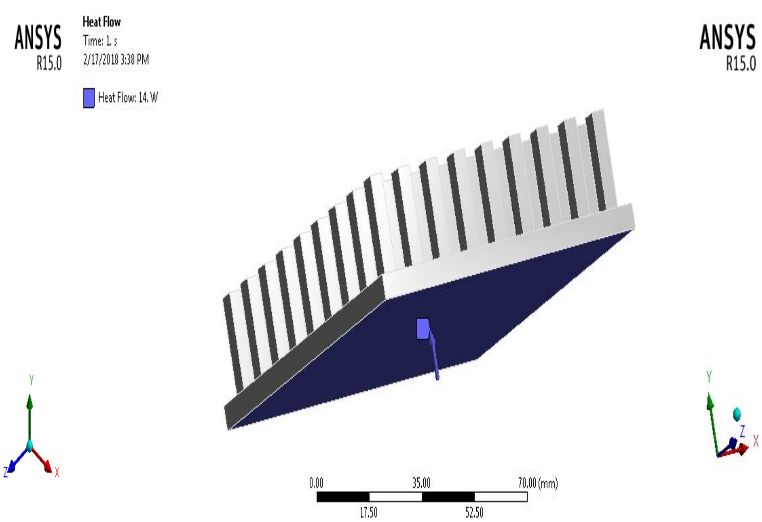

Figure-3.5: Loading and boundary conditions – Heat flow

Heat sink a heat flow of 14 Watts is given at the bottom surface in figure-3.5, the heat load reaches to 14 W and convection of 6e-6 $W/mm^2.^0C$ to the remaining faces shown in figure-3.6.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue IV, April 2018- Available at www.ijraset.com

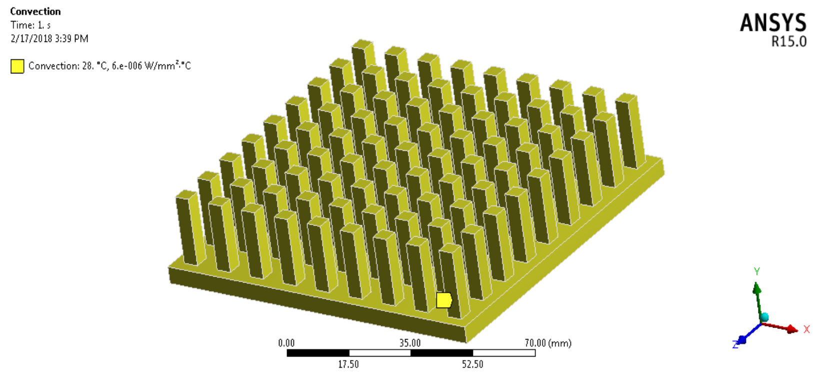

Figure-3.6: Loading and boundary conditions – Convection

After applying the boundary conditions, run the simulation for 1 second, we will obtain the results of temperature distribution and heat flux contours with maximum and minimum values.

V. RESULTS AND DISCUSSION

For steady state thermal analysis an amount of heat flow of 14 W is given as input to the base plate. A convection coefficient of 6e-6 W/mm^2.^{0}C has been applied to the remaining surfaces of the heat sink. Using Mechanical APDL solver, the maximum and minimum temperatures, heat fluxes of the heat sink within one second have been estimated. Then the result pertaining to steady state thermal analysis involves temperature and heat flux contours are explained are explained below.

A. Temperature Distribution

The temperature is distribution obtained with 84.41^{0}C of maximum temperature at the centre of the base plate and minimum temperature of 83.957^{0}C is obtained at the fin tips at the tip of the fins around the sides of the plate. The simulated result is as shown in figure-4.1.

B. Heat Flux Distribution

The total heat flux distribution of orientation to Y-axis obtained with maximum of 0.0080043 W/mm^2.^{0}C and minimum of 0.00044351 W/mm^2.^{0}C from the simulated result as shown in figure-4.2

We can observe the dynamic pressure variation at the pin fin tips at 3 m/s and 4 m/s velocity is shown in the below figure-4.3

The turbulence kinetic variation at the tip of the fins at air velocities of 3 m/s and 4 m/s are compared and shown in figure -4.4.we can clearly observe the turbulence kinetic energy is maximum for 3 m/s and minimum for 4 m/s.

The turbulence eddy dissipation at the tip of the fins at air velocities of 3 m/s and 4 m/s are compared and shown in figure-4.5 and the turbulence kinetic eddy dissipation for 4 m/s is maximum and 3 m/s is minimum.

The surface Nusselt number at the tip of the pin fins for 3 m/s and 4 m/s are compared as shown in the below figure-4.6 and the surface Nusselt number for 4 m/s is higher than the Nusselt number for 3 m/s air velocity.

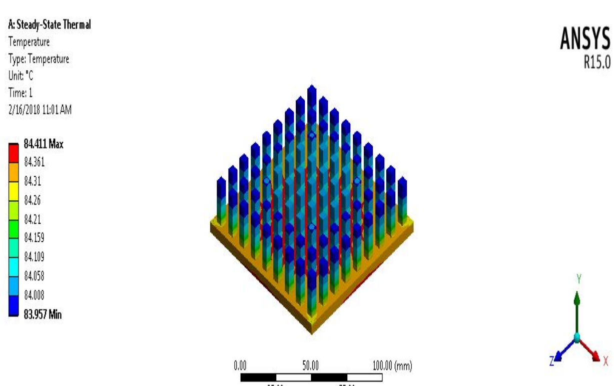

Figure-4.1: Temperature distribution of heat sink

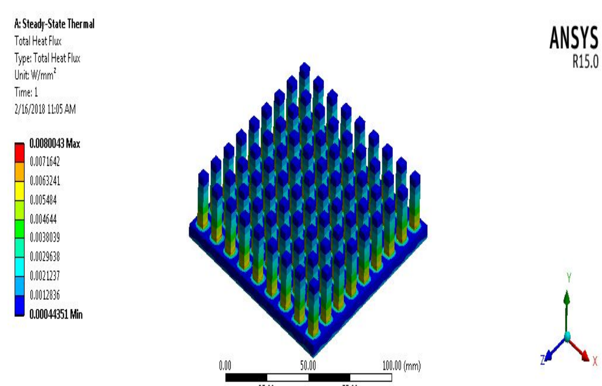

Figure-4.2: Total heat flux distribution of heat sink

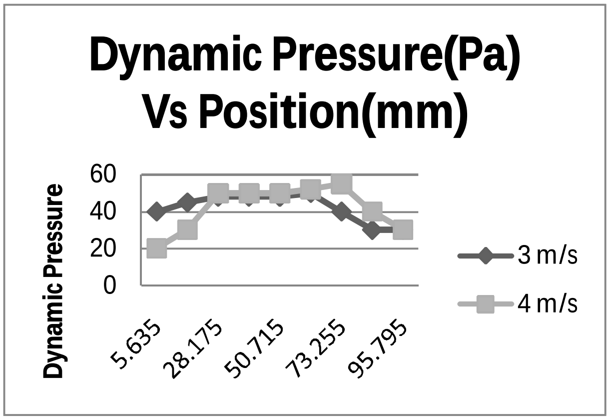

Figure-4.3: Dynamic Pressure Vs Position

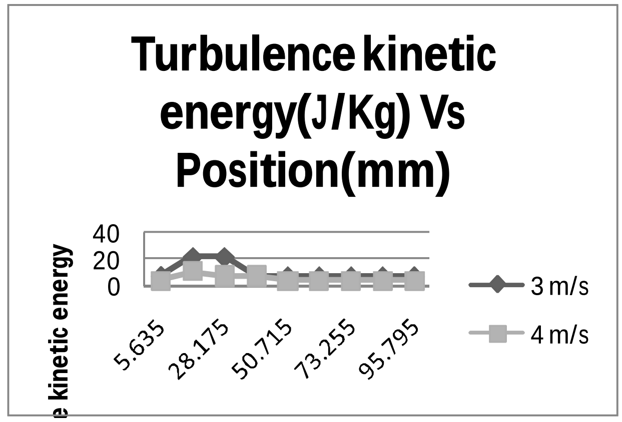

Figure-4.4: Turbulence Kinetic energy Vs Position

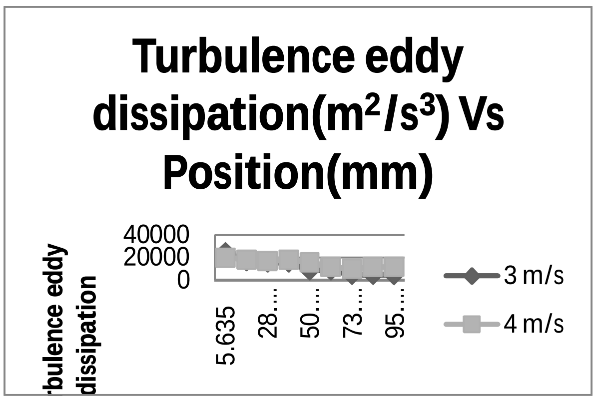

Figure-4.5: Turbulence eddy dissipation Vs Position

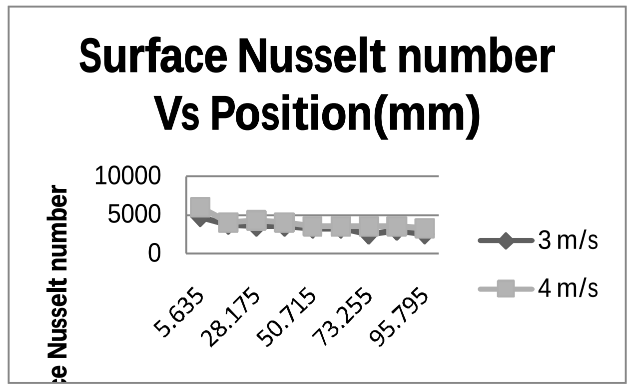

Figure-4.6: Surface Nusselt number Vs Position

V. CONCLUSIONS

In the present work, the Finite Element Analysis with Computational Fluid Dynamics (CFD) of a Heat Sink with rectangular pin fin has been carried for steady state thermal analysis. The temperature distribution and total heat flux distribution of heat sink has been simulated successfully. The dynamic pressure variation, turbulence kinetic variation, turbulence eddy dissipation and the surface Nusselt number at the tip of the pin fins for a velocity of 3 m/s and 4 m/s have been calculated and validated with theoretical results.

REFERENCES

[1] K. Kumar, P. Vinay, R. Siddhardha, Thermal and Structural Analysis of Tree Shaped Fin Array, Int. Journal of Engineering Research and Applications, Vol-3, Issue- 6, Dec 2013

[2] Bopanna K, D Nizamuddin Ali khan, Experimental and CFD Analysis of Aluminum Heat Sinks for Avionics Applications. IJIRST –International Journal for Innovative Research in Science & Technology| Volume 3 | Issue 02 | July 2016.

[3] Stanescu, G., Fowler, A. J. and Bejan, A. The optimal spacing of cylinders in free-stream cross-flow forced convection. Int. J. Heat Mass Transfer, 1996, 39,311–317.

[4] Mon, M. S. and Gross,U. Numerical study of fin-spacing effects in annular-finned tube heat exchangers. Int. J. HeatMass Transf., 2004, 47, 1953–1964.

[5] V. Kumar, Dr. V. N. Bartaria, CFD Analysis of an Elliptical Pin Fin Heat Sink Using Ansys Fluent V12.1, International Journal of Modern Engineering Research (IJMER), Vol-3, Issue 2, April 2013

[6] Mirza Md. Symon Reza, CFD Analysis of Forced Convection over Radial Heat Sink, Journal of Modern Science and Technology Vol. 4. No. 1. September 2016 Issue. Pp. 135 – 14

[7] Anuj Kumar,Amit Dhiman,Laszlo Baranyi, CFD analysis of power-law flow and heat transfer around a confined semi-circular cylinder , International Journal of Heat and mass Transfer 82 (2015) 159-169.

[8] Deepak Gupta, Momin Nausheen, A.D. Dhale , CFD Analysis & Simulation of Pin Fin for Optimum Cooling of MotherBoard, IJEDR,Volume 2, Issue 2,2014 2321-9939.

[9] Shivdas S. Kharche, Hemant S. Farkade, Heat Transfer Analysis through Fin Array by Using Natural Convection International Journal of Emerging Technology and Advanced Engineering, Volume 2, 2014.

[10] Sachin R. Pawar, R. Yadav, Computational Analysis of Heat Transfer by Natural Convection from Triangular Notched Fin Array, IJST- International Journal of Science Technology & Engineering, Vol-1, Issue 10, April 2015.

[11] A. A. Walunj, D. D. Palande, Experimental Analysis Of Inclined Narrow Plate- Fins Heat Sink Under Natural Convection, IPASJ International Journal Of Mechanical Engineering(IIJME), Vol. 2, Issue- 6, June 2014.
[12] Shaeri, M.R., Yaghoubi, M., Numerical analysis of turbulent convection heat transfer from an array of perforated fins. Int. J. Heat Fluid Flow, 2009, 30 (2), pp. 218-228, 2009
[13] Bhramara P., T. K. K. Reddy, K. Prashanth Reddy, CFD Analysis of Desk Top Heat Sink. Journal of Enhanced Heat Transfer, 15(3) 1–12, 2008
[14] Nilesh Khamkar, Avinash Waghmode, Atul Joshi, Pramod Supekar, Dr. Ashwini Kumar, Prof. Kiran Londhe and Prof. Vipul R.Kaushik,. Heat Sink design for optimal performance of compact electronic appliances-A Review, Journal for Advanced Research in Applied Sciences. Volume 4, Issue 5, Oct-2017 Pages: 13-21