iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** III **Month of publication:** March 2018

DOI: http://doi.org/10.22214/ijraset.2018.3453

CFD Modeling and Analysis of a Heat Sink with Rectangular Pin Fin

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

Abstract: In semiconductor and optoelectronic devices such as lasers and light emitting diodes, the heat dissipation ability is less. In such cases, the heat sinks are used as heat dissipation devices. In computers, heat sinks are used to cool central processing units or graphics processors. A heat sink is a passive heat exchanger device or substance that absorbs excessive or unwanted heat generates and usually made out of Copper or Aluminium. Copper is used because it has many desirable properties for thermally efficient and durable heat exchangers. Aluminium heat sinks are used as a low-cost, lightweight alternative to copper heat sinks, and have a lower thermal conductivity than copper.

CFD (Computational Fluid Dynamics) is a segment of Fluid mechanics that involves using numerical techniques to solve fluid flow problems.CFD analysis can also be used for product design development and process troubleshooting. In the present investigation, CFD analysis of a heat sink has been carried with a rectangular pin fin for the study of fluid properties like variation in temperature, pressure, turbulent kinetic energy with variation of input velocities. The results are tabulated and are validated with theoretical results.

Keywords: Computational Fluid Dynamics, Heat Sink, Transient analysis, 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 modeling 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.

A. CFD flow analysis

Increasing the flow velocity reduces the thermal resistance and increases the pressure drop simultaneously and circular pin fin heat sink displayed higher heat transfer rate than the plate fin heat sink [1]. The maximum temperature values obtained by CFD analysis of heat sink using simulation software ANSYS Fluent are slightly more than the theoretical values with some assumptions [2]. CFD analysis of Splayed pin fin heat sinks for Aluminium, Copper and Hybrid materials were compared and the results concluded that in the sense of junction temperature splayed pin fins are efficient and The Hybrid pin fin heat sinks have better performance than aluminium and copper pin fin heat sinks [3]. 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 [4].

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 III, March 2018- Available at www.ijraset.com

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 [5]. Experimental and numerical investigations of pressure drop and heat transfer characteristics of single-phase laminar flow in 231μm by 713 μm channels has been carried [6]. It was concluded that hyoid structure is giving better heat transfer coefficient from numerical analysis and also on simulation [7]. The natural convection heat transfer coefficient has been enhanced by using V- fin array using ANSYS CFX and experimentally where the plate fins where the fins were arranged at an inclination of 60^0 [8]. The performance of a desktop heat sink under forced convective conditions of air cooling has been studied using ANSYS-FLOTRAN [9].The shape optimization of the plate-fin type heat sink with an air deflector has been performed numerically to minimize the pressure loss subjected to the desired maximum temperature and geometrical constraints [10]. A numerical simulation of a heat sink using the k–ε turbulence model with fins of non-uniform height with a confined impingement cooling has been performed to examine the effects of the fin shape of the heat sink on the thermal performance [11].

II. MODELING AND DESIGN OF HEAT SINK

The design parameters include the heat sink material, the number and geometry of the fins and their alignment and the base plate thickness.

A. Heat Sink Material

Heat sinks are made from a good thermal conductor such as copper or aluminum alloy. Copper (401 W/m K at 300 K) is significantly more expensive than aluminum (237 W/m K at 300 K) but is roughly twice as efficient as thermal conductor. Aluminum has the significant advantage that it can be easily formed by extraction, thus making complex cross-sections possible. Aluminum is also much lighter than copper, offering less mechanical stress on delicate electronic components. Therefore, in the present work, Aluminium is chosen as the material for the heat sink.

B. Creation of the 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.

The detailed specifications of the heat sink design are shown in the below Table-2.1

C. Geometry

The top view and front views of the 2D geometry and 3D model are shown in Figures-2.1,2.2 and Figure-2.3 respectively.

The material properties for modelling of the heat sink with rectangular pin fin are shown in the Table- 2.2 and properties of air are tabulated and shown in Table-2.3

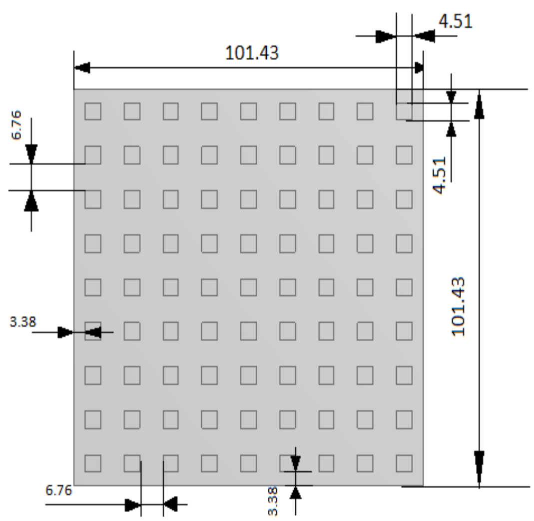


Figure-2.1: Top view of the model of the heat sink

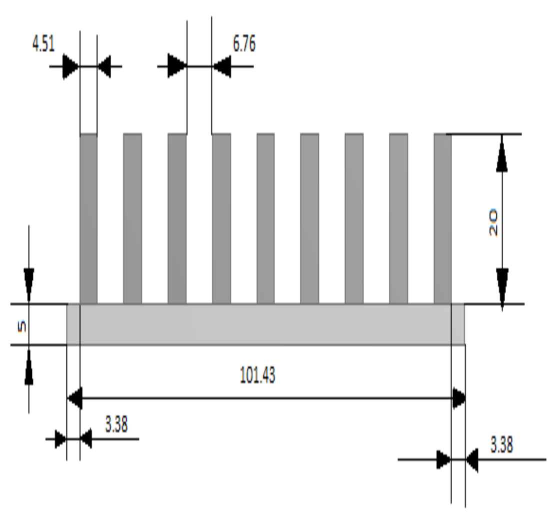


Figure -2.2: Front view of the model of the heat sink

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 III, March 2018- Available at www.ijraset.com

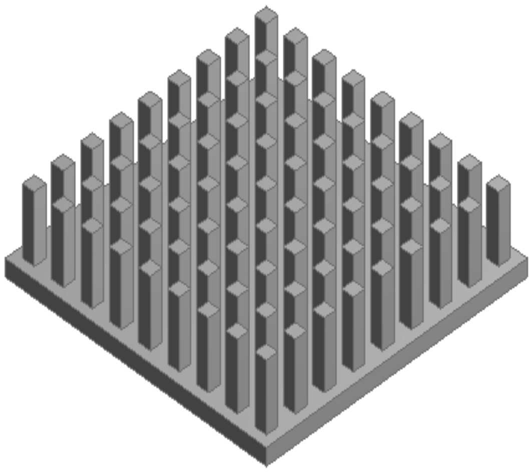


Figure-2.3: 3D model of a heat sink

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-2.1: 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-2.2: 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-2.3: Properties of air

III.MESH GENERATION & SIMULATION

A four node three dimensional tetrahedron element SOLID72 has been used to mesh the geometry of both the heat sink and enclosure. SOLID72 is well suited to model irregular meshes (such as produced from various CAD/CAM systems). The element is defined by four nodes having six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z directions. The element also has stress stiffening capability.

A. Boundary Conditions

In this analysis heat sink is modeled as solid domain with heat source of 14W. In this case heat sink material considered as aluminum. The analysis is done at atmospheric temperature of 301.15 K. Boundary conditions are entered as follows:

1) Base plate bottom surface: Heat Load of 14W and aluminium metal properties is assigned.
2) Base top (wall): Base top is receiving heat from the bottom surface of the heat sink
3) Fin bottom, front face, left, right, rear face (Walls): Heat transfer to surrounding atmosphere by convection.
4) Inlet (velocity inlet): Air enters into the Heat sink with 3 and 4 m/s in X direction according to the geometry.
5) Outlet (Pressure Outlet): After passing through the heat sink air enters into atmosphere, so at outlet atmospheric pressure is assumed.
6) After applying the above boundary conditions. Simulation is performed under transient state conditions till the convergence is reached.

B. Transient state CFD analysis of heat Sink at inlet air velocity of 3 m/s

An enclosure is created around the heat sink to study the flow properties of air flowing over the heat sink is shown in the below figure-3.1. The generated mesh is shown in figure-3.2

The flow analysis is in transient state. An inlet velocity of 3 m/s and outlet velocity of 0 m/s are given as boundary conditions of enclosure and temperature of 357.61 K to the base plate of the heat sink with a wall thickness of 1e-18 m with heat generation rate of 0.4 W/m^3. The models selected here are energy equation and k-epsilon (2 equations) viscous model. The SIMPLE solution

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 III, March 2018- Available at www.ijraset.com

method is used and solution is initialized. Calculation is run with fixed time step size of 0.1 s with 100 time steps of 2 iterations per time step with 1 report interval and 1 update interval.

After the completion of the calculation in the preprocessing the graphs of pressure, temperature, surface Nusselt number and turbulent kinetic energy with respect to position are obtained. In the post processing, the velocity vectors and contours of pressure, temperature etc., are obtained in the results.

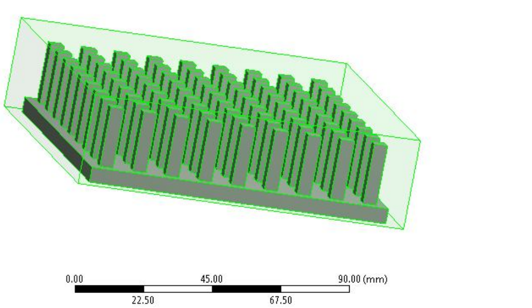


Figure-31: Enclosure around the heat sink

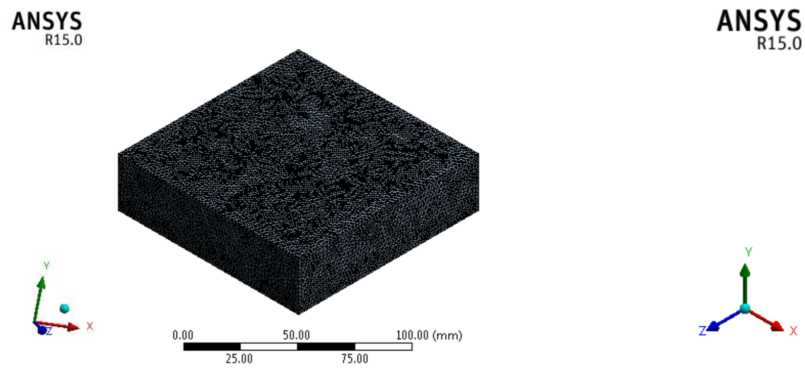


Figure-3.2: Mesh generation of heat sink with enclosure

IV. RESULTS AND DISCUSSION

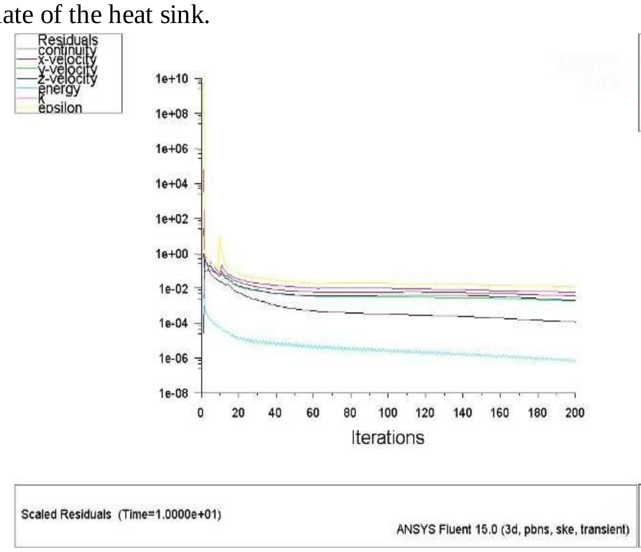
A. *Transient state CFD analysis of heat sink at inlet air velocity of 4 m/s*

This process is similar to the above CFD analysis the velocity of air is changed. An inlet velocity of 4 m/s and outlet velocity of 0 m/s are given as boundary conditions of enclosure and temperature of 357.61 K to the base plate of the heat sink with a wall thickness of 1e-18 m with heat generation rate of 0.4 W/m^3. The models selected here are energy equation and k-epsilon (2 equations) viscous model. The SIMPLE solution method is used and solution is initialized. Calculation is run with fixed time step size of 0.1 s with 100 time steps of 2 iterations per time step with 1 report interval and 1 update interval.

After the completion of the calculation in the preprocessing the graphs of pressure, temperature, surface Nusselt number and turbulent kinetic energy with respect to position are obtained. In the post processing, the velocity vectors and contours of pressure, temperature etc., are obtained in the results.

B. *Cfd analysis of heat sink in Transient State at Inlet air Velocity of 3 m/s:*

As explained earlier, the CFD analysis involves variation of velocity and the residuals Vs velocity obtained at velocity 3 m/s for 10 seconds is shown in figure-4.1. The maximum temperature obtained from thermal analysis is taken as the initial temperature at base plate of the heat sink.



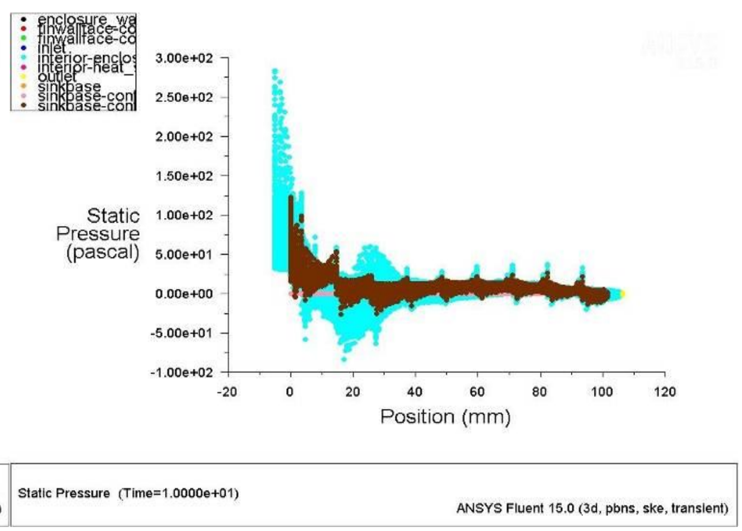
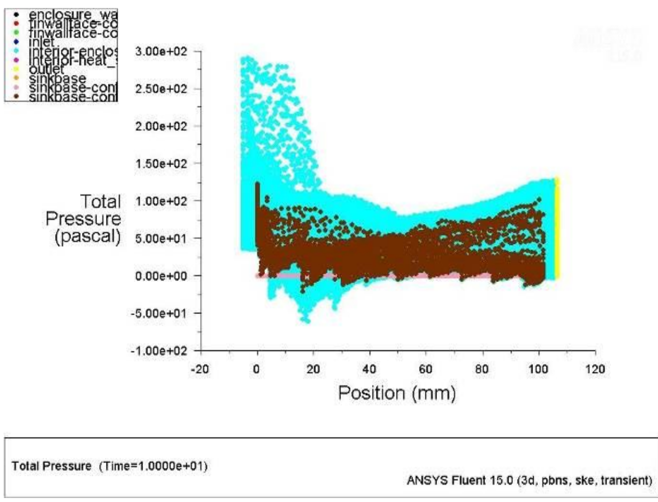


Figure-4.1: Scaled residuals Vs Iterations (air velocity of 3 m/s) Figure-4.2: Static pressure Vs Iterations(air velocity of 3 m/s)

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 III, March 2018- Available at www.ijraset.com

The static pressure Vs position graph is obtained after the calculation of the solution is completed. The static pressure is higher at inlet and reaches the minimum at the 17 mm position as shown in the figure-4.2 and we can clearly observe a sudden pressure drop. The total pressure variation can be observed in the graph of total pressure Vs position, the total pressure of air reaches its maximum value at the inlet and gradually decreases and also obtains negative pressure and then gradually increases with minimum at the center of the heat sink fin. The outlet total air pressure is 125 Pascal obtained from the figure-4.3.



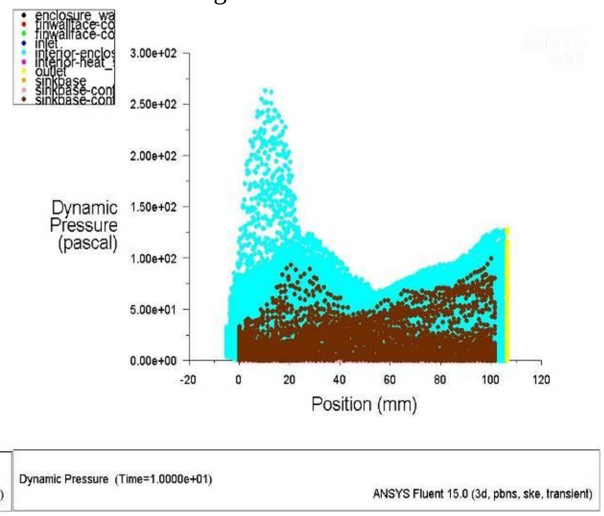
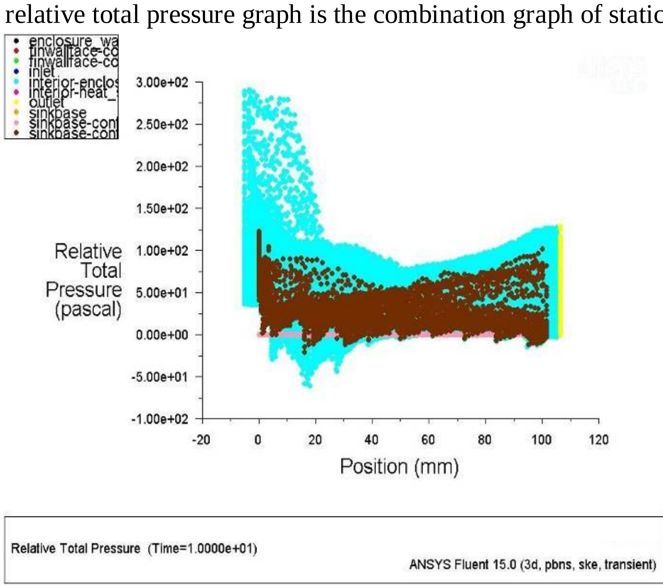


Figure-4.3: Total pressure Vs Position (air velocity of 3 m/s) Figure-4.4: Dynamic pressure Vs Position(air velocity of 3 m/s)

Now the dynamic pressure increases at the inlet to 275 Pascal and gradually decreases the minimum of 70 Pascal at the centre and increases gradually reach a value of 125 Pascal at the outlet(Figure-4.4).

The relative total pressure graph is the combination graph of static, dynamic and total pressure as shown in the figure-4.5



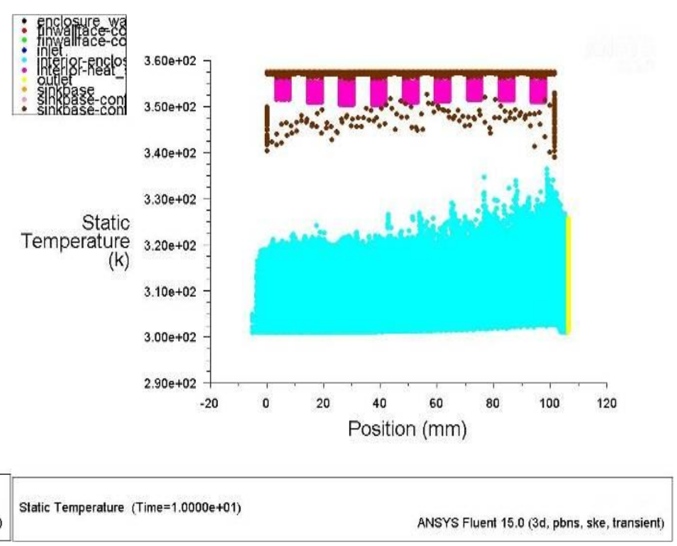


Figure-4.5: Relative total pressure Vs Position Figure-4.6:Static temperature Vs Position(velocity of 3 m/s)

The static temperature varies at each fin from 340K to 350K and we will obtain the maximum air temperature 338K at 100 mm as shown in the below figure-4.6. The total temperature of fins gradually varies from 305K to 350K and the temperature of air ranges from 300K to 380K for the time of 10 seconds and we can observe the scattered minimum temperature dots from inlet to outlet in the below figure-4.7. The turbulent kinetic energy of air 45 m^2/s^2 at 10 mm and decreases to17.5 m^2/s^2 and then sinusoidal wave of increasing and decreasing gradually and reaches to 12.5 m^2/s^2 at the outlet as shown in the below figure4.8

A maximum velocity of 20.72 m/s is obtained at the red coloured vectors and minimum of 0 m/s is obtained at the dark blue coloured vectors in post processing as shown in the below figure 4.9. We can observe the 3D pressure contour of heat sink with enclosure of varying pressure with maximum of 2.836e+002 Pascal and minimum of -7.754e+001 Pascal in the below figure-4.10

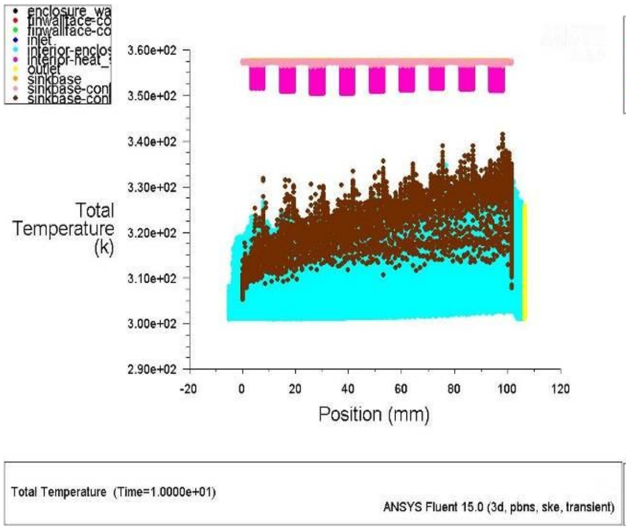


Figure-4.7: Total temperature Vs Position

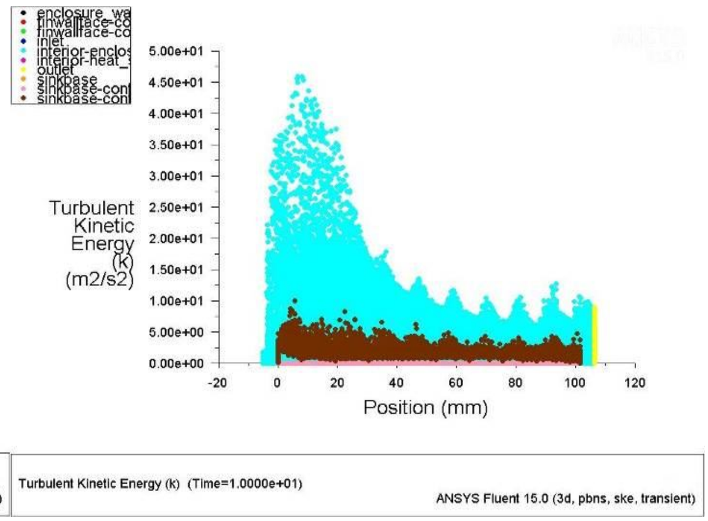


Figure-4.8: Turbulent Kinetic energy Vs Position

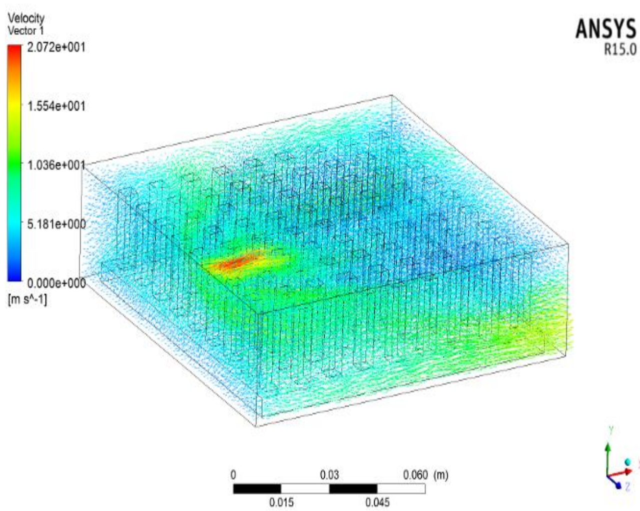


Figure-4.9: Velocity vector (3 m/s)

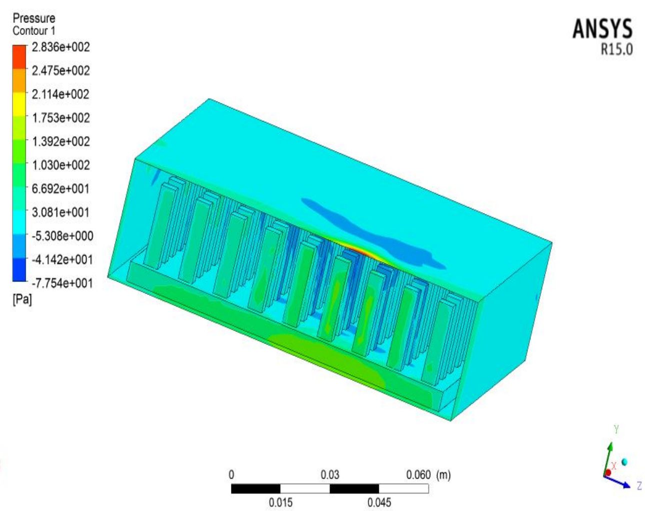


Figure-4.10: Pressure Contour (3 m/s)

The temperature contour of heat sink with enclosure at transient state ranges from 301 K to 357.6 K and the temperature of air on the walls of the enclosure can be clearly seen shown in the below figure-4.11.The temperature range obtained are from the transient state and they change with time and the heat sink with rectangular pin fins are dissipating the heat to the air flowing over it and the maximum heat transfer is occurring to the top surface of the enclosure when compared to the remaining two surfaces of the enclosure.

The turbulence kinetic energy ranges from 2.428e-005 J/kg to 4.592e+001 J/kg and we can observe that the maximum turbulence kinetic energy is occurring in between the third and fourth pin fins from the left in the three rows(Figure-4.12).

Similarly, the results have been obtained for air velocity of 4 m/s. The comparison of temperatures at the tip of the fins at 3 m/s and 4 m/s velocity is shown in the below figure-4.13 and the temperature obtained at the 4 m/s is lower than the temperature obtained at 3 m/s velocity of air.

The comparison of pressure at air velocities of 3 m/s and 4 m/s is shown in the figure-4.14. There is a pressure drop at the second pin fin for 3 m/s and remains constant for the remaining pin fins and for the 4 m/s velocity gradually decreases for the first three pin fins and remains constant throughout the remaining pin fins.

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-4.15.we can clearly observe the turbulence kinetic energy is maximum for 3 m/s and minimum for 4 m/s

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 III, March 2018- Available at www.ijraset.com

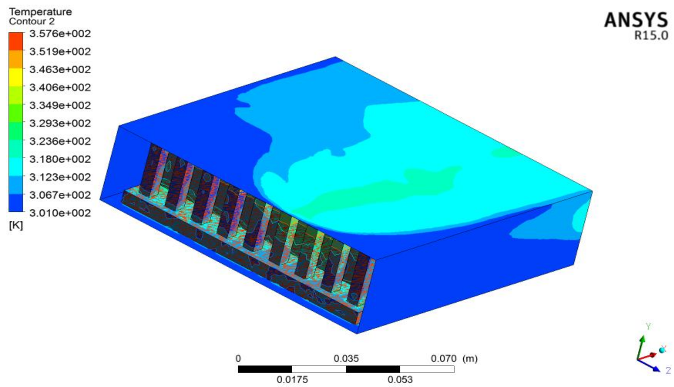


Figure-4.11: Temperature contour (3 m/s)

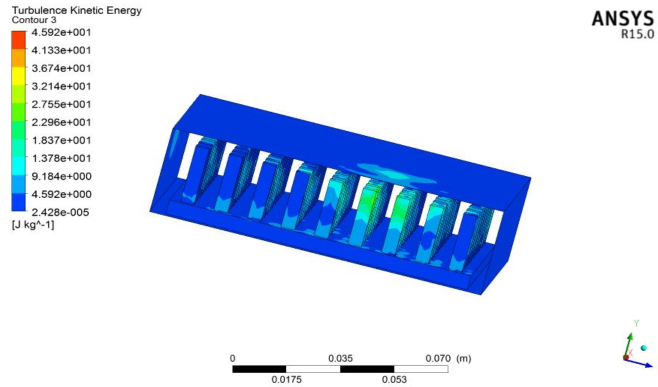


Figure-4.12: Turbulence Kinetic energy(3 m/s)

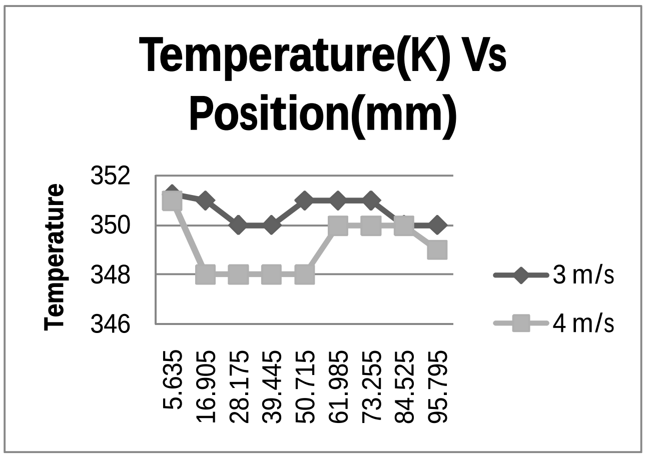
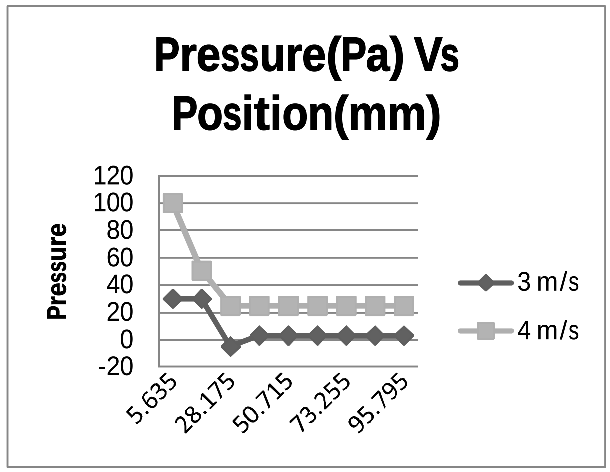
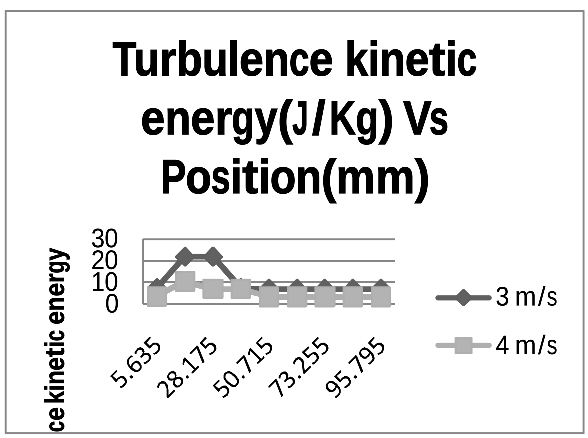


Figure-4.13: Temperature Vs Position



Figur-4.14: Pressure Vs Position



.

Figure-4.15: Turbulence Kinetic energy 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 transient fluid flow analysis. The interaction between the heat sink with rectangular pin fin and its environment has been analyzed by FEM. The results obtained for heat sink with rectangular pin fin have been established by comparing the FEM and analytical techniques.

REFERENCES

[1] Mohammad Saraireh, Computational Fluid Dynamics Simulation of Plate Fin and Circular Pin Fin Heat Sinks, Jordan Journal of Mechanical and Industrial Engineering Volume 10 Number 2, June.2016.

[2] S. Jeevaraj, Heat Transfer Analysis of Heat Sink by Computational Fluid Dynamics (CFD), International Journal on Emerging Technologies (Special Issue on NCRIET-2015), 6(2), 2015.

[3] AgnihothraSarma O, A Ramakrishna, CFD Analysis of Splayed Pin Fin Heat Sink for Electronic Cooling, International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 10, December- 2012

[4] 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.

[5] 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.

[6] 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

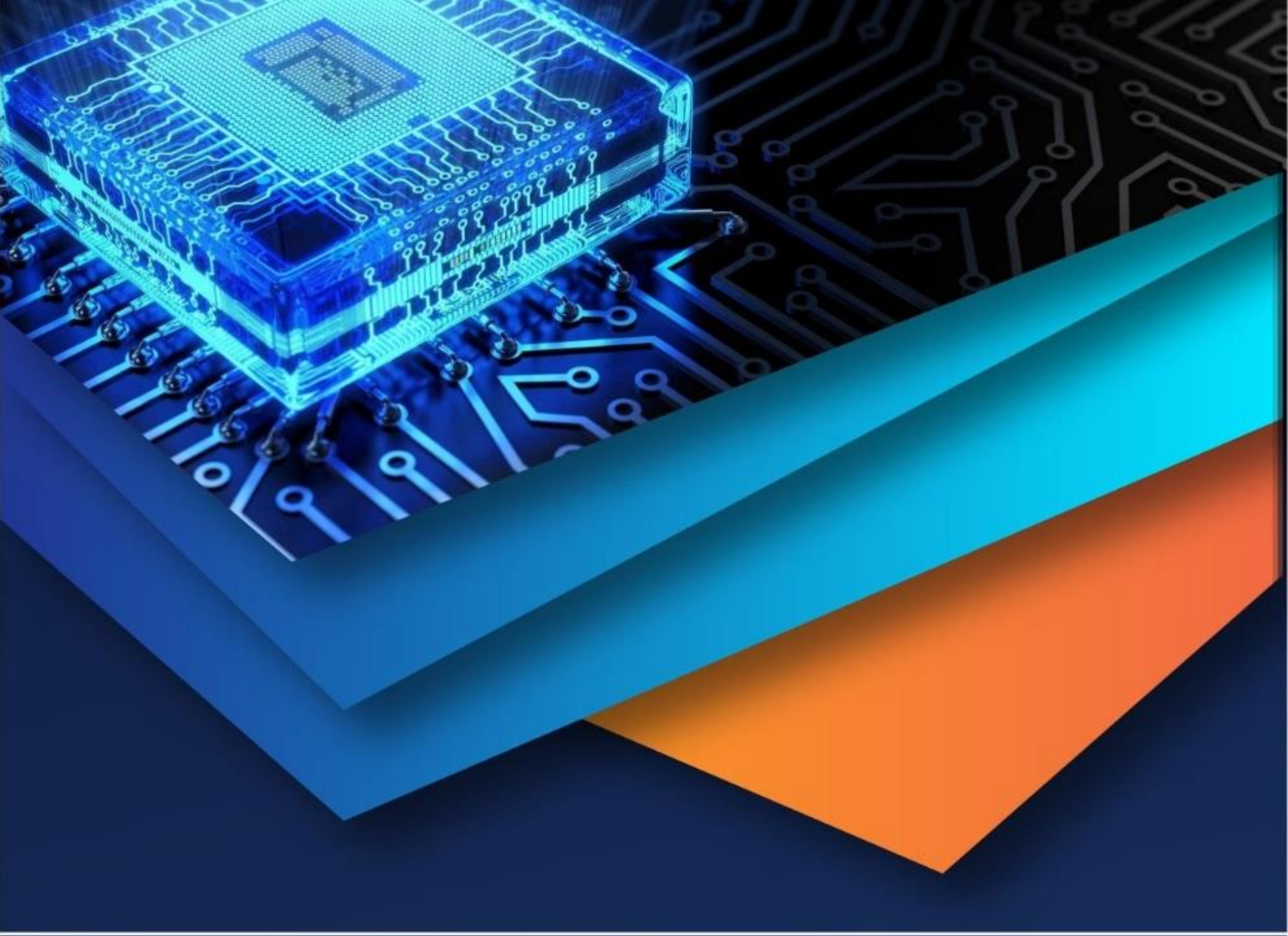
[7] T.Therisa,B.Srinivas and A.Ramakrishna, Analysis of Hyoid Structured And Perforated Pinfin Heat Sink In Inline And Staggered Flow, International Journal of Science Engineering and Advance Technology,IJSEAT, Vol. 2, Issue 4, April-2014

[8] R. Hagote, S. K. Dahake, Enhancement Of Natural Convection Heat Transfer Coefficient By Using V- Fin Array, International Journal Of Engineering Research And General Science, Vol-3, Issue-2, April 2015

[9] 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.

[10] Park, Park-Kyoun Oh, Hyo-Jae Lim, The application of the CFD and Kriging method to an optimization of heat sink, International Journal of Heat and Mass Transfer 49 3439–3447, 2006.

[11] Yue-Tzu Yang, Huan-Sen Peng, Numerical study of pin-fin heat sink with un-uniform fin height design, Int. J. Heat Mass Transfer, 2008, 51 (19-20), pp. 4788-4796.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY