

Fluid flow and heat transfer investigation of perforated heat sink under mixed convection



^{#1}Mr. Shardul R Kulkarni, ^{#2} Prof.S.Y.Bhosale ,

¹shardul.kulkarni794@gmail.com

^{#1} Research scholar

Department of mechanical engineering
PES's MCOE Shivaji nagar pune -411005

^{#2}Asst prof

Department of mechanical engineering
PES's MCOE Shivaji nagar pune -411005

ABSTRACT

The present paper represents experimental investigation of heat transfer from plain heat sink and modelling, simulation in CFD to investigate the fluid flow and heat transfer characteristics of a fin arrays with lateral circular perforation under mixed convection mode. The simulation is carried out using the fluid flow ICEM- CFD of ANSYS V14.0. In this study, results shows that formation of the stagnant layer around the solid fin array which slow-downs the heat dissipation rate. The increase in the fluid flow movement around the fin results increase in the heat dissipation rate. It can be achieved by creating perforation to the fins. The analysis is carried out with CFD model to investigate flow pattern, temperature variations in computational domain, heat transfer coefficients & Nusselt no .Analysis is carried with both plane and perforated fin arrays with different size of CFD domains in order to ensure results with different fin spacing. A constant heat flux is assumed for heat sink

Keywords— Heat transfer, Heat sink, Mixed convection.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

In today's modern world heat transfer from the electronic equipments is one of the concerns. As cooling of such electronic equipments involves use of fins and mostly air as a cooling medium. In active method of cooling where forced convection is used where as in passive method natural convection with parallel plate channels with various fin configurations is used. Primary objective of designing electronic cooling systems is to achieve high heat transfer coefficients and compactness. Compared to bare plate, a finned surface increases the heat transfer area but fluid flow rate is reduced so only with proper design overall

improvement in heat transfer is achieved. Many experimental and numerical studies cover natural as well as forced convection but compared to them mixed convection studies are very less. Sane S.S. et al. [2], Suyawanshi S.D. et al. [1] investigated the performance of notched, inverted notched horizontal rectangular fin arrays under natural convection. Pawar A.L. et al. [6] examined that for the perforated horizontal fin array under natural convection the values of average heat transfer coefficient are higher than plain fins. Dogan M et al. [4] investigated the mixed convection heat transfer in the natural convection dominated mode depends on the fin height, spacing and initially with increase in fin spacing the average convection heat transfer

coefficient increases and takes maximum value after which to decrease.

H N deshpande [3] in his investigation of perforated rectangular fin arrays marked changes in heat transfer coefficients with fin spacing under natural convection. calculated optimum fin spacing and shown that performance of fins can be improved by perforations at the centre in triangular pattern. Taji.S.G. et al. [7-8] experimentally and numerically investigated the horizontal rectangular fin array by varying fin spacing and keeping length, height same for different heater inputs and different flow velocities in assisting mode of mixed convection for performance enhancement.

II. MIXED CONVECTION

The buoyancy forces that arise as a result of temperature difference and cause the fluid flow in natural convection. In forced convection this buoyancy forces are negligible. But in some cases this forces are effective in fluid flow and heat transfer in such cases the flow is a combination of forced and natural convection these flows are referred as mixed convection. In these cases neither natural nor forced convection is dominant and both effects are of same magnitude. Flows associated with low forced velocities can be categorised as mixed convection.

The ratio of Gr/Re^2 known as a Richardson number (Ri) gives a qualitative indication of influence of buoyancy on forced convection.

If $Gr/Re^2 > 10$ natural convection is dominant where as if $Gr/Re^2 < 0.1$ forced convection is dominant. Value of Ri ranging 0.1 to 10 is a case of mixed convection. $Ri < 1$ shows the dominating forced mixed convection.

In assisting mode natural convection can assist forced convection and enhances heat transfer. This concept can be utilised to enhance heat transfer for electronic equipment cooling. Mixed convection is significant in heat exchangers, solar driers, cooling equipments with low velocities. They also occur in atmosphere and ocean.

III. NUMERICAL STUDIES

As per available studies until now the heat transfer and fluid flow from perforated horizontal rectangular fins under mixed convection flow velocity is not investigated numerically yet. Therefore the numerical investigations are carry out on horizontal rectangular fins with perforations in triangular pattern under mixed convection by keeping the fin length and height constant. And compared it with experimental results of imperforated rectangular fins. To study the pattern of heat transfer and fluid flow characteristics.

A. Use of codes

Every numerical algorithm has its own characteristics of results with inherent error patterns. At the end of a simulation the user must make judgment whether the results are 'good enough'. It is to assess the validity of models of physics embedded in a program. Anyone wishing to use CFD in a serious way must realize that it is not substitute for experiment, but a powerful additional problem-solving tool. In the present investigation commercially available well-established and proven CFD solver ANSYS FLUENT is used. The natural convection flow under investigation is modelled by set elliptic partial

differential equations describing the conservation of mass, momentum and energy in three rectangular Cartesian coordinates. The numerical model is based on a control volume-finite difference formulation. The equations are integrated over each control volume to obtain a set of discretized linear algebraic equations in the format given above are called finite volume equations. Finite volume equations describe processes affecting the value of ϕ in cell P in relation to its neighbour cells together with the source term. These equations are solved by the widely used

B. Numerical modelling.

. The numerical model is based on control volume finite difference formulation. The set of equations are integrated over each control volume to obtain discretized linear algebraic equations, termed as finite volume equations. Infinite number of fins, with negligible thickness is assumed. The fin surfaces, with base are assumed as a source, held at uniform temperature. Laminar natural convection is the mechanism for heat transfer from the fin array. Radiation heat loss is not considered. Schematic drawing of fin array under investigation is shown in, with actual simulated part of one fin. As shown in, computational domain is reduced to one fin channel. . The imposed boundary conditions are as shown in The fin surface and base are held at a constant temperature T_s All the remaining boundaries are assigned as pressure outlet where air enters or leaves the channel at the ambient temperature T_∞ and corresponding density ρ_0 . Here the ambient pressure is used as stagnation boundary condition with the incoming mass having the ambient temperature. The static pressure is assumed equal to the pressure of surrounding atmosphere. Molecular (laminar) transport boundary conditions are applied to the fin walls to simulate the effects of laminar friction and heat transfer, known as no-slip boundary conditions. Flow under investigation was modeled by a set of partial differential equations describing the conservation of mass, momentum and energy in three dimensional Cartesian coordinate system

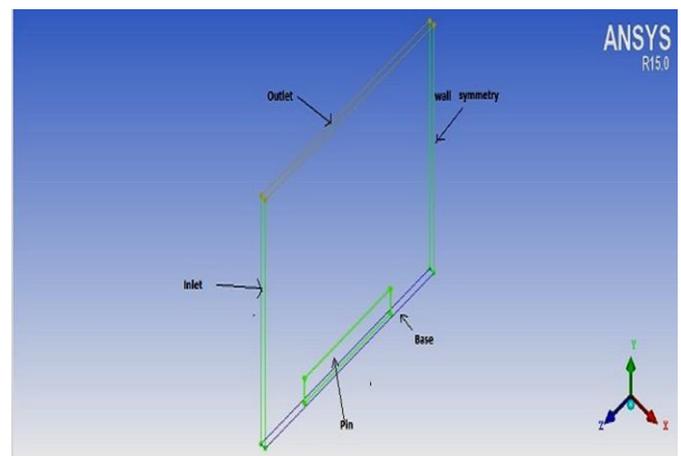


Fig.1 .geometry created in ANSYS -ICEM

As shown in fig 1 a single fin is considered on a wall having dimensions $200 \times 40 \times 2$ as numerous experimentation is carried out[] with same size of fin as short fin arrays ($L/H=5$). The assumptions employed in the governing equations are in agreement with steady, incompressible, laminar flow of air. The properties of air, viz. density,

thermal conductivity, viscosity are taken as a function of temperature. Radiation heat transfer is not modeled. The reference density ρ_0 was calculated from the inlet temperature. The numerical model is based on a control volume formulation. The above equations are integrated over each control volume to obtain a set of discretized linear algebraic equations of the form:

$$a_p \phi_p = \sum a_{nb} \phi_{nb} + b$$

Finite volume equations describe processes affecting the value of ϕ in cell P in relation to its neighbor cells together with the source term b. These equations are solved by the ANSYS FLUENT solver, employing the SIMPLE algorithm for the pressure correction process along with the solution procedure for the hydrodynamic equations. Second order up wind schemes are used for interpolating velocity and temperature fields.

C. Mesh generation and boundary conditions.

A domain has to be built around the fin channel to study mass flow and thus the heat flow from the fin, because the area of interest is the outside of fin, which is the interface between the air and fin surface. Thus, the domain consists of air. Domain height is selected as 10 times the maximum fin height, by ensuring that the temperature changes at a height around 10 times the fin height are negligible. The conditions at this height can be taken as atmospheric conditions. The schematic of the channel modelled initially, along with the co-ordinate system the geometric model created in ICEM CFD 15.0 a thickness of domain is dependent on fin spacing (s) a symmetry condition is used for walls to minimise domain size. It is unstructured mesh. The mesh points are selected in such a way that they are placed closer to the boundaries where higher gradients of temperature and velocity are observed i.e. the fin flat and fin base surfaces. Hence a grading is provided to the edge meshes. This was done to reduce the mesh size and yet have a fine mesh around the critical areas. The boundary layer thickness at the mean fin height was calculated at program run time and mesh points were decided to cover the boundary layer sufficiently by having at least ten mesh points inside that thickness.

All boundary conditions get implemented by the inclusion of additional source and/or sink terms in the finite volume formulation for computational cells at the boundaries. In natural convection flows there is no information regarding the velocity and temperature fields before the start of calculations. Since governing equations are invariably coupled, the temperature field causes the velocity field to develop and which in turn affects the temperature field. The base plate (wall) is assigned certain value of heat flux; This value of heat flux is decided according to the heater input. Starting with the assumption of isothermal fin surface, the methodology and solution scheme using periodic boundary condition at the plane perpendicular the base plate, and parallel to the fin flat while accounting for the variation in air properties is presented in The histogram of obtaining the best match between experimental and analytical value of average surface heat transfer coefficient is shown in.

D. Results and discussion.

As disused above CFD models with varying heater inputs as well as varying inlet air velocities are created .for mixed

convection air velocities from 0.75 m/s,0.85 m/s,0.95 m/s, 1.05 m/s,1.15 m/s are used to obtain different cases for heater inputs 25w,50w,75w & 100W. It has been insured that these velocities gives Ri no within the range of mixed convection.

These CFD models are formed for both plane as well as perforated fin as it has been suggested by [3].As the long horizontal fins are used (L/H = 5) consequently air might not reach at the central section of the fins so ineffective central surface of fins which is removed by creating the triangular pattern perforations.

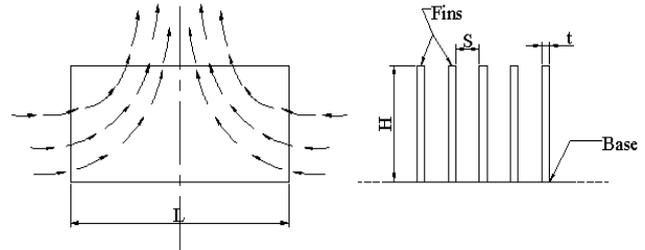


Fig.4 single chimney flow pattern

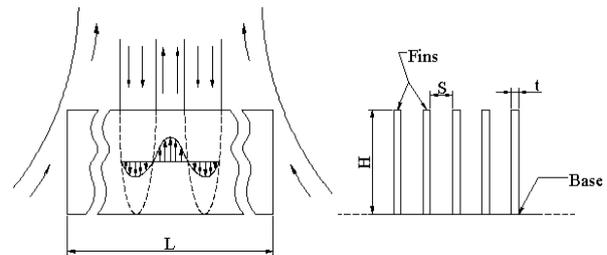


Fig.5 sliding chimney flow pattern

As shown in figures is preferred by design point view .whereas sliding chimney flow pattern is not preferred. Sliding chimney flow pattern is present where there is low spacing and effect of increased area is not results in improved heat transfer rates due to sliding chimney so assisted flow is used to translate sliding chimney and give better performance.

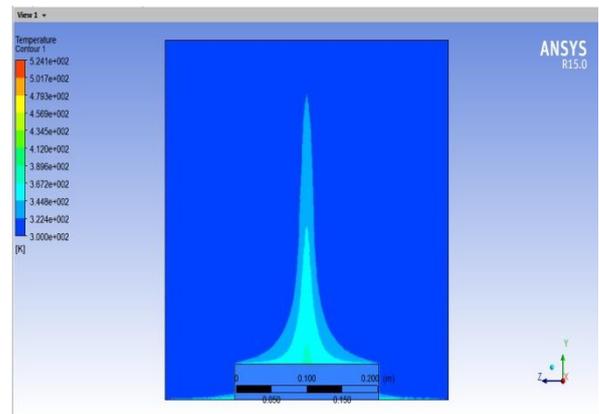


Fig.1 temperature contour of plane fin array

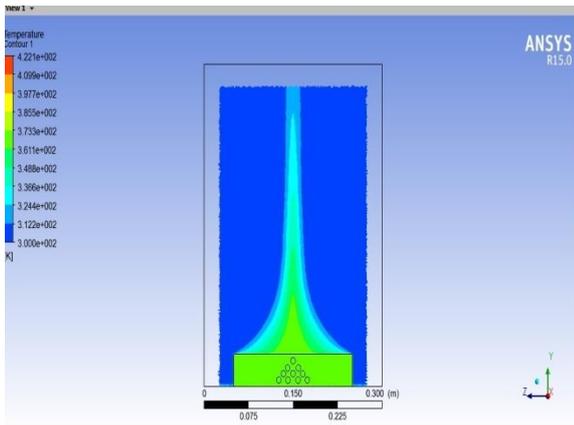


Fig.6 temperature contour of perforated fin array

The temperature Contours on plane y-z plane are shown in Figure 2 and 3 for spacing $S = 10$ mm. The Single chimney flow patterns are observed for higher fin spacing. Whereas the choked type flow patterns are seen for lower fin spacing. From temperature counters, it is observed that the central portion of fin array almost remains hot this can be solved by providing perforations where the area is removed and heat transfer rate is improved.

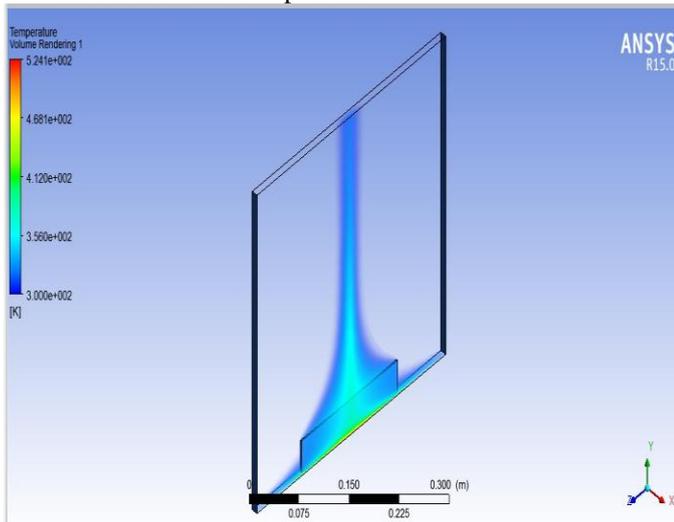


Fig.7 temperature contour of perforated fin array

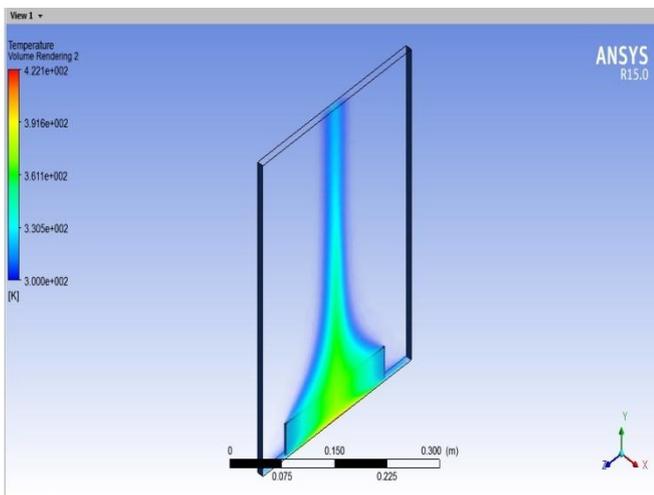


Fig.8 volume rendering of temperature perforated fin array(s=8)

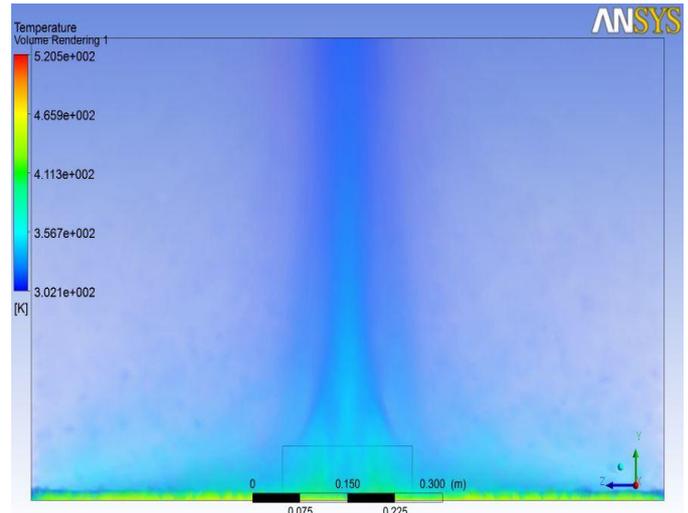


Fig.9 volume rendering of temperature perforated fin array(s=2)

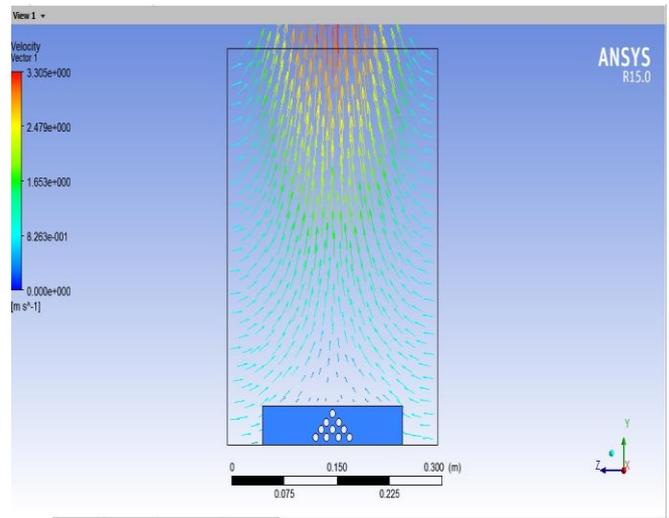


Fig.10 velocity vector plot for perforated fin (s=8)

IV. CONCLUSION

It can be seen from above CFD cases that heat transfer rate for a long fin ($L/H=5$) can be improved by perforation at middle section, a single chimney flow pattern is observed in assisted mode of convection even in CFD analysis where base has been given uniform heat flux. whereas for low fin spacing ($s=2,4$) sliding chimney flow pattern exist.

ACKNOWLEDGMENT

The author would like acknowledge to Prof H.N. Deshpande for his valuable contribution and Dept of Mechanical engineering of PES's "Modern college of engineering."

REFERENCES

- [1] S .D. Suryawanshi and N.K. Sane, "Natural convection heat transfer from horizontal rectangular inverted notch fin arrays", ASME J. of Heat Transfer 131, 2009, pp. 1-6
- [2] .S. Sane , G.V. Paishwad and N.K. Sane, "Natural convention heat transfer enhancement in horizontal rectangular fin array with inverted notch",

- 18th National and 7th ISHMT-ASME Heat and mass Transfer Conference at IIT, Guwahati, 2006.
- [3] H. N. Deshpande , Prof. S. G. Taji Experimental Study of Heat Transfer from Horizontal Rectangular Fins with Perforations under Natural Convection International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Vol. 3 Issue 2, February - 2014
- [4] M. Dogan and M. Sivrioglu, “Experimental investigation of mixed convection heat transfer from longitudinal fins in a horizontal rectangular channel”, Int J of Heat and Mass Transfer 53, 2010, pp.2149–2158.
- [5] S.G. Taji, G.V. Paishwad, N.K. Sane Enhanced performance of horizontal rectangular fin array heat sink using assisting mode of mixed convection. International Journal of Heat and Mass Transfer 72 (2014) 250–259
- [6] A.L. Pawar, S.G. Taji, and N.K. Sane, “An experimental investigation on perforated horizontal rectangular fin array under natural convection .
- [7] S.G. Taji, G.V. Paishwad, N.K. Sane and R.Z. Deshmukh, “Experimental investigation of heated horizontal rectangular fin array under mixed convection”, Int. Review of Mechanical Engineering (IREME), Praise Worthy Prize, Italy 6, 2012, pp.104-112
- [8] S.G.Taji, “Experimental and Numerical Study of Heated Horizontal Rectangular Fin Arrays under Mixed Convection”, Ph.D Thesis, University of Pune, India , 2014.