© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

CFD ANALYSIS OF PERFORATED FINS

Neeraj Yadav[1], Rajneesh Kumar Gedam [2]
P.G. Student, Dept. of Mechanical Engineering, R K.D.F. College of Technology, Bhopal, M.P., India[1]
Associate Professor, Dept. of Mechanical Engineering, , R K.D.F. College of Technology, Bhopal, M.P., India[2]

ABSTRACT: Recently the need for high performance and compact size electronic device is rising exponentially. With the reduction in size and increase in the number of task performed by these devices the need for better and compact heat removal device is also increasing. Various researchers are researching in the field to provide an efficient heat removal device which has high efficiency and is compact in size. Electrical and mechanical goes in series with each other. In this study the objective was to analyze Heat sink fin with different perforation shapes. Heat sinks are used as a cooling device in various applications and fields.

In this study various heat sink model were designed with varying perforation shapes in order to increase contact area. As it is a proven fact larger the contact area better will be the cooling. This was kept in mind while designing the hollow perforated fins for the analysis..

Keywords: Perforation, Fins, Heat Sink, Temperature

1. INTRODUCTION

With the increase in high performance and compact size electronic device the need for better and compact heat removal device is also increasing. Researches to provide an efficient heat removal device which has high efficiency and is compact in size are being done by researchers. In this study 3D models of pin fin heat sink with various perforation shape were studied for heat transfer rate.

2. HEAT SINK

A heat sink (also commonly spelled heatsink) is a passive heat exchanger that transfers the heat generated by mechanical or electronic device to a fluid medium, generally air or a liquid coolant, where it is dissipated away from the device, thereby allowing regulation the temperature of device at optimal levels. In order to cool central processing units or graphics processors in computers, heat sinks are used. Heat sinks are used with high-power semiconductor devices like power transistors and optoelectronics like lasers and LEDs (light emitting diodes), where the heat dissipation ability of the component itself is insufficient to moderate its temperature. . Copper and/or aluminum are primarily used for manufacturing heat sink. Copper is used since it possess many desirable properties for thermally efficient and durable heat exchangers. First and foremost, copper possess excellent heat conduciveness. This means that due to high thermal conductivity of copper, it allows heat to pass through quickly. Aluminum is used in applications where weight is a big concern.

3. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is the application of mathematics, physics and computational software to visualize how a gas or liquid flows -- as well as how the gas or liquid affects objects as they flow through them. Navier-Stokes equations are the base of Computational fluid dynamics. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. The steady improvement in computing power, since the 1950's, played a role in the development of CFD. This branch of fluid dynamics, supplements experimental and theoretical fluid dynamics by providing alternative and potentially cheaper means of testing fluid flow systems. It also allows testing of the conditions that are not possible or extremely difficult to measure experimentally and are not amenable to analytic solutions

4. LITERATURE REVIEW

Non-uniform Temperature Distribution in Electronic Devices Cooled by Flow in Parallel MicroChannels, was discussed by Hetsroni et al[1]. Two-Phase Flow Patterns in Parallel Microchannels was studied by Hetsroni et al. They analyzed the effect of geometry on flow and heat transfer, finding that an increasingly uniform heat flux resulted in an increased irregularity of temperature distribution on the chip surface.

Bau[2] conducted an optimization study to minimize the temperature gradient and the maximum temperature for microchannel heat sink. It was demonstrated that further reduction in maximum temperature and temperature gradient could be achieved by varying the cross-sectional dimensions of the microchannel. The penalty of this method is the dramatically increased pressure drop due to the acceleration along the flow direction.

Culham et al. [3] presented an analytical approach for characterizing electronic packages, based on the steady-state solution of the Laplace equation for general rectangular geometries, where boundary conditions are uniformly specified over specific regions of the package. The basis of the solution is a general three dimensional Fourier series solution which satisfies the conduction equation within each layer of the package. The application of boundary conditions at the fluid solid, package board and layer-layer interfaces provides a means for obtaining a unique analytical solution for complex IC packages. They compared the values with published experimental data for both a plastic quad flat package and a multichip module to demonstrate that an analytical approach can offer an accurate and efficient solution procedure for the thermal characterization of electronic packages.

Davies et al.[4] presented the method to correct the thermal resistance of electronics components is to adjust the junction-to-ambient thermal resistance to account for operational conditions. For forced convection applications, they proposed two factors; the first accounts for any upstream aerodynamic disturbance and the second addresses purely thermal interaction. Thus if an upstream powered component interacts with a downstream component, the two factors are combined. They found that both factors may be quantified in terms of readily measured temperatures and then used as coefficients to adjust the standard thermal resistance data for operational conditions. They applied this approach to a symmetrical array of board mounted 160- lead devices and the data shows how the factors vary with component position, non-dimensional power distribution and Reynolds number. Based on data they proposed a new method of generating operational component thermal resistances.

Masud et al.[5] tried to validate the CFD package FLUENT with the experimental data obtained by then earlier. Here they have taken a heated chip with temperature 353 K and the air inlet velocity at temperature 293 K. The inlet velocities were varied from 1 m/s to 7 m/s. Various turbulence models have been tested, and the effect of the channel inlet flow on the heat transfer rate has been determined by considering both a uniform and fully-developed condition. The substrate adiabatic heat transfer rate has been determined by considering both uniform and fully developed condition. The substrate adiabatic heat transfer coefficient is also numerically determined. The results indicate that the flow in the vicinity of the module is three dimensional, and exhibits flow separation and vortex formation, hence leading to a complex distribution of the local heat transfer coefficient on the substrate. In general the flow structure was in good agreement with the experiments. The predicted turbulence intensity did not agree well with the measurements. The turbulence treatment near the wall is very important and wall functions are not suitable.

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

Dhiman et al. [6] investigated the flow and heat transfer characteristics of an isolated square cylinder in cross-flow placed symmetrically in a planar slit for a range of conditions. They obtained 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. In addition, variation of the local Nusselt number on each face of the obstacle and representative isotherm plots are presented to elucidate the role of Prandtl number and blockage ratio on drag coefficient and heat transfer.

Cheng et al. [7]numerically investigated the fluid flow and heat transfer characteristics of mixed convection in three-dimensional rectangular channel with four heat sources. The SIMPLEC algorithm was applied to deal with the coupling between pressure and velocity, and new high-order stability guaranteed second-order difference scheme was adopted to discretize the convection term. They studied the influence of four parameters: Richardson number, heat source distribution, channel height and inclination angle. They analyzed the numerical results from the viewpoint of the field synergy principle, which says that the enhanced convective heat transfer is related not only to the velocity field and temperature field, but also to the synergy between them. They found that the effects of the four parameters on the thermal performance can all be explained with the field synergy principle. To obtain better electronic cooling, the synergy between the velocity and temperature gradient should be increased when other conditions are unchanged.

Kumara et al. [8]investigated the complex unsteady flow through and around a channel in the presence of an obstruction at the entry is studied by solving directly the unsteady Navier-Stokes equations. They considered the Reynolds number of 4000, as experimental results is available for comparison. The computed results are in close agreement with experiments. The computations help with better understanding of the phenomenon of reverse flow and fluid pumping.

Tuckerman and Pease[9] used silicon microchannels, with water as the working fluid, to dissipate power from an electronic chip. The microchannels were etched in a silicon sample with an overall dimension of 1cm . They had a channel width of approximately 60 μm and a parameterized channel height varying between 287 μm and 376 μm. These microchannels effectively dissipated heat up to 790 W/cm2 while maintaining a chip temperature below 110 °C.

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

Wong and Peck [10]evaluated experimentally the effect of altitude on electronic cooling. As material properties of air vary as a function of altitude due to changes in atmospheric pressure and temperature, these changes have a negative impact on the heat transfer effectiveness and result in higher component temperature when compared to sea level conditions. They carried out the experiments in a hypobaric chamber using electronic printed circuit boards populated with heated rectangular blocks placed in a small wind tunnel. The altitude was varied between 0 and 5000 m above sea level and the air speed is varied between 1 and 5 m/s. The results show the local adiabatic heat transfer coefficient and thermal wake function diminish with altitude. This information is useful for design and analysis of electronic equipment for operation over a range of altitudes and air speeds typically encountered in forced air convection cooling applications.

Liu and Garimella[11] showed that conventional correlations offer reliable predictions for laminar flow characteristics in microchannels over a hydraulic diameter range of 240 to 974µm.

Harms et al.[12] performed experiments on an array of microchannels and determined that local Nusselt numbers can be accurately predicted in microchannels by conventional correlations. They also determined that proper plenum design and consideration are necessary to be able to apply the theoretical Nusselt number and friction factor equations to microchannel experiments.

METHODOLOGY

1. <u>Designing of the model:</u> for this study one Heat sink model is selected and accurate dimensions were measured for designing. Measurement of the dimension is done by using scale and vernier caliper. After measuring the dimensions a 3D model was prepared in CATIA V5. Two models were prepared with different perforation shapes were designed for this study as mentioned below.

1. Heat sink Model with hollow fins and Rectangular perforation.
2. Heat sink Model with hollow fins and Oval perforation

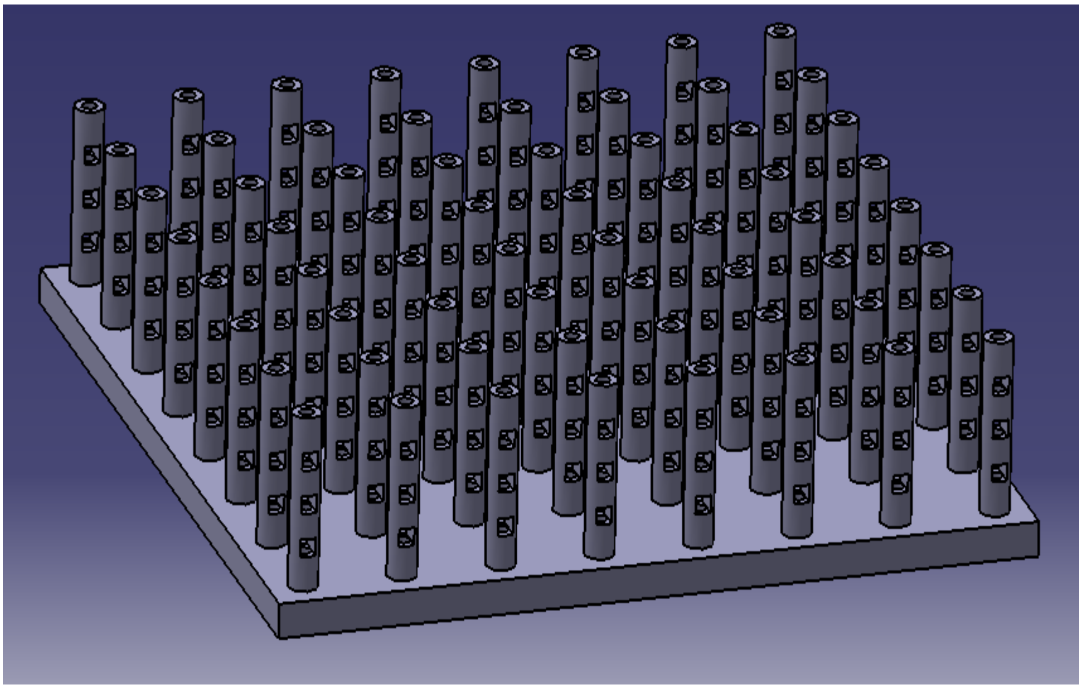


Fig: 1 Heat sink Model with hollow fins and Rectangular perforation.

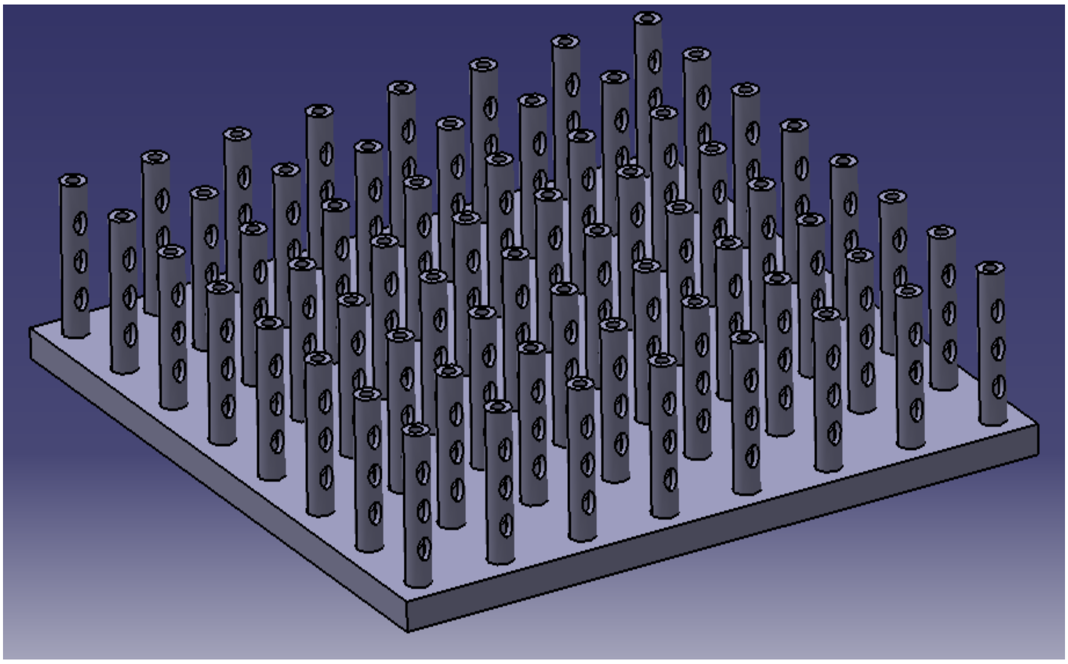


Fig: 2 Heat sink Model with hollow fins and Oval perforation.

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

Computational Fluid Dynamics: 3D CAD model created in CATIA V5 is then converted into stp file to be imported in ANSYS for the further calculation and analysis part on it. Once the model is imported an enclosure was created in workbench geometry for fluid medium in our case air. Inlet and outlet are provided in enclosure. The model then divided into number of mesh or definite numbers of nodes and elements

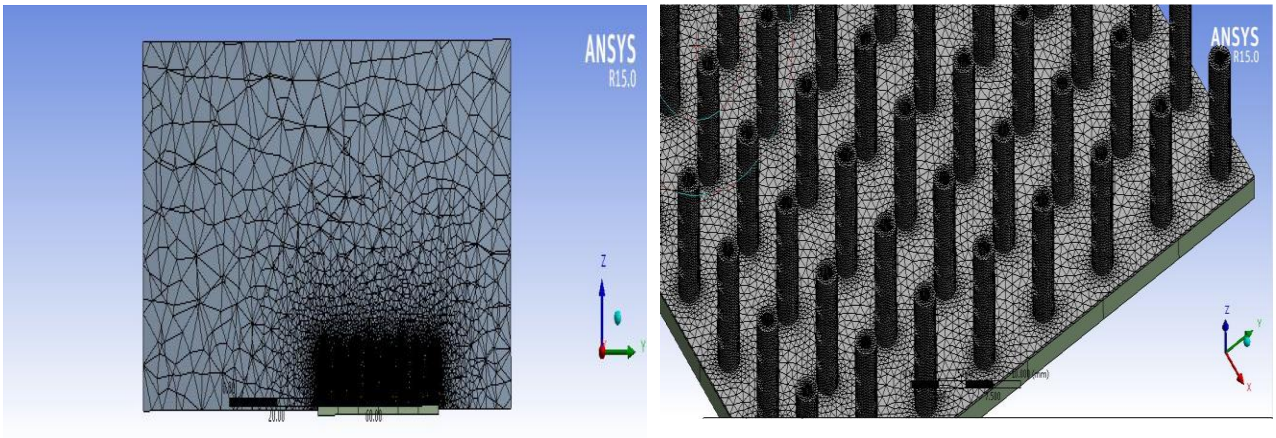


Fig: 3 Meshing of fins inside the enclosure

3. **Fluent input parameters:** certain parameters were needed to be set before the solution was initialized. Those are given below

1 Assigning Materials and Energy Equation- Aluminum was assigned as the material for model as it is lightweight and has high heat transfer rate and heat dissipation. Air was chosen as the fluid medium to transfer heat from fin to atmosphere.

Governing equation for three dimensional steady state incompressible fluid flow was *k-ε* epsilon equation.

2 Boundary Conditions – the computational problem is reduced in size by exploiting the symmetry of PHS to apply symmetry boundary condition along the sides of the channel. The fluid and thermal condition taken were

- At inlet
  Velocity > 10m/s, temperature = 282k
- At bottom base of the heat sink
  Q= 20000 W/m^2

© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

After defining the boundary conditions solution is run

5. RESULTS AND DISCUSSION

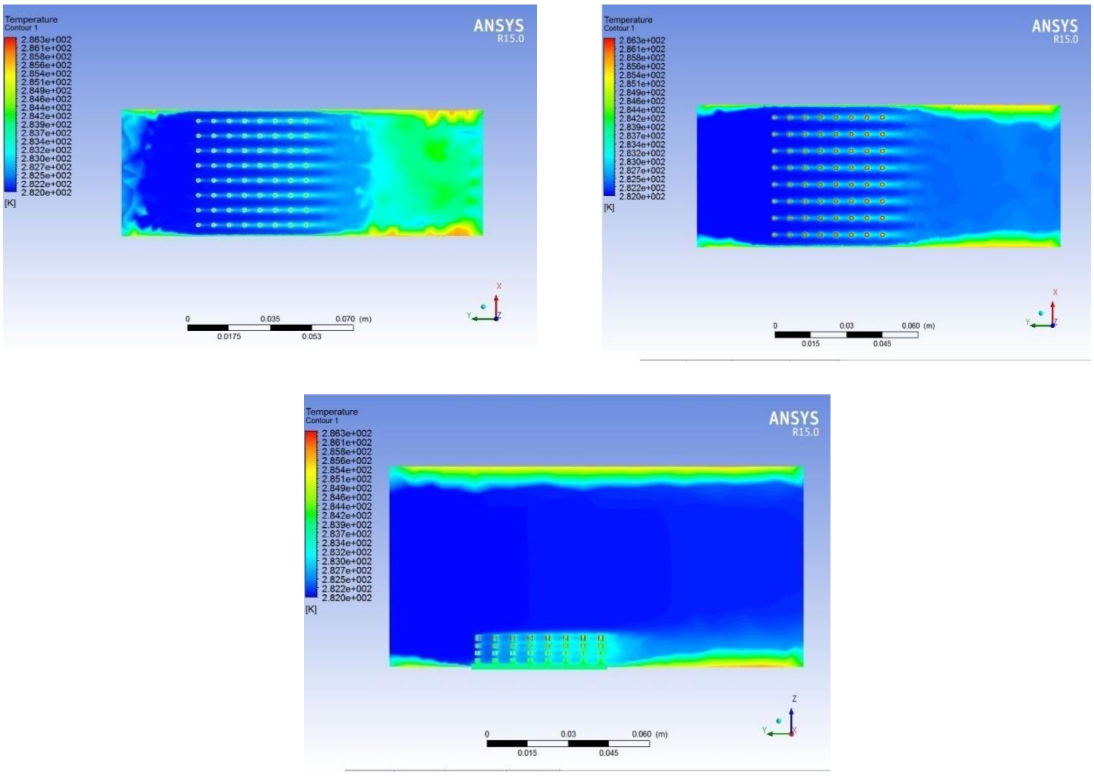
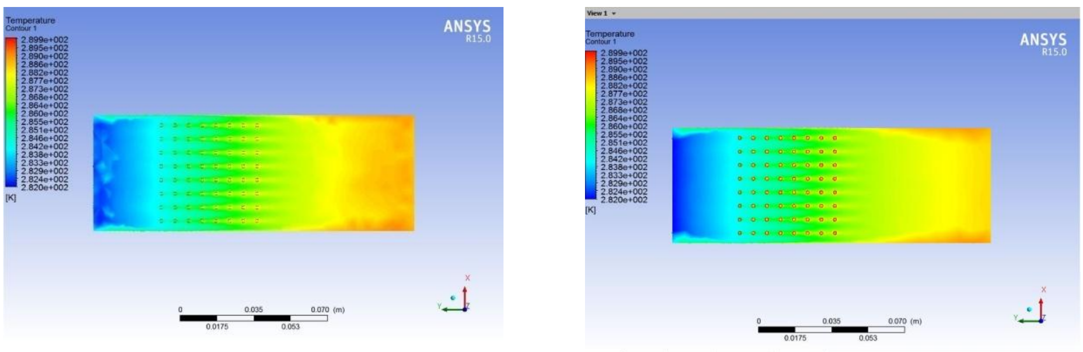


Fig:4 Temperature contour on HEAT SINK of hollow fins with Rectangular perforation



© INTERNATIONAL JOURNAL FOR RESEARCH PUBLICATION & SEMINAR
ISSN: 2278-6848 | Volume: 08 Issue: 01 | January - March 2017
Paper is available at www.jrps.in | Email : info@jrps.in

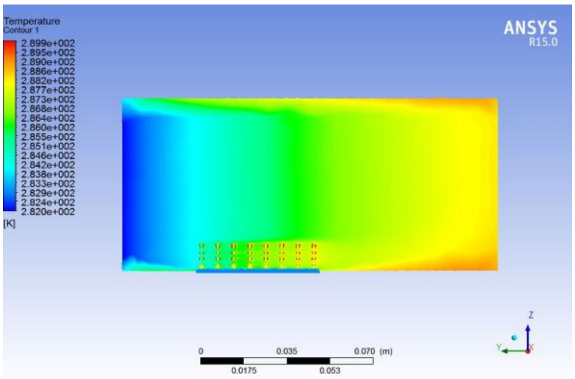


Fig:5 Temperature contour on HEAT SINK of hollow fins with Oval perforation

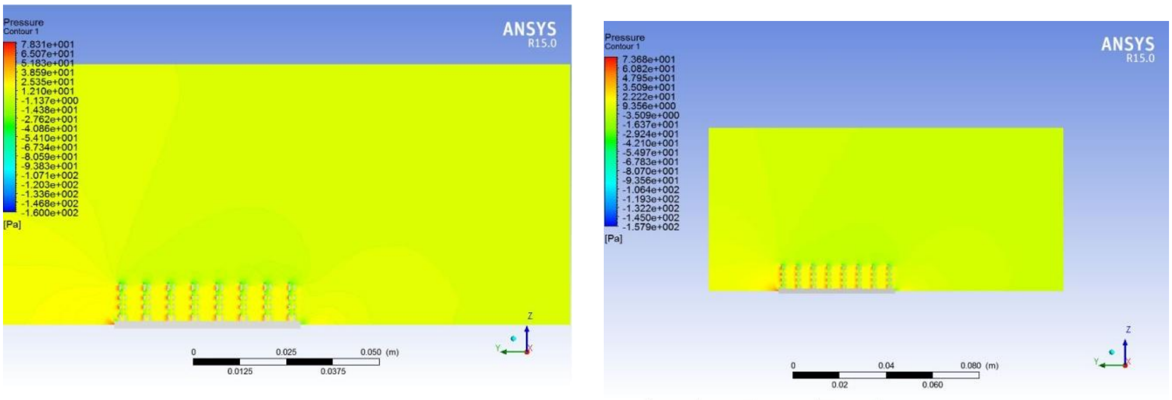


Fig:6 Pressure contour on HEAT SINK of hollow fins

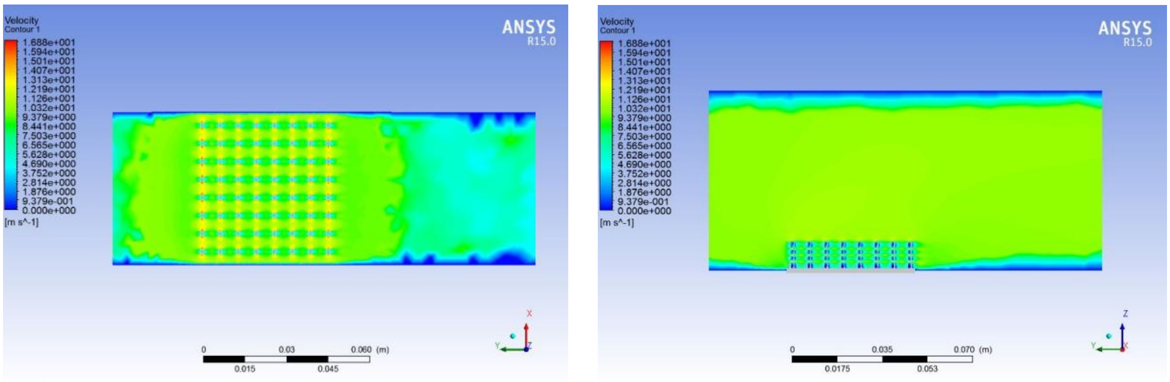


Fig:7 velocity contour on HEAT SINK of hollow fins with Rectangular perforation

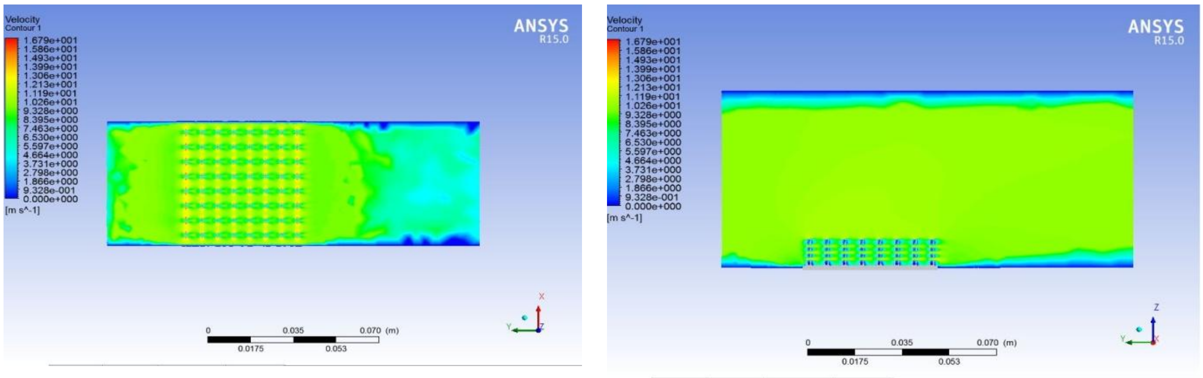


Fig:8 velocity contour on HEAT SINK of hollow fins with Rectangular perforation

From the study done above temperature at the outlet of fins were found to be 297K, 286K and 283K for circular, oval and rectangular fins respectively.

6. CONCLUSION

In the current study CFD analysis was performed on various perforation shapes on fins it is deduced from that graphs plotted in the result section that hollow fins with rectangular perforation is best suitable for validation.

Temperature result obtained were the main factor taken into consideration in this study. As temperature is the main parameter that affects the efficiency of the heat sink. In rectangular perforation result obtained for temperature is 283K where as in oval perforation it was observed to be 286K.

As pressure was kept constant in this study main results obtained are based on temperature condition. Although the rectangular perforation showed best results on validation but the production of such fins requires a very complex process. Latest techniques and further studies can provide future scope a easy process for producing such fins.

.

REFERENCES

1. Hetsroni, G., Mosyak, A. and Segal, Z., (2001), “Non-uniform temperature distribution in electronic devices cooled by flow in parallel microchannels”, IEEE Transactions on Components and Packaging Technologies, Vol. 24, pp. 16-23.
2. Bau, H.H., (1998), “Optimization of conduits’ shape in micro heat exchangers”, International Journal of Heat and Mass Transfer, Vol. 41, pp. 2717-2723.

3. Culham, J.R., Yovanovich, M.M. and Lemczyk, T.F., (2000), "Thermal characterization of electronic packages using a three-dimensional fourier series solution", ASME Journal of Electronic Packaging, Vol. 122, pp. 233-239.
4. Davies, M.R.D., Cole, R. and Lohan, J., (2000), "Factors affecting the operational thermal resistance of electronic components", ASME Journal of Electronic Packaging, Vol. 122, pp. 185-191.
5. Masud, Bebnia,,Wataru, Nakayama. and Jeffrey, Wan., (1998), "CFD simulations of heat transfer from a heated module in an air Stream: comparison with experiments and a parametric study", IEEE Intersociety Conference on Thermal Phenomena, Vol. 54, pp. 539-544. [83]
6. Dhiman, A.K., Chhabra, R.P. and Eswaran, V., (2005), "Flow and heat transfer across a confined square cylinder in the steady flow regime: effect of peclet number", International Journal of Heat and Mass Transfer, Vol. 48, pp. 4598-4614.
7. Cheng, Y.P., Lee, T.S. and Low, H.T., (2006), "Numerical analysis of mixed convection in three-dimensional rectangular channel with flush mounted heat sources based on field synergy principle", International Journal for Numerical Methods in Fluids, Vol. 52, pp. 987-1003.
8. Tuckerman, D.B. and Pease, R.F., (1981), "High performance heat sinking for VLSI", IEEE Electronic Devices Letters, Vol. 2, pp. 126-129. [84]
9. Wong, H. and Peck, R.E., (2001), "Experimental evaluation of air-cooling electronics at high altitudes", ASME Journal of Electronic Packaging, Vol. 123, pp. 356-365.
10. Liu, D. and Garimella, S.V., (2002), "Investigation of liqiud flow in microchannels", Eighth AIAA/ASME Joint Thermophysis and Heat Transfer Conference, St. Loius, Missouri, Vol. 5, pp. 2002-2776.
11. Harms, T.M., Kazmierczak, M.J. and Gerner, F.M., (1999), "Developing convective heat transfer in deep rectangular microchannels", ASME International Journal of Heat and Fluid Flow, Vol. 20, pp. 149-157.