Study, Optimisation and Numerical Simulation of Natural Convection around the Radial Heat Sink

Md. Farid[1]* Dr. Manoj Kumar Chopra[2] Dr. S. S. Pawar[3]

[1] M.Tech Scholar, Thermal Engineering, SRK University, India

[2] Professor and H.O.D, RKDF Institute of Science and Technology, Bhopal, India

[3] Professor, RKDF Institute of Science and Technology, Bhopal, India

Abstract – This work is on numerical study of natural convection around a radial heat sink having circular base and rectangular fins of various configurations using ANSYS WORKBENCH version 15.0. The material under consideration is aluminium and the free stream fluid is air. The heat transfer rate from the fins, outer wall and the overall heat transfer rate has been evaluated and compared for variety of fin configurations. Surface Nusselt number and surface overall heat transfer co-efficient has been found out. Temperature contours for different fin configuration has been plotted showing the convection currents formed around the heated fin. Velocity contours for various fin configurations also plotted and the motion of heated fluid is shown. Plots for Nusselt number and heat transfer co-efficient are also shown.

The aim of the work is to optimize the existing radial fin. The objective is to increase the heat transfer from the source to the ambient air. This can be achieved by reducing the thermal resistance or by increasing the surface area for the natural convection. In this study the effect on natural convection by different geometrical modification is studied. In this study, natural convection from a heat sink with a circular base and rectangular fins is numerically analyzed, and the thermo-flow pattern is observed. The effects of the holes, slots number of holes and slots, slot length, slot height, hole diameter and heat flux on the thermal resistance and the average heat transfer coefficient are investigated. Study is also carried out on optimizing the existing radial heat sink by changing the mass distribution configuration.

Keyword:- Heat Sink, Angle Orientation, ANSYS, CFD

---------------------------------X---------------------------------

1. INTRODUCTION

The application of fins or plates in heat exchanger is to dissipate heat from internal to external environment. This property of the plate attracts the researchers and analysts because the plate has various applications in the field of heat exchangers, boiling water, steam funnels, heaters, coolers, electrical conveyors, electronic devices, etc. These researches always focused on enhancing of intensity and productivity on the economical path. As all the studies always concentrated on designing a part considering minimum weight and minimum balance is critically required. For increasing the heat transfer capacity blades are broadened in order to achieve a better heat transfer in various devices. So while designing it is considered that, a heat sink is located at appropriate distance and has minimum contact with other parts in a constrained convection and regular constrained working conditions. In order to increase heat capacity, some gadgets are used for increasing the heat transfer in a constrained. There are different shapes of blade are available, but due to straight ness of rectangular blades, are extensively used in assembling. Characteristic convection from a square with blades may well be utilized to reenact wide assortment of planning applications even as offer higher understanding into more and more complicated frameworks of heat transfer, for example electric devices, heat exchangers, refrigerators etc. Cooling of a hardware device with fine heat sinks is a usual practice due to its easy assembling and applying process and high unwavering quality.

Constrained clusters on vertical and even faces are used in the various construction applications, in order to dissipate heat to outer environment. For the purpose of handling, the normally accessible factors are geometry and direction.

The best method for the improvement of heat transfer capacity of any device can be achieved by using the balanced components and the utilization of whole surface area of device. In the various studies, huge amount of balances are used for the small heat sink in gas as working abject. Balancing a plate is an old one technique for reducing heat sink. Now the spiral balancing design of heat sink is in use. And this works with the combination of roundabout plate with various blades layered on the plates. And the plates are joined by the various types of fitments like mechanical, cement, welding, binding, and brazing.

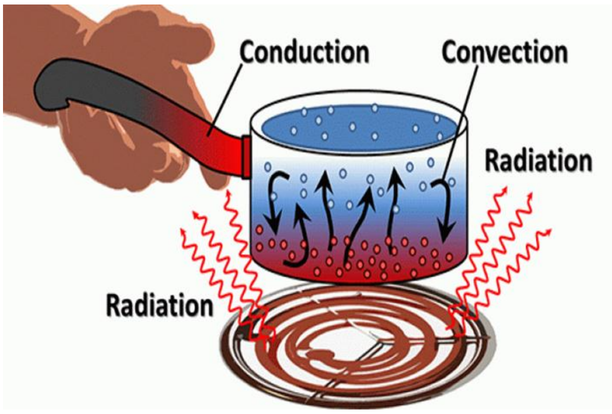

Figure 1 Example of Heat Transfer Process (Source: https://www.machinedesign.com/whats-difference-between/what-s-difference-between-conduction-convection-and-radiation)

1.1 Heat Transfer and Thermodynamics

In various fluids different type of heat transfer phenomenon is seen which the result of heat transfer operations. Various fluids undergo different heat exchange phenomenon to exchange their heat via surrounding, liquid to liquid heat exchange, solid to liquid heat exchange and vice versa. This phenomenon of heat transfer perceives due to some of mechanical changes in the body or some thermodynamical changes. These all changes of heat transfer or we can say energy transfer depends on the factors like temperature of body, motion, and external and internal environmental conditions etc. So, the exchange of temperature due to difference between the two sources or medium is called heat transfer.

1.2 5 Modes of Heat Transfer:

Heat transfer can be achieved by three different modes;

- CONDUCTION
- RADIATION
- CONVECTION

1.2.1 Conduction

When the two objects are in contact and have different temperatures. The heat of the warmer object will travels towards the cooler object, and it will continue until the temperature of both objects will become same. And this is performed by the help of collision of molecules.

1.2.2 Radiation

Basically, radiations are the emitting from the object in the form of electromagnetic waves and travel from one object to other, and this performed without any physical contact between the two objects.

1.2.3 Convection

Convection is the transfer of molecules movement within the fluid like gases, liquids etc. Convection cannot take place in the solids because of the movement of molecules is not possible. Simple example of convection is, when the water is boiled, as we are transferring heat from the bottom layer of water it transfer heat to the upper layers by convection.

2. LITERATURE REVIEW

Kim et al, [1] in the present study CFD analysis has been carried out to find the efficiency of the system having extended surfaces as fins. For the complete study a chamber having the fin structure is designed to perform an analytical study on it. For the analytical study different no. of fins, Rayleigh number is defined to have the desired solution from the overall study. Rayleigh number is calculated using the Nusselt no. and further calculations were carried out. In the study heat transfer phenomenon in the chamber having plate fins and extended fins were simulated to have desired results. Overall from chamber having plate fins showed better results as compared to other designs used for the simulation. The result showed that it shows around 36% higher performance as compared to extended surfaces designs.

Kim et al, [2], in the present study author presented his work on cooling of LED light where different researches are still prevailing to improve its overall performance. In the current work a hollow fin heat sink and a round heat sink is utilised for the simulation purpose. From the study various parameters such as heat transfer, temperature and velocity were identified to provide a suitable conclusion from the study. In this work air is circulated in the heat sink to find its effectiveness for further cooling. From the study a slant shape heat sink best results with for convection. Such studies are based on forced convection phenomenon. In complete analytical work Rayleigh number ranging from the values 300000-1000000 was selected with different fin numbers ranging from 9-72. Variations in Fins are done to increase the surface area for air contact which is seen responsible for the result improvement study.

Lee et al [3], In the present investigation, a heat sink of LED lighting was enhanced regarding its fin-tallness profile to acquire solid cooling execution for high-control LED lighting applications. In this, convection and radiation heat transfer is modelled and tested in order to approve its numerical design. For the result depiction and optimum value consideration Pareto procedure was utilised. From the study the results of radiant heat transfer and convection were obtained. From the study overall 45% efficiency of fin was improved to provide efficient solution in LED Performance. This performance of fins can eradicate various heat based problems seen at the time of working. This study was conducted analytically using CFD as the base operation for the simulation for the meshing ICEM surf was utilised.

Lee et al [4], in this author explained about the heat sink performance used as heat transferring device used in various application. Heat sink has high application in electrical devices were electrical systems are maintained in their desired temperature using the heat sink cooling proficiency. For finding the efficiency of overall system in the present work convection and radiation is utilised where air with certain velocity is passed in the chamber or enclosure where heat sink is placed, When the air gets in contact with these extended surfaces which are made of materials having high thermal conductivity gets cooled with their interaction. In the current work Length of the fin is varied to have more surface area for the contact of heat sink. As a result high efficiency is seen in the present work. For the CFD analysis a model is created using the modelling software and further the geometry is used for the heat transfer simulation. During the complete working various operations were performed on the heat sink such as name selection, meshing and pre-processing of boundary conditions etc. results from the study showed the improvement of 12.3% whereas the mass is increased by 20%.

Wange et al [5], in the present study an experimental and an analytical study has been performed on this heat sink. From the study results of heat transfer coefficient were estimated to find the efficiency of overall heat sink device. Results obtained from the experimental study were validated with the results of numerical study performed using different fluid flow software's. Heat sink having large surface contact area showed better results as compared to the heat sink having less surface area of contact. The study was based on the forced convection procedure where air was used as the parameter for the judgment and different contours of temperature, velocity and pressure were obtained to compare the results of experimental and analytical based simulation. Further from the study it is concluded that maximum the surface area maximum will be the heat transfer from the heat sinks.

Hwan et al [6], in present study, the experimental and numerical investigation of natural convection in radial heat storage sink, which have a horizontal circular base and rectangular fins. The flow pattern of chimney, i.e., Air enters from outside and then it is passes from fins after heating and then flow from the inner region of the heat sink. In this, parameters on the basis of three different geometries (fins, fin length, and number of fins) and a parameter without any add (Heat flux) on the thermal resistance and average heat transfer for the heat sink. And also correlation is used to predict average Nusselt for radial heat sink. And it is resulted that from the study, thermal resistance decreased and the heat transfer coefficient increased in proportion to heat flux applied to heat sink.

3. OBJECTIVE

1. To study the effect of holes, slots on the radial heat sink fins and understanding the possibility of reduction of mass in the radial heat sink for same thermal performance.

2. To study the effect of orientation of the radial heat sink.

3. To study the effects of interrupted fins, understanding the aftermath of the distribution of mass.

4. METHODOLOGY

Step of working

1. Selection of base model for the further study.

2. Modelling of geometry using the selected parameters.

3. Name selection on different components and domains used.

4. Meshing of overall system.

5. Result simulation of heat sink.

6. Result evaluation on behalf of heat transfer, temperature and velocity etc.

5. DESCRIPTION OF REFERENCE MODEL

Seung-hwan yu, Kwan-soo Lee, Se-jin Yook [6] studied the effect of fin length, height, number of fins on a single operating parameter heat flux. Numerical simulation was conducted on Ansys fluent based on finite volume method. Geometric parameters of the model are mentioned in below

Table 1. Description of the Reference Model

Number of fins	20
Outer radius of base plate	75 mm
Length of the fin	55 mm
Length of the fin	21.3 mm
Thickness	2 mm
Inner radius of base plate	12.5 mm

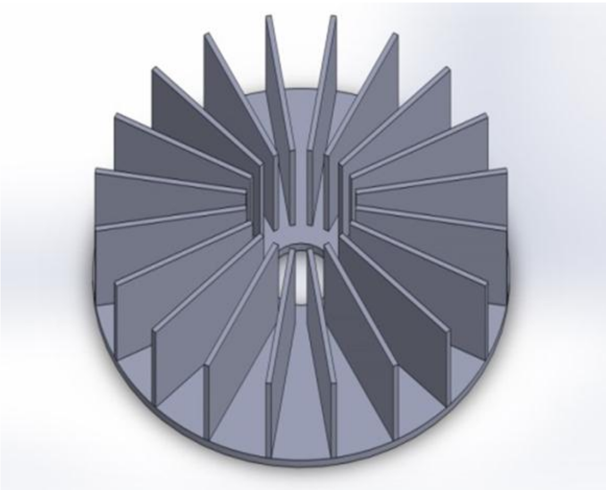

Figure 2. Radial heat sink with a circular base and rectangular fins

6. MESHING

The model in Fig 4.2 shows the heat sink with its domain generated in solid works 2016 and exported as .STEP file to the Ansys fluent 15. Boolean operation of subtracting base from domain was done. Assigned solid to the base and fluid to the domain.

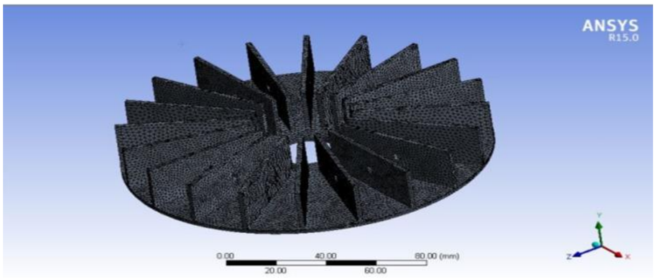

Figure 3: 4mm heats sink meshing

7. RESULT

VALIDATION OF SIMULATION PROCEDURE

The procedure used in this numerical analysis is validated with available numerical analysis. A rectangular plate heat sink analysis carried out by [6] (Ref)'s model is created using CAD software with existing parameter For the purpose of validation of the solution procedure.

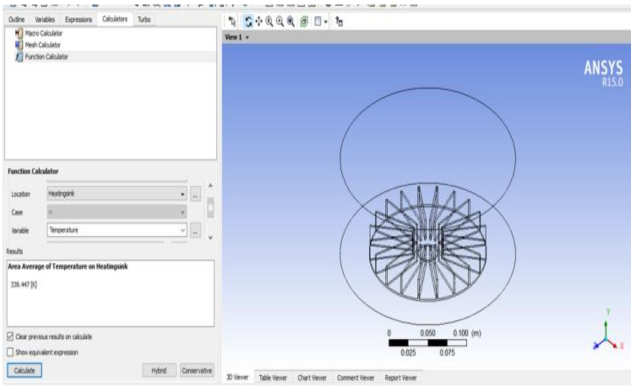

Figure 4: Average temperature of reference model heal sink

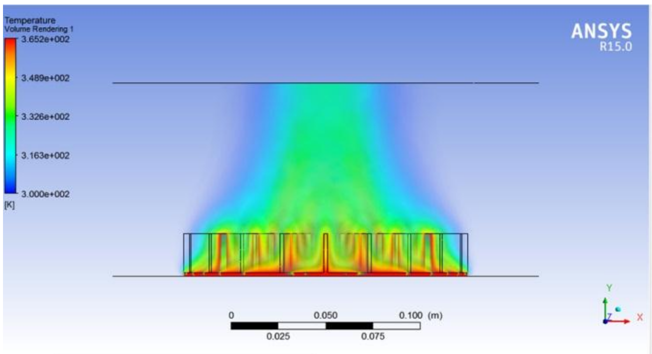

Figure 5: Temperature flow of reference model heal sink

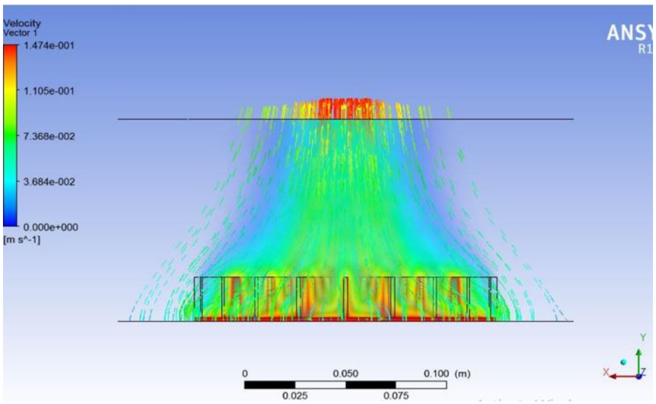

Figure 6: Velocity vector of reference model heal sink

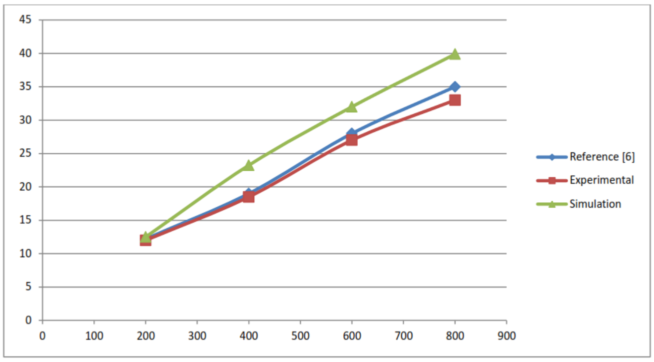

Figure 7: Comparison of simulation and reference results

EFFECT OF HOLES IN FIN

Radial heat sink with holes 4mm 8mm 12mm was studies for 800W/m^2 was studied. And the results were tabulated

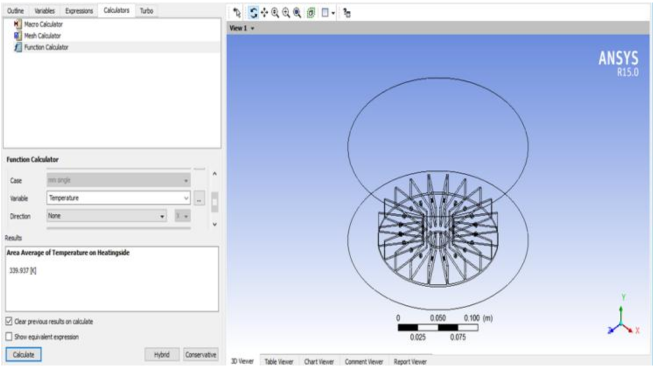

Figure 8: Average temperature of 4mm hole heal sink

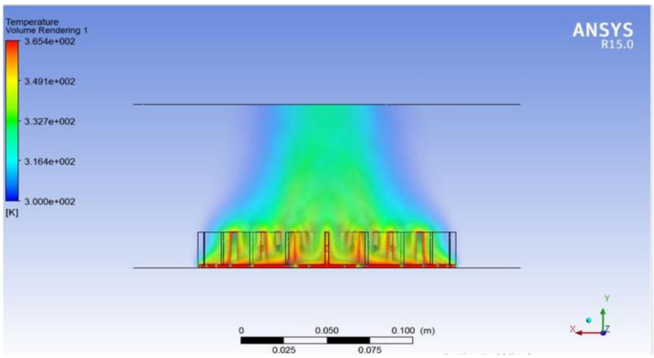

Figure 9: Temperature flow of 4mm hole heal sink

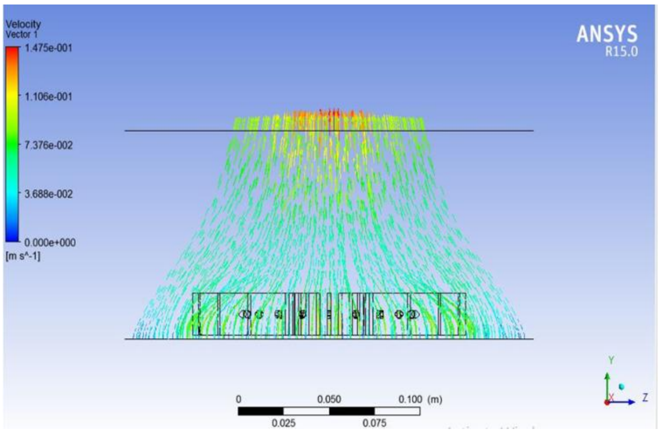

Figure 10: Velocity vector of 4mm hole heal sink

Result for 8mm hole diameter heat sink

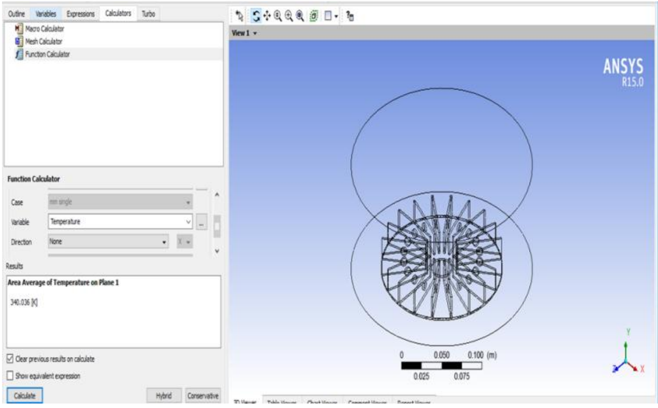

Figure 11: Average temperature of 8mm hole heal sink

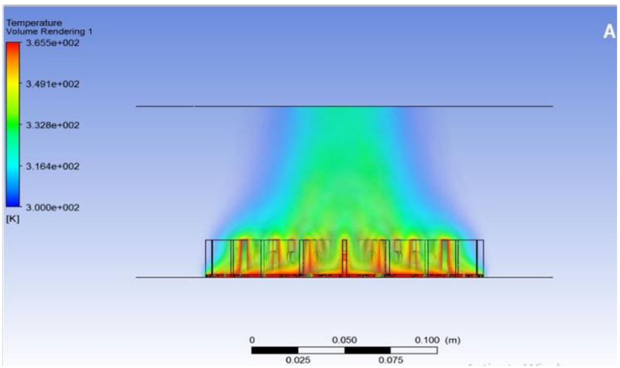

Figure 12: Temperature flow of 8mm hole heal sink

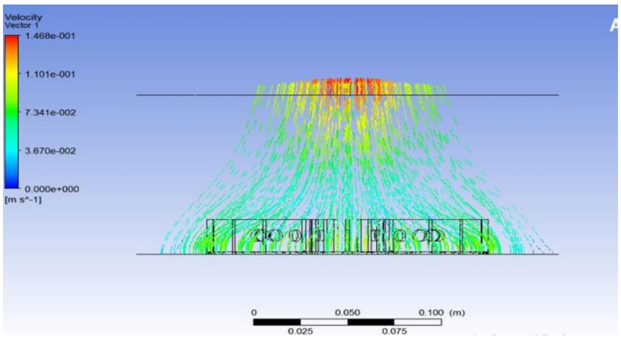

Figure 13: Velocity vector of 8mm hole heal sink

Result for 12mm hole diameter heat sink

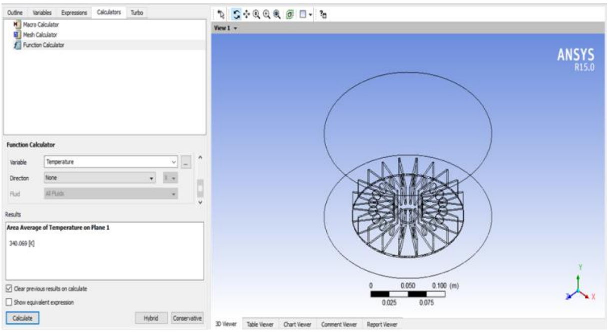

Figure 14: Average temperature of 12mm hole heal sink

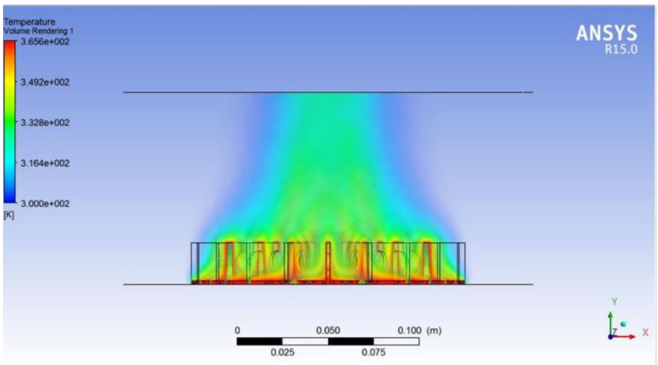

Figure 15: Temperature flow of 12mm hole heal sink

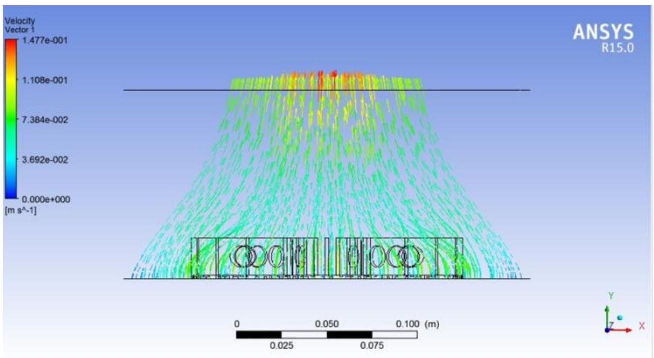

Figure 16: Velocity vector of 12mm hole heal sink

Table 2: Temperature base for holes on fins and thermal resistance

Hole Diameter	Average temperature	Thermal resistance
No hole	339.447	2.8818
4mm	339.937	2.9176
8mm	340.036	2.92489
12mm	340.069	2.9273

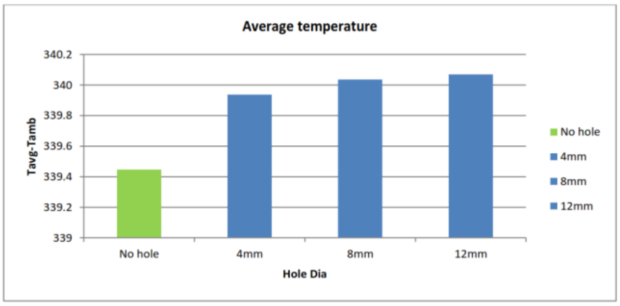

Figure 17: Rise in temperature of the base v/s hole diameter

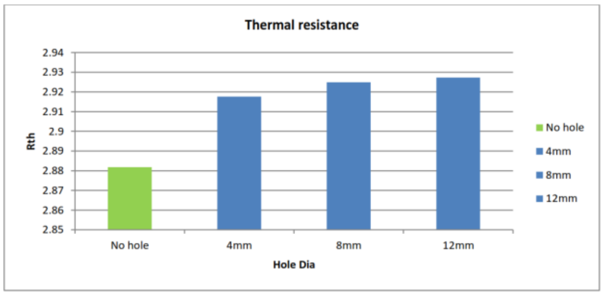

Figure 18: Thermal resistance v/s hole diameter

It can be seen from the Fig 5.14 that as the hole size increases the difference between the ambient and average temperature also increases. Thermal resistance also increasing. So making a hole for an optimized fin will not yield any additional benefit. Fig 5.15 shows the thermal resistance for different sized hole.

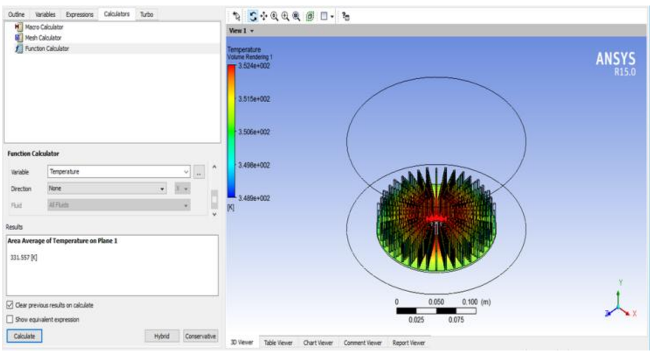

Figure 19: Temperature gradient of heat sink isometric view and avg temp

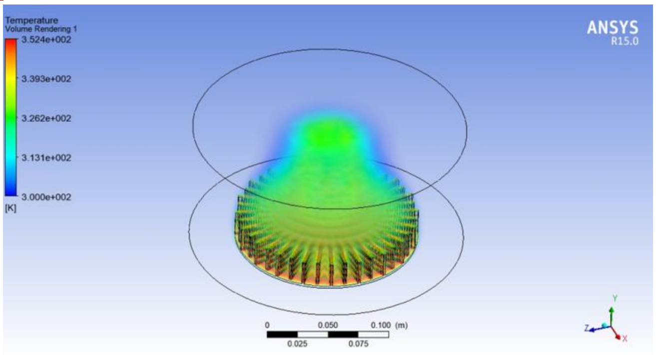

Figure 20: Temperature contour of fin and domain

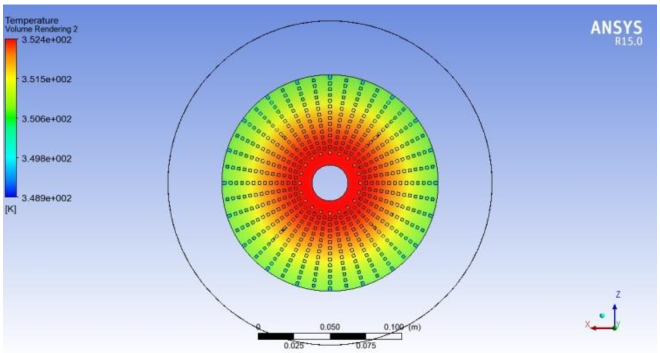

Figure 21: Temperature gradient of heat sink top view

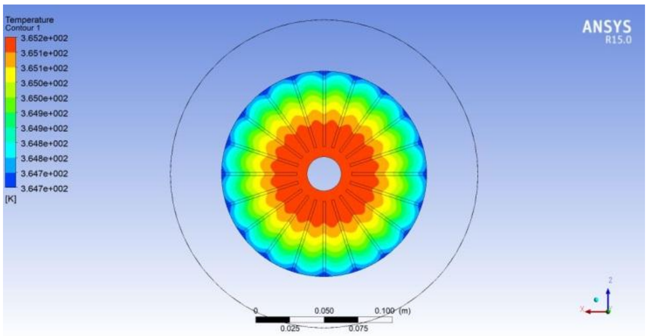

Figure 22: Temperature distribution at the base of reference heat sink

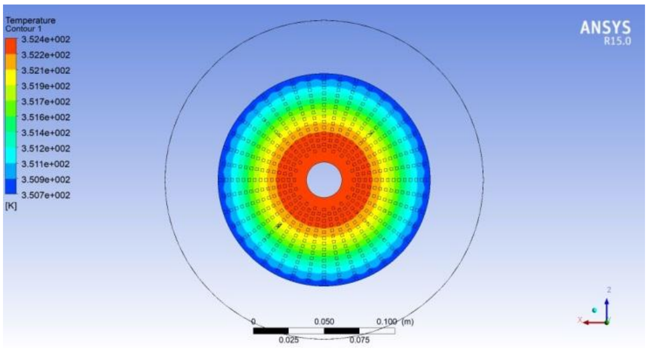

Figure 23: Temperature distribution at the base of IA3 heat sink

7.4 Thermo Flow Characteristics

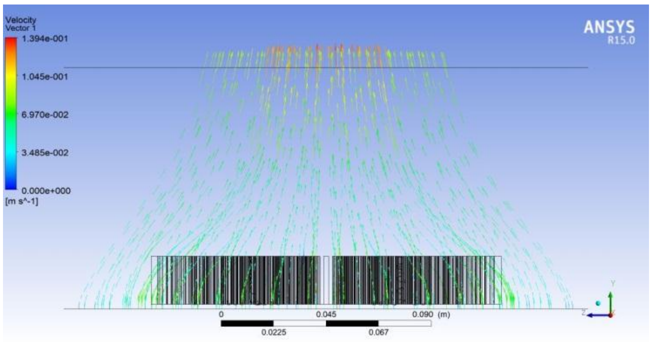

Figure 24: Flow visualization and velocity contour of air

EFFECT OF INSTALLATION ANGLE

Table 3: Base temperature of reference and IA3 for orientation

Angle	Reference model temp (K)	IA3 Temp (K)
0	339.447	317.047
30	337.742	327.756
45	336.951	330.616
60	335.902	333.72
90	336.759	335.536

From the above table when compared to the reference model the IA3 base temperature is increased for increase in orientation angle. So IA3 performs better when kept in horizontally. But for reference model the base temperature increased till it the installation angle 60°. At 60° it has a minimum base temperature and again at 90 degree the base temperature has increased slightly.

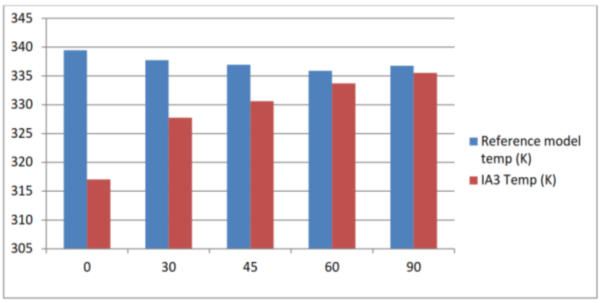

Figure 25: Comparisons of temperature variation in reference model and IA3 with respect to orientation angle

8. CONCLUSION

The thermal performance and fluid flow of three dimensional radial heat sink is studied using Finite volume method. The heat sink orientation for natural convection was analysed. By visualizing the numerical analysis data, we observed airflow structures within the channels formed by the fins as well as in the vicinity of the heat sink. The general flow pattern was like that of a chimney; i.e., the cooling air entering from outside was heated as it passed between the fins, and then rose from the inner region of heat sink. Studies were performed to compare the effects of the Holes, Slot, Interrupted, Redistribution fin mass and orientation on the thermal resistance and the heat transfer coefficient. The Holes, slot, interruption increased, the thermal resistance and heat transfer coefficient generally decreased. However, after redistributing the interrupted fin mass heat transfer increased, obtained an effective low heat sink temperature and found more even heat distribution at the base of the heat sink when compared to the reference model. The thermal resistance decreased and the heat transfer coefficient increased in proportion to the heat flux applied to the heat sink base. Based on the analysis the fin with 2mm cut redistributed mass is a better model when compared with the reference model. It can be seen that from the analysis both mass reduction and thermal performance enhancement cannot be done simultaneously.

REFERENCES

1) Kuen Tae Park, Hyun Jung Kim, Dong-Kwon Kim (2014). “Experimental study of natural convection from vertical cylinders with branched fins”, Experimental Thermal and Fluid Science, 54, pp. 29–37.

2) Byoung Hoon An, Hyun Jung Kim, Dong-Kwon Kim (2012). “Nusselt number correlation for natural convection from vertical cylinders with vertically oriented plate fins”, Experimental Thermal and Fluid Science, 41, pp. 59–66

3) Daeseok Jang, Se-Jin Yook, Kwan-Soo Lee (2014). “Optimum design of a radial heat sink with a fin-height profile for high-power LED lighting applications” Applied Energy, 116, pp. 260–268.

4) Daeseok Jang, Seung-Hwan Yu, Kwan-Soo Lee (2012). “Multidisciplinary optimization of a pin-fin radial heat sink for LED lighting applications”, International Journal of Heat and Mass Transfer, 55, pp. 515– 521

5) S M. Wange & R. M. Metkar (2013). “Computational Analysis of Inverted Notched Fin Arrays Dissipating Heat by Natural Convection”, International Journal of Engineering and Innovative Technology, 2-11, pp. 245-249.

6) Seung-Hwan Yu, Daeseok Jang, Kwan-Soo Le (2012). “Effect of radiation in a radial heat sink under natural convection”, International Journal of Heat and Mass Transfer, 55, pp. 505–509.

7) Seung-Hwan Yu, Kwan-Soo Lee, Se-Jin Yook (2011). “Optimum design of a radial heat sink under natural convection”, International Journal of Heat and Mass Transfer, 54, pp. 2499–2505.

8) Seung-Hwan Yu, Kwan-Soo Lee, Se-Jin Yook (2010). “Natural convection around a radial heat sink”, International Journal of Heat and Mass Transfer 53, pp. 2935–2938

9) S. R. Dixit, D. P. Mishra, T. C. Panda (2012). “Experimental Analysis of Heat transfers & Average Heat Transfer Coefficient Through fin Array With or Without Notch Using Free Convection”, International Journal of Advance Research, 1-2, pp. 11-21.

10) S.S. Sane, N. K. Sane, G.V. Parishwad (2008). “Computational Analysis Of Horizontal Rectangular Notched Fin Arrays Dissipating Heat By Natural Convection” 5 th European Thermal-Sciences Conference, The Netherlands.

11) Qie Shen, Daming Sun, Ya Xu, Tao Jin, Xu Zhao (2014). “Orientation effects on natural convection heat dissipation of rectangular fin heat sinks mounted on LEDs”, International Journal of Heat and Mass Transfer, 75, pp. 462–469.

12) E. A. M. Elshafei (2010). “Natural convection heat transfer from a heat sink with Hollow/perforated circular pin fins”, Energy, 35, pp. 2870- 2877.

13) Mika Maaspuro, Aulis Tuominen (2013). “Thermal analysis of LED spot lighting device operating in external natural or forced heat convection”, Microelectronics Reliability, 53, pp. 428–434.

14) K.E. Starner, H.N. McManus, An experimental investigation of free convection heat transfer from rectangular fin arrays, J. Heat Transfer 85 (2) (1963) 273– 278.

15) J.R. Welling, C.B. Wooldridge (1965). Free convection heat transfer coefficients from rectangular vertical fins, Trans. ASME J. Heat Transfer 87 (3) pp. 439–444.

16) F. Harahap, H.N. McManus (1967). Natural convection heat transfer from horizontal rectangular fin arrays, J. Heat Transfer 89 (1), pp. 32–38.

17) R.T. Huang, W.J. Sheu, C.C. Wang (2008). Orientation effect on natural convective performance of square pin fin heat sinks, Int. J. Heat Mass Transfer 51 (9–10) pp. 2368–2376.

18) S. Baskaya, M. Sivrioglu, M. Ozek (2000). Parametric study of natural convection heat transfer from horizontal rectangular fin arrays, Int. J. Therm. Sci. 39 (8) pp. 797–805.

19) Yunus A Cengel: Heat and mass transfer fundamentals and applications 5th edition.

20) John F. Anderson theory on computational fluid dynamics 6th edition.

Corresponding Author

Md. Farid*

M.Tech Scholar, Thermal Engineering, SRK University, India