Journal of Modern Science and Technology
Vol. 4. No. 1. September 2016 Issue. Pp. 135 – 143

CFD Analysis of Forced Convection over Radial Heat Sink

Mirza Md. Symon Reza[1], Ashutosh Kumar Yadav[2] and Sagar Dasgupta[3]

The Electronics industry is continually striving to achieve improved performance and smaller scales. Both of the targets result in increased heat flux densities within the electronic devices. Mainly the portable devices which are the market driving forces, severely faces this problem of increased heat flux densities. This phenomenon ispushing the researchers forth to implement new methodologies to provide thermal management within such devices. This work addresses the potential of low profile integrated fan and heat sink solutions to electronics thermal management issues, particularly focusing upon possible solutions in low profile portable electronics. We investigated a heat sink design with mini-channel features, applicable to low profile applications. After the analysis on the design the results were discussed for given operating conditions. The operating conditions which includes operating temperatures, fan speed, etc. The geometry of theradial heat sink was modeled in CATIA V5 design software. The analysis was performed on ANSYS-FLUENT software.

Field of Research*:* Heat Transfer, CFD, Forced Convection, Radial Heat Sink

1. Introduction

A heat sink is a component designed to lower the temperature of an electronic device by dissipating heat into the surrounding air. A heat sink without a fan is called a passive heat sink and one with a fan is called active heat sink. Heat sinks are used with high-power semiconductor devices such as power transistors and optoelectronics such as lasers and light emitting diodes (LEDs), where the heat dissipation ability of the basic device is insufficient to moderate its temperature. 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. Heat sink attachment methods and thermal interface materials also affect the die temperature of the integrated circuit. Thermal adhesive or thermal grease improve the heat sink's performance by filling air gaps between the heat sink and the heat spreader on the device. In a radial heat sink the fins are arranged radially at regular interval. A radial heat sink composed of a circular base and pin fins. The fins are radially arranged at regular intervals. The base of the heat sink is positioned horizontally, while the fins are attached vertically. The fin arrangement is repeated around the circumference of the base. Fans are used when natural convection is insufficient to remove heat.

[1]Mirza Md. Symon Reza, National Institute of Technology, Jamshedpur, India,
Email: mirza.11ugme612@nitjsr.ac.in
[2]Ashutosh Kumar Yadav, National Institute of Technology, Jamshedpur, India,
Email: ashutosh.11ugme014@nitjsr.ac.in
[3]Sagar Dasgupta, Motilal Nehru National Institute of Technology, Allahabad, India,
Email: sagar.me15@mnnit.ac.in

Fans may be fitted to the computer case or attached to CPUs, GPUs, chipset, PSU, hard drives, or as cards plugged into an expansion slot. Common fan sizes include 40, 60, 80, 92, 120, and 140 mm. 200, 230, and 250 mm fans are sometimes used in high-performance personal computers.

Studies have been going on to higher the rate of heat dissipation from the electronic devices in order to increase the durability. There have been studies regarding natural convection over plate fins, both square and rectangular type. Recently researchers are concentrating on radial heat sinks as they are commonly used in most of the portable electronic devices. In this research work, computational analysis of forced convection over radial heat sink is done, taking the most common type of heat sink in consideration. It will pave the way for future research on various types of heat sink with varying material and configuration in order to optimize the size, weight, heat transfer capability of the heat sink as per the necessity. Studies have been done on heat dissipation over various fin configurations. But analysis on radial heat sink has started recently as the use of radial heat sink increased in past few years, basically in portable electronic devices. Few of the studies were on natural convection, while this research covers computational analysis of forced convection over radial heat sink. This study deals with the case of L-Type radial heat sink. Here, the heat sink and fin design was chosen in accordance with recent market trends. Throughout the analysis, the heat sink dimensions were kept constant. The sink dimension is that of contemporary sinks used in Laptops and graphic processors.

In this paper, Literature review is presented to introduce the scholars about the chronological researchers relating this topic in section 1. In section 2, the governing equations, and details of the geometry on which the simulation is done along with the boundary conditions of the simulation is presented. In Section 3, results are published and discussion was listed based on the result simulated by CFD software ANSYS. In section 4, a conclusion is given and scope of future research is stated.

2. Literature Review

Heat sink research and development has had a long history which is still ongoing with efforts to improve design and performance. As Incopera and DeWitt state, “With heightened concern for energy conservation, there has been a steady and substantial increase in activity. A focal point of this work has been heat transfer enhancement, which includes the search for special heat exchanger surfaces through which enhancement may be achieved.” 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. Early works in the field of forced convection include G. Stanescu, A. Fowler & A. Bejan, The optimal spacing of cylinders in free-stream cross-flow forced convection, Int. J. Heat Mass Transfer (1996). This paper focuses on forced convection in cylinders and gives insight into possible designs of storage cylinders, tanks etc. Further work includes N. Anand, S. Kim & L. Fletcher paper on ‘The effect of plate spacing on free convection between heated parallel plates’, J. Heat Transfer (1992). In 2005, R. Prasher & R. Mahajan got patent for ‘Two phase cooling utilizing micro channel exchangers and channeled heat sink. In the same year, J. Valenzuela, T. Jasins & Z. Sheik wrote a paper on ‘Liquid cooling for high-power electronics, power electronics’ which described numerous liquid based cooling schemes.

Some studies have addressed piezoelectric based fans, but their low flow rate combined with the requirement for high voltages makes them unlikely competitors in the portable electronics market place. Recently, studies addressing the aerodynamics effects of geometrical scaling in both axial fans and radial fans, has led to the development of low profile rotary fans for potential use in electronics cooling applications. Also, Yongping Chen, Mingheng Shi & Jiafeng Wu, wrote a paper on 'Three dimensional numerical simulation of heat and fluid flow in noncircular micro channel heat sinks' which added to increasing reserves of literature on heat sink analysis. In the following year Reng-Tsung Huang, Wen- Junn Sheu & Chi-Chuan Wang, discussed Orientation effect on natural convective performance of square fin heat sinks in their study published in International Journal of Heat and Mass Transfer.

Other technologies under development for thermal management of portable electronic devices are phase change materials, micro-heat pipes and high conductivity materials. However such technologies focus upon heat spreading and transport within a device rather than active removal of heat from the device. For example, the phase change materials store heat to be dissipated over time, while heat pipes and high conductivity materials only provide paths of reduced thermal resistance to the flow of heat within devices. Therefore, at present this heat is finally being removed by some combination of natural convection, conduction and radiation in many devices.

A recent approach to enhancement of heat transfer densities is the use of multiple scales in heat exchanger design. This approach is in line with the approach to achieve better designs by endowing the flow configuration with the freedom to morph. The multi-scale approach has been demonstrated numerically for multi scale cylinders with natural convection by, T. Bello-Ochende, A. Bejan Int. J. Heat Mass Transfer (2005). Further T. Bello-Ochende, A. Bejan, wrote a paper on 'Constructal multi- scale cylinders in cross flow', Int. J. Heat Mass Transfer in the same year. Both these papers discussed various numerical techniques for better flow analysis. Other works include T. Bello-Ochende & A. Bejan, paper on 'Maximal heat transfer density: Plates with multiple lengths in forced convection',Int.J.Thermal Sci. (2004) S.A.

Nada's paper on 'Natural convection heat transfer in horizontal and vertical closed narrow enclosure with heated rectangular finned base plate', International journals of Heat and Mass Transfer 50 (2007) was further proof of the research that was being conducted in this field. M Dogan & M. Sivrioglu, 'Experimental investigation of mixed convection heat transfer from longitudinal fins in a horizontal rectangular channel', International journals of Heat and Mass Transfer (2010) was also noteworthy as it described experimentally the convective heat transfer from long fins in heat sink. L-Type radial heat sink was analyzed under forced convection. CFD analysis was done and the operating conditions were chosen to be the normal working conditions of Laptops and graphic processors. The result was tabulated. Various plots generated via CFD Ansys software.

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. 'Temperature contour' visualize the rate of heat dissipation from the fin surface.

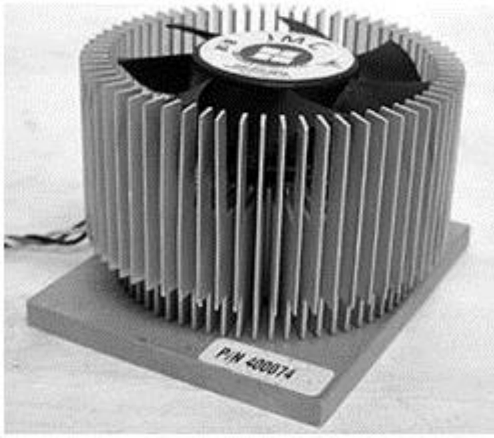



Figure 1: Radial Heat Sink

3. Governing Equations

Mass Conservation Equation

$$\frac{\partial \rho}{\partial t} + \nabla . \rho \vec{V} = 0$$

Momentum Conservation Equation

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}(\lambda + \mu)\nabla . \vec{V} + \frac{\partial}{\partial x_i}\left(\mu \frac{\partial u_i}{\partial x_i}\right) + \rho b_i$$

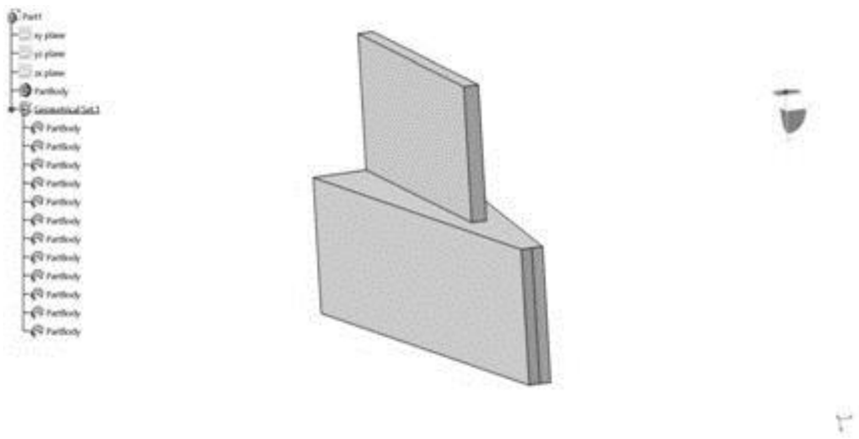
Energy Conservation Equation (for incompressible flow)

$$\rho C_p \frac{DT}{Dt} = -\left(\frac{d\dot{q}_x}{dx} + \frac{d\dot{q}_y}{dx} + \frac{d\dot{q}_z}{dx}\right) + \dot{q_h}$$

4. Geometry

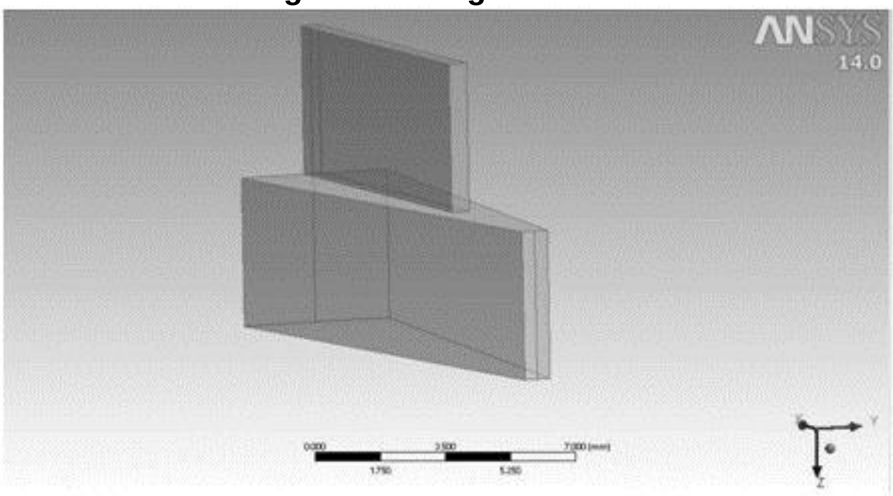
A 3-dimensional geometry of radial heat sink is created in CATIA. The regular rectangular meshes have been taken. An axis-symmetric model has been made in order to reduce the time consumption in simulation. The main components of the design are fins and base. The dimensions of various components are as follows and the subsequent figure which are taken from CATIA. The fin's height is taken as 4 cm, thickness 0.5 cm and length 14 cm. The base's outer radius 20 cm inner hole radius 4 cm.

Figure 2: Design of a Single Fin CATIA



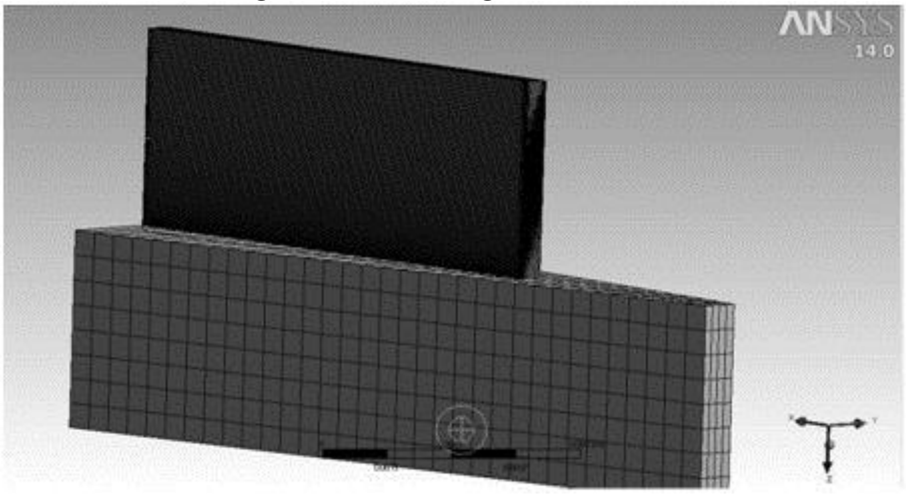
Later the design was imported into the workbench for the analysis purpose. The model was first imported into design modeler, here all the sections and part are named and an enclosure is created for the model.

Figure 3: Design Modeler



Now after the design modeler the model was imported into the meshing workbench. Here meshing is done for the model. Since our aim was mainly concentrated on fin therefore the base was coarsely meshed and the fin was meshed very fine for getting better result.

Figure 4: Meshing Workbench



After the meshing operation the model was finally imported into the fluent setup workbench for the analysis purpose**.**

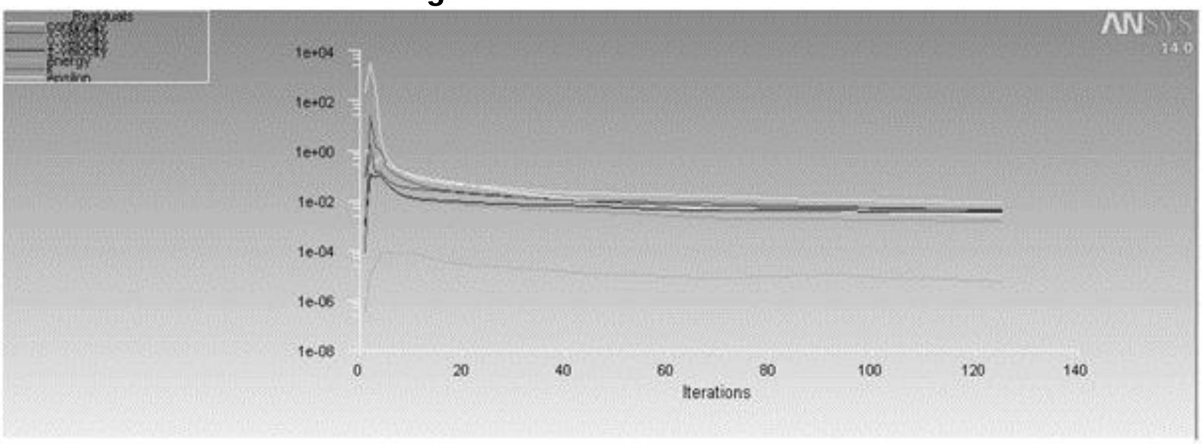
5. Result and Discussion

The simulation for heat sink with L-type configuration was run for a set of operating conditions stated above. Given that this type of heat sink is very common for practical uses. This type of heat sink is easier to manufacture. The result generated by the analysis is presented below in the form of charts and plots. This analysis provides a clear view of the heat phenomenon which was never been done earlier, will open the window of the effective use of such fins in different engineering applications.

5.1 Cooling Behavior

The base of the heat sink was kept at the constant temperature of 350℃ which is the expected temperature at normal operating conditions. The residual monitor curve shown below gives the convergence of continuity equation, energy equation and velocity curve.

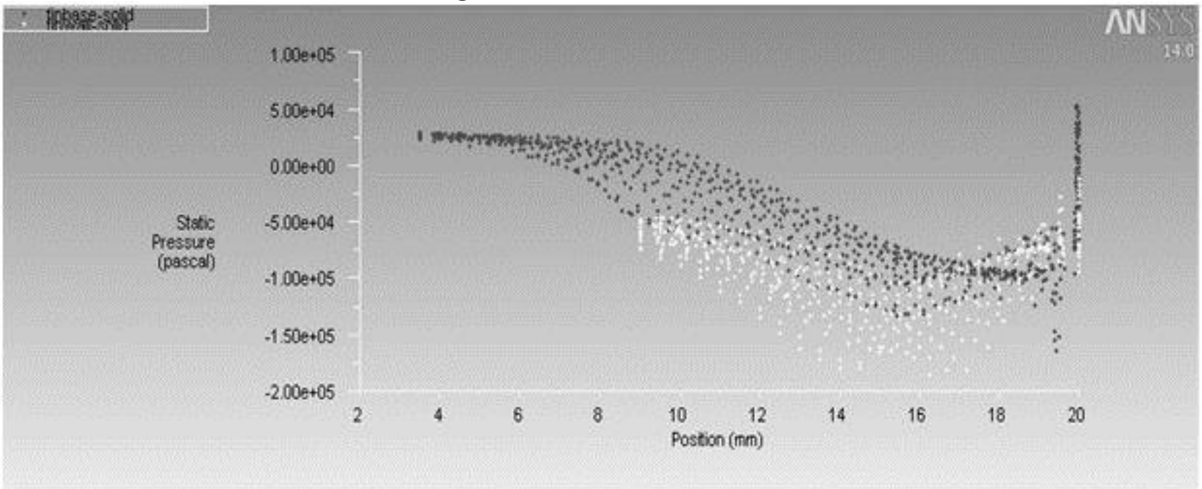
Figure 5: Residual Monitor



5.2 Pressure Behavior

The pressure variation is shown in the following plot of pressure variation along the length of the fin. The pressure at the inlet of the fin was taken to be equal to 1 atm. The plot below shows that pressure decreases as we move along the fin length.

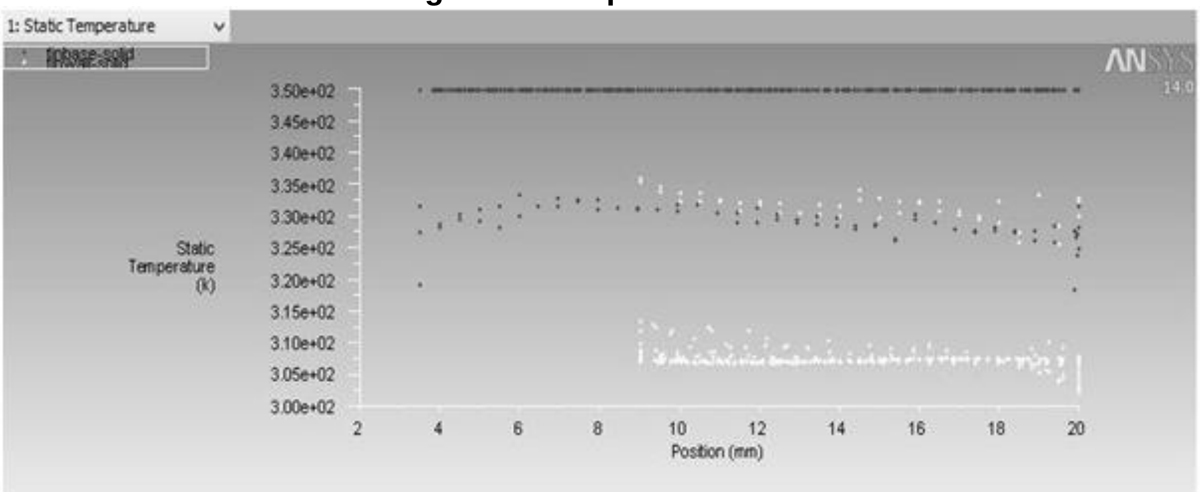
Figure 6: Pressure Plot



5.3Temperature Plot

The Temperature of the base of the Fin was kept constant at 350℃. The ambient temperature is taken to be 27ºC. The temperature variation along the fin is shown by the temperature contour shown below. It can be seen from the plot that with the passage of time heat sink dissipates heat into the atmosphere and subsequently the temperature of the heat sink changes. The temperature at the base of the fin is the largest while it is nearly uni format the top of the fin. The dotted curve in the middle shows that maximum heat transfer takes place through the middle portion of the fin which is receiving maximum air flow.

Figure 7: Temperature Plot



5.4 Temperature Contour

The analysis of the heat sink gave the following temperature contour. The contour for the side view of the fin and the top view of the base without the fin is shown below.

Figure 9: Side View of Temperature Contour

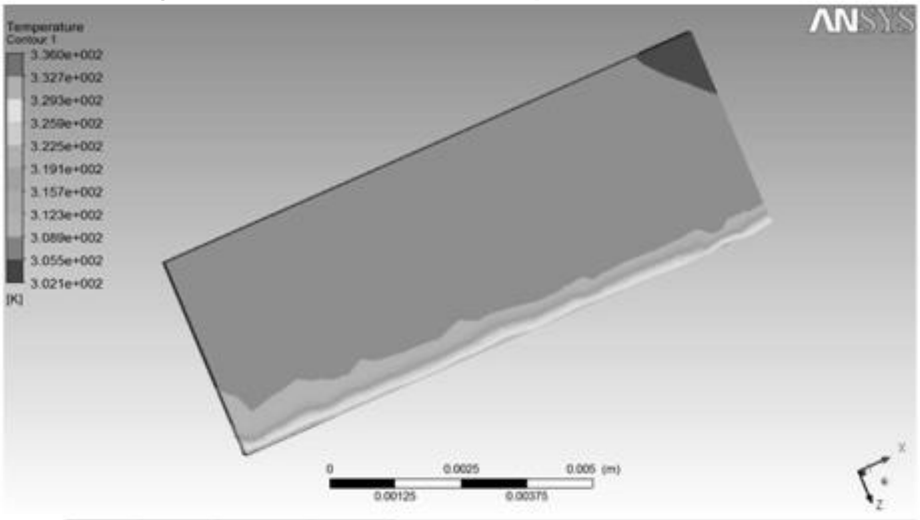
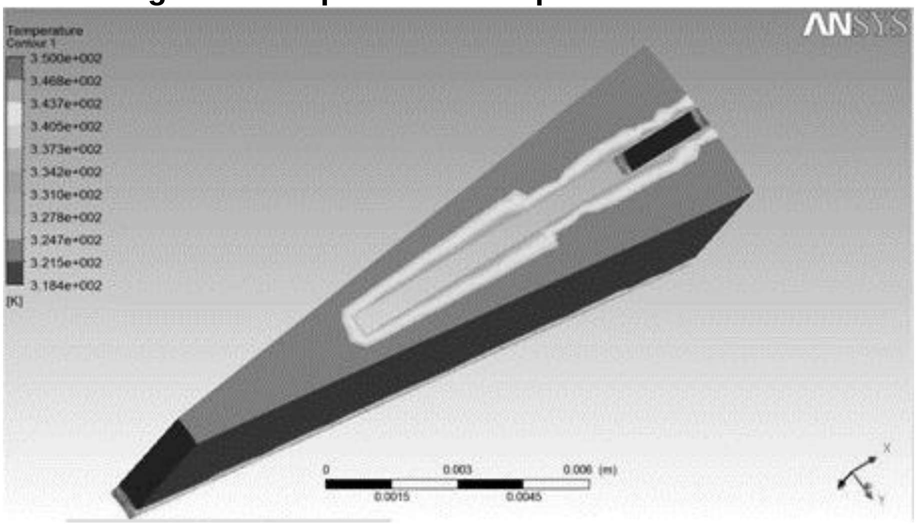


Figure 10: Top View of Temperature Contour



6. Conclusion

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. 'Temperature contour' visualize the rate of heat dissipation from the fin surface. Comparison can be done using of various fin configurations, various materials, various operational condition and the best suited heat sink can be chosen following this research work.

This study has some limitations as well. The analysis has been performed using L-type fins only. For getting a more diverse idea of cooling of heat sinks several other designs can be used. The research so far has focused on the use of common materials for the construction of heat sink. So far the materials used were aluminum, copper and steel, etc. The further advancement can be the use of piezoelectric materials which provide much larger heat transfer. In the present study, air has been chosen as the cooling medium. Larger heat transfer coefficients are possible with use of a liquid cooling medium. However their feasibility with small devices remains an issue. The analysis carried out was based on normal variation in operating conditions. Extreme cases such as when the device running at high temperatures is not analyzed. In many cases heat transfer by natural convection can be dominant than by forced convection. Here we have considered forced convection as the only mechanism.

Reference

Acikalin, T, Sauciuc, I and Garimella, SV 2005, 'Piezoelectric actuators for low formfactor electronics cooling', The ASME/Pacific Rim Technical Conference and Exhibition on Integration and Packaging of Micro, Nano, and Electronic Systems (InterPACK '05), San Francisco, July 17–22, 2005.

Anand, N, Kim, S and Fletcher, L 1992, 'The effect of plate spacing on free convection between heated parallel plates', *J. Heat Transfer, Vol.114, No. 2, Pp. 515–518.*

Bejan, A and Fautrelle, Y 2003, 'Constructal multi-scale structure for maximal heat transfer density', Acta *Mech. Vol. 163, No. 1–2, Pp. 39–49.*

Bejan A, Dincer I, Lorente, S, Miguel, AF and Reis, AH 2004, 'Porous and Complex Flow Structures', Modern Technologies, Springer-Verlag, New York, 2004.

Bello-Ochende T and Bejan, A 2005, 'Constructal multi-scale cylinders with natural convection', *Int. J. Heat Fluid Flow Vol. 48 Pp. 4300–4306*.

Chen, Y, Zhang, C, Shi, M and Wu, J 2009, 'Three dimensional numerical simulation of heat and fluid flow in noncircular micro channel heat sinks', *International Communications in Heat and Mass Transfer*, Vol.36, No.36, Pp. 917-920.

Huang, RT, Huang, WJ and Wang, CC 2008, 'Orientation effect on natural convective performance of square fin heat sinks', *International journals of Heat and Mass Transfer, Vol. 51, Pp. 2368-2376.*

Kimber, M, Garimella, SV and Raman A 2006, 'Experimental mapping of local heat transfer coefficients under multiple piezoelectric fans', ASME International Mechanical Engineering Congress and Exposition, IMECE2006-13922, Chicago, IL, November 2006.

Stanescu, G, Fowler A and Bejan A 1996, 'The optimal spacing of cylinders in free stream cross-flow forced convection', *Int. J. Heat Mass Transfer, Vol. 39, No. 2, Pp. 311–317.*