Numerical Investigation of Fin Effectiveness of Heat Sink With Different Fins

Bodi Dharmateja[1], D.Kiran Babu[2]
[1]Dept of Mechanical Engineering
[2]Associate Professor, Dept of Mechanical Engineering
[1, 2] Sri Chaitanya DJR College Of Engineering, Vijaywada , A.P

Abstract- *Thermal management of electronic components has been one of the primary areas of focus in advanced heat transfer research and development. This has been especially true in the evolution of microelectronics over the past several decades. Heat transfer behavior is complex, as heat is dissipated in the chip, conducted into the substrate and then transferred to the surroundings by some combination of thermal conduction, convection and radiation. The increase of power density in microelectronic packages has underlined the need for employing effective cooling devices and cooling methods to maintain the operating temperatures of electronic in this paper Various shapes and types of fins is Modelled and simulated using CFD software it is observed that there is significant increase In staggered fins.*

Keywords- CFD heat sink , Fin types.

I. INTRODUCTION

The performance of heat sinks has been the focus of many investigations in recent years, and the subject has been treated analytically, numerically, and experimentally. Most of the work has dealt with heat sinks in fully shrouded configurations, but several authors have begun to address the issue of heat sinks in ducted flow with tip or lateral clearance or both. The optimal design of heat sinks is also addressed in some research studies using parametric optimization, entropy generation minimization, and least energy optimization techniques.

Many studies of fully shrouded heat sinks are found in the open literature. The common objective of these studies was to design a heat sink for optimal thermal performance. They studied the influence of fin spacing, fin thickness, number of fins, fin height and fin length on thermal performance of heat sinks. Some of these studies were purely analytical or experimental; some analytical studies with experimental validation were also found. Some studies tried to incorporate fan power in the form of pressure drop in their optimization work. Some of these studies are described below as representations from each category of research

II. LITERATURE REVIEW

Goldberg (1984) constructed three air cooled, forced convection heat sinks and tested each one. Each heat sink had a different fin thickness, with the channel to fin width ratio restricted to unity, and the flow limited to the laminar regime. The air flow for each heat sink was adjusted to provide a rate of 30 L/min. As expected, the design with the largest pressure drop and smallest channel width yielded the smallest thermal resistance. Only experimental observation was provided in the literature.

Yokono et al. (1988) performed experimental studies of heat transfer from extruded heat sinks of short (height ≤ 5mm) fins exposed to variable fin spacing, height and air velocity. They suggested that the fin's heat dissipation capability was proportional to the supplied air velocity and heat dissipation was found large with an increase in fin height. The heat transfer coefficient for fins increased with an increase in fin interval and with a decrease in fin width, regardless of fin height. They proposed the following non-dimensional expressions to evaluate cooling performance for small fins.

$$Nu_{2s} = 0.33\left(Re_{2s}\frac{s}{B}\right)^{0.63}$$

where B and s are width and spacing of a heat sink.

They compared their work with the cooling performance in natural convection, but their work was limited to fins of short height, and the influence of larger fin height (H >5mm) was not examined in their research.

Azar et al. (1992) performed experimental studies on narrow channel (s = 1.1mm) heat sink with air flow arrangement of side-in-side-exit and top-in-side-exit [Fig. 2.2] and found no significant difference in heat sink performance. They performed some experiments with tip clearance and found that the use of heat sinks with tip clearance did not lead to a significant improvement in thermal performance. However, they did not provide any methodology to determine the heat sink thermal performance by experimental correlation or analytical modeling.

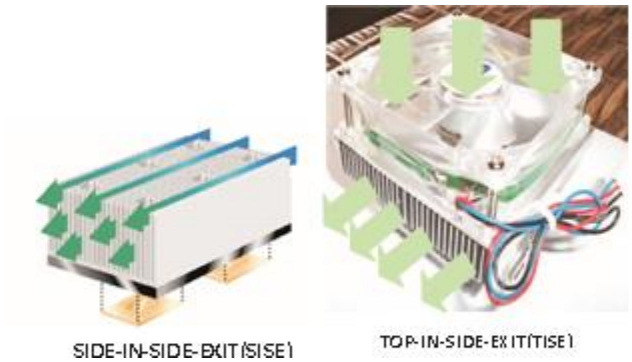


Figure 2.2: Different Types of Flow Arrangement in Heat Sinks

Holahan et al. (1996) presented an analytical model for calculating thermal and pressure drop performance in compact, laminar flow parallel plate heat sinks. They adapted laminar convective heat transfer coefficients from existing parallel plate correlations. They also developed a laminar pressure drop model which was applicable for a simple side-inlet-side-exit (SISE) flow pattern and a complex top-inlet-side-exit (TISE) flow pattern [Fig. 2.2]; the model was shown to handle arbitrary flow patterns. TISE model results were found in good agreement with experimental and CFD data. In that model, they also compared the thermal performance of side-inlet-side-exit (SISE) and top-inlet-side-exit (TISE) heat sinks and found that SISE showed better performance at higher pumping power (>2 watts) and TISE was better at lower pumping power. The model is limited to low Reynolds numbers ranging from 100 to 1000.

Copeland (2000) presented an analysis of simultaneously (hydraulic and thermal) developing flow using compact heat exchanger data fitted to Churchill-Usagi equations [14] for the performance calculation of a plate fin heat sink. They combined laminar fully developed theory to the developing flow (hydraulic and thermal) theory of Shah and London (1978). They also addressed the influence of spreading resistance in their model.

Teertstra et al. (1999) presented an analytical forced convection asymptotic model for the average heat transfer rate from a plate fin heat sink in a duct flow configuration for the full range of Reynolds number, from fully developed to developing flow. Given a uniform velocity at the heat sink inlet, the model can predict heat transfer from the fin array. No pressure drop model was proposed. Teertstra et al. validated this model with their experiments, and an excellent match (2.1% RMS error) was obtained.

Saini and Webb (2002) proposed a simple model based on developing laminar flow using curve fit data of local friction factor and Nusselt number from Shah and London (1978) and compared their model with experimental hydraulic and thermal performance of two plate fin heat sinks. The model under-predicts thermal resistance and pressure drop by nearly 8% and 20%, respectively.

Kim and Kim (2004) presented a compact modelling method based on the volume-averaging technique and its application to the analysis of fluid flow and heat transfer in straight fin heat sinks. They modelled a straight fin heat sink as a porous medium and developed volume averaged momentum and energy equations for developing flow in shrouded straight fin heat sinks. They determined the permeability, which is related to the viscous shear stress caused by frictional resistance of the fins, analytically from the Poiseuille flow between two infinite parallel plates under a constant heat flux. Using the same method, they also determined the interstitial heat transfer coefficient related to the heat transfer from the fins to the fluid. They compared the model data with the experimental data for pressure drop and heat transfer and showed that the porous medium approach accurately predicts the pressure drop and heat transfer characteristics of straight fin heat sinks.

Analytical and Numerical

Sathyamurthy et al. (1996) investigated inline and staggered parallel-plate arrays and obtained good agreement between their numerical results and experiments. Their results illustrated that the thermal performance of the staggered fin configuration was better than the planar fin configuration over the power and flow ranges examined. This enhanced thermal performance, however, was realized at the expense of an additional pressure drop.

Narasimhan et al. (2003) developed, demonstrated and validated a boundary layer methodology for the application of compact, porous block models for the hydrodynamic behavior of parallel plate heat sinks in laminar flow. They compared the porous block data with the results obtained from several hundred laminar-flow CFD simulations.Heat sink optimization for a fully shrouded case can take the form of parametric optimization,

Azar et al. (1992) reported a method of design optimization and presented contour plots showing the thermal performance of an air cooled narrow channel heat sink in terms of fin thickness and channel spacing parameters. The optimization method was presented assuming the pressure drop across the heat sink was known.

Knight et al. (1992) presented a optimization scheme for thermal design of air cooled finned heat sinks which gave the lowest thermal resistance under specified operating constraints. They examined the influence of number of fins (or fin pitch) on the thermal performance of a particular heat sink. They also validated their scheme with experimental data. Their choice of laminar and turbulent friction factor correlation for calculation of pressure drop in the optimization scheme over predicted the actual flow rate that ended up with lower thermal resistance in the predicted data than the experimentally observed data.

Bejan (1996) and Morega (1993) reported the optimal geometry of an array of fins that minimized the thermal resistance between the substrate and the flow forced through the fins. Staggered parallel-plate fin arrays were optimized in two steps, first the optimal fin thickness was selected and then the optimal size of fluid channel was determined. They also compared the minimum thermal resistance of staggered parallel-plate arrays and continuous fins. Furthermore, the dimensionless pressure gradient was plotted against Reynolds number.

Copeland (2000) investigated optimum dimensions of fin thickness and pitch for a variety of realistic operating conditions. According to them, fin thickness or pitch does not need to be fully optimized to achieve high performance, but the value of fin thickness or pitch must be near its corresponding optimum value of pitch or thickness.

Iyengar and Bar-Cohen (2000) considered heat sinks of fixed overall dimensions at specific points on fan curves (specific combinations of volume flow rate and pressure drop). Analyses were performed to maximize thermal conductance and conductance per unit mass. A small reduction in thermal performance permitted significant reduction in weight. In addition, aluminum, magnesium and copper were also analyzed. A variety of manufacturing techniques were discussed and dimensional constraints of each were used to perform comparative analyses.

Culham and Muzychka (2001) presented a procedure that allowed the simultaneous optimization of heat sink design parameters based on minimization of the entropy generation associated with heat transfer and fluid friction. The model demonstrated an unconstrained nonlinear procedure for obtaining optimum design conditions without resorting to parametric analysis using repeated iterations with a thermal analysis tool.

III. METHODOLOGY

3.0 Computational Fluid Dynamics :

Computational Fluid Dynamics or CFD as it is popularly known is used to generate flow simulations with the help of computers. CFD involves the solution of the governing laws of fluid dynamics numerically. The complex sets of partial differential equations are solved on in geometrical domain divided into small volumes, commonly known as a mesh (or grid).

CFD enables analysts to simulate and understand fluid flows without the help of instruments for measuring various flow variables at desired locations.

The development CFD analysis leads to a considerable reduction of investment and operating costs by optimized design, by an increased availability of the system. This can be achieved by an appropriate design of the manifold. CFD analysis helps to evaluate and avoid velocity and temperature peaks in manifold sections which are of special relevance regarding material stress and deposit formation. CFD analysis helps to predict the temperature, velocity and pressure distribution in the system.

Traditionally this has provided a cost effective alternative to full scale measurement. However, in the design of equipment that depends critically on the flow behavior, for example the aerodynamic design of an aircraft, full scale measurement as part of the design process is economically impractical. This situation has led to an increasing interest in the development of a numerical wind tunnel.

The development of more powerful computers has furthered the advances being made in the field of computational fluid dynamics. Consequently CFD is now the preferred means of testing alternative designs in many engineering companies before final, if any, experimental testing takes place.

3.1 ADVANTAGES:

- CFD allows numerical simulation of fluid flows, results for which are available for study even after the analysis is over. This is a big advantage over, say, wind tunnel testing where analysts have a shorter duration to perform flow measurements.
- CFD allows observation of flow properties without disturbing the flow itself, which is not always possible with conventional measuring instruments.

- CFD allows observation of flow properties at locations which may not be accessible to (or harmful for) measuring instruments. For example, inside a combustion chamber, or between urbine blades.
- CFD can be used as a qualitative tool for discarding (or narrowing down the choices between), various designs. Designers and analysts can study prototypes numerically, and then test by experimentation only those which show promise.

3.1.2 Applications:

Biomedical:

Flow modeling with computational fluid dynamics (CFD) software lets you visualize and predict physical phenomena related to the flow of any substance. It is widely used in medical, pharmaceutical, and biomedical applications to analyze.

Electronics:

Ansys provides a full spectrum of problem solving products for the electronics industry.

The Ansys flagship CFD software, FLUENT, as well as the electronics industry custom-designed Icepak suite, offer high-performance electronics cooling solutions covering a wide range of real life problems on any level.

Industrial:

To meet the vast fluid flow modeling needs of a broad spectrum of industries around the world, Fluent has been at the forefront of developing and driving computational fluid dynamics (CFD) for more than two decades. Diverse modeling capabilities allow Fluent's software products to tackle problems from most major industry sectors.

Environmental :

Protecting and improving the quality of our environment today requires innovative design solutions that establish compliance with ever-expanding and more stringent regulations. Flow modeling with Fluent's computational fluid dynamics (CFD) software helps you tackle your environmental flow problems in the most efficient and cost-effective way.

Civil :

Within the built environment, it is critical to assess a number of important building characteristics at the design stage, including the ability to improve the energy efficiency of a building, quantify solar radiation effects, analyze wind flow effects, study possible fire and smoke hazard scenarios, and predict occupant comfort.

NTU :

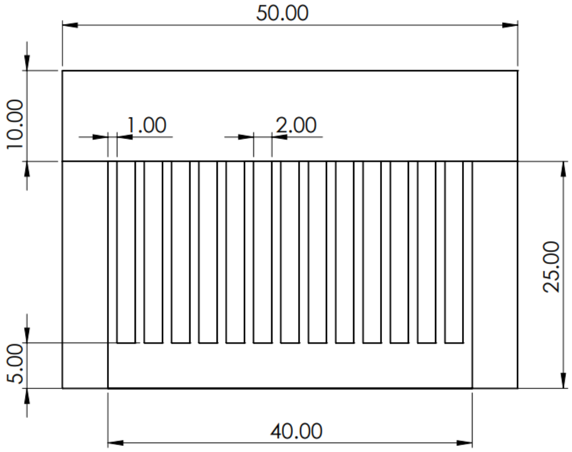
The NTU, or Number of Transfer Units, is a dimensionless parameter that relates the heat transfer convective resistances to the coolant flow heat capacity. While the details are beyond the scope of this short column, a typical heat sink (or cold plate) can be described with the following equations (assuming that the simplifying assumption of one surface temperature is reasonable).

1 Actual heat transfer = Ccool*(Tcool-out – Tcool-in);
(Ccool is the mass flow times the heat capacity for the coolant)
2 Maximum possible heat transfer = Ccool*(Tsurf – Tcool-in)
3 Effectiveness = E = (Tcool-out – Tcool-in)/ (Tsurf – Tcool-in)
4 Effectiveness = 1 – exp (-NTU), where NTU = hA/Ccool

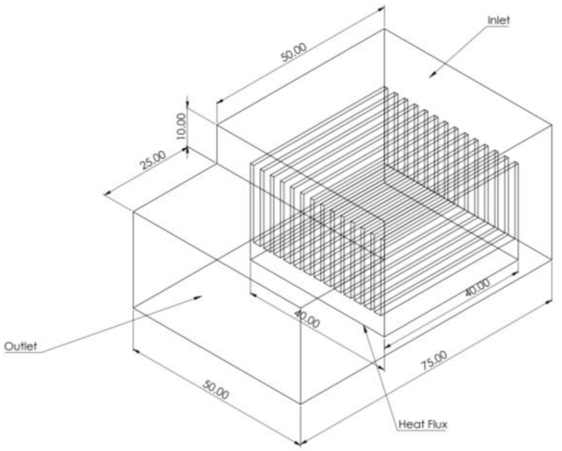
One way to increase the NTU term is to decrease the coolant heat capacity but while our effectiveness increased, the resulting temperature for the surface may not be acceptable. The other way is to increase the hA term which means either larger area or a higher effective heat transfer coefficient. The engineering challenge is to minimize the decrease in effectiveness as coolant flow rates increase. Note that the limit is when the surface temperature and the coolant exit are at the same temperature, the communication of ideas to non-specialist.

3.11 **CAD design and Type of Fins**

For this project three types of designs have been done Flat Fin Heat Sink , Wavy fin heat sink , Staggered Fin Heat Sink.

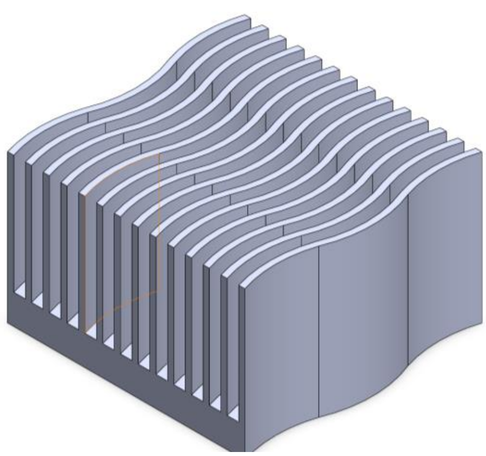


(a)

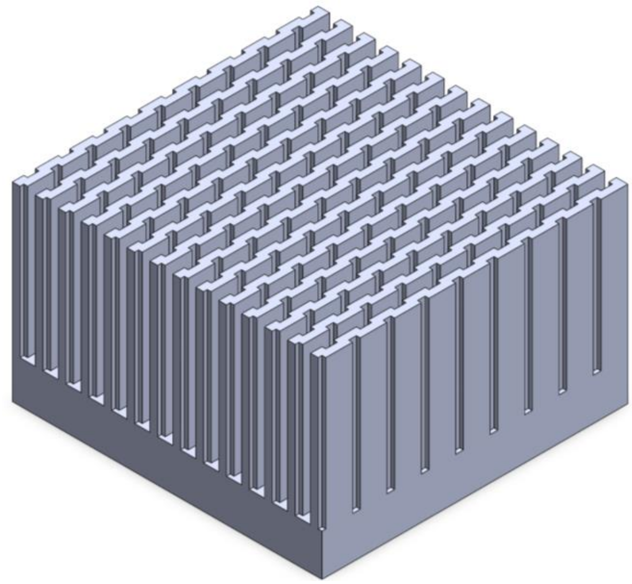


(b)

Fig 3.2 Heat sink and Computational domain Design Details



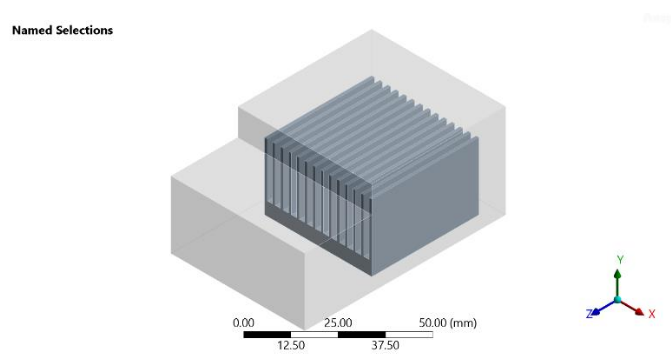
(a)



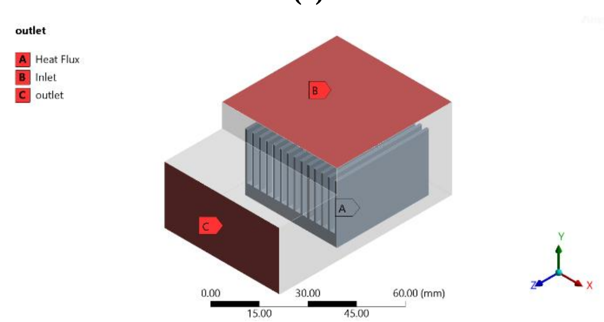
(b)

Fig 3.3 3d model used for Simulation (a) Wavy fin Heat sink (b) Staggered fin heat sink.

3.12 Computational Domain and Heat sink



(a)



(b)

Fig 3.4 Computational Domain and the Heat sink

The above figure shows the computational domain of the heat sink where the volume around a solid model is created to allow the air flow at different velocities a fluid solid interface is created by Boolean in the model. To allow heat transfer between surrounding air and the heated Fins. Where the inlet and outlet is shown in fig 3.2 b heat flux to the solid model is given @ bottom surface of the heat sink

IV. RESULTS AND DISCUSSION

4.0 Introduction

In this Chapter the Discussion of Cfd analysis Results of the Flat fin Heat sink , Wavy Fin Heat sink and Staggered Fin heat sink is discussed.

ANSYS Fluent

ANSYS Fluent software is the most-powerful computational fluid dynamics (CFD) tool available, empowering you to go further and faster as you optimize your product's performance. Fluent includes well-validated physical modeling capabilities to deliver fast, accurate results across the widest range of CFD and Multiphysics applications.

Turbulence modeling is the construction and use of a model to predict the effects of turbulence. A turbulent fluid flow has features on many different length scales, which all interact with each other. A common approach is to average the governing equations of the flow, in order to focus on large-scale and non-fluctuating features of the flow. However, the effects of the small scales and fluctuating parts must be modelled.[1]

Boundary Condition and Setup

Solver	Pressure Based solver
Viscous method	K-epsilon
Solution Method	SIMPLE
Inlet	Velocity inlet
outlet	Pressure Outlet
Velocity inlet	7000 to 70000
Inlet Temperature	296.6K
Heat Flux	18750 W/m2

Case 1 Flat Fin Heat sink @ Re=70000

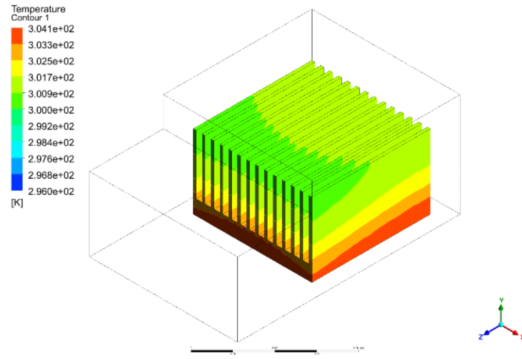


Fig 4.1 Temperature Distribution of Flat fin Heat sink

The above figure represents the temperature distribution of flat fin Heat sink where the maximum temperature of 304.1 K is seen near the heat flux wall due to air flow through the fins the temperature near the inlet side of the fin s 301K approximately overall temperature distribution is represented in different color codes where the red colored region represents high temperature and the blue colored region represents the low temperature as shown in the left side of the picture.

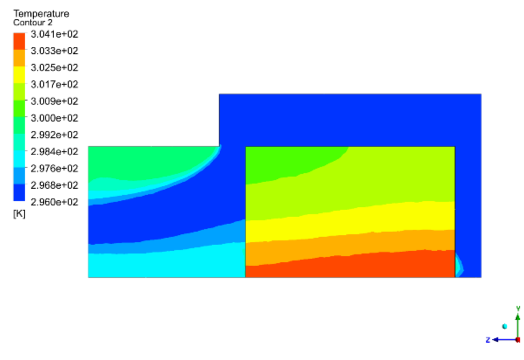


Fig 4.2 Temperature distribution of Flat fin heat sink along the length of the air flow.

The above figure represents the temperature distribution along the air flow path it is observed that the maximum temperature in air flow is seen in the region of heat flux where the distribution of temperature is shown in figure 4.2 red colored region is the heat sink temperature and near blue regions are the air temperature where the inlet temperature of air is around 296 K.

Table 1 Flat Fin Heat sink Results

Straght Fin				
S.No	Re	Air Temperature	Fin Temperature	Fin effectiveness
1	7000	299.761	312.513	0.227759947
2	17000	298.595	307.461	0.226420033
3	35000	298.021	304.27	0.244377267
4	55000	297.833	302.775	0.270553506
5	77000	297.741	301.827	0.298781534

Case 2 Wavy Fin Heat sink @ Re=70000

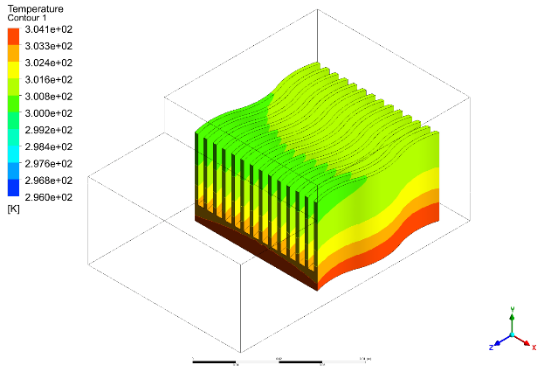


Fig 4.3 Temperature Distribution of Wavy fin Heat sink

The above figure represents the temperature distribution of Wavy fin Heat sink where the maximum temperature of 304.1 K is seen near the heat flux wall due to air flow through the fins the temperature near the inlet side of the fin s 300.8 K approximately overall temperature distribution is represented in different color codes where the red colored region represents high temperature and the blue colored region represents the low temperature as shown in the left side of the picture.

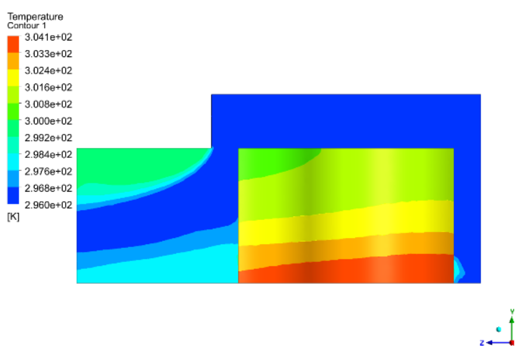


Fig 4.4 Temperature distribution of Wavy fin heat sink along the length of the air flow.

The above figure represents the temperature distribution along the air flow path it is observed that the maximum temperature in air flow is seen in the region of heat flux where the distribution of temperature is shown in figure 4.4 red colored region is the heat sink temperature and near blue regions are the air temperature where the inlet temperature of air is around 296 K.

.

Table 2 Wavy Fin Heat sink Results

Wavy Fin				
S.No	Re	Air Temperature	Fin Temperature	Fin effectiveness
1	7000	299.768	312.326	0.230797501
2	17000	298.61	307.466	0.227629513
3	35000	298.04	304.275	0.24652568
4	55000	297.853	302.767	0.273828875
5	77000	297.759	301.809	0.302805991

Case 3 Staggered Heat sink @ Re=70000

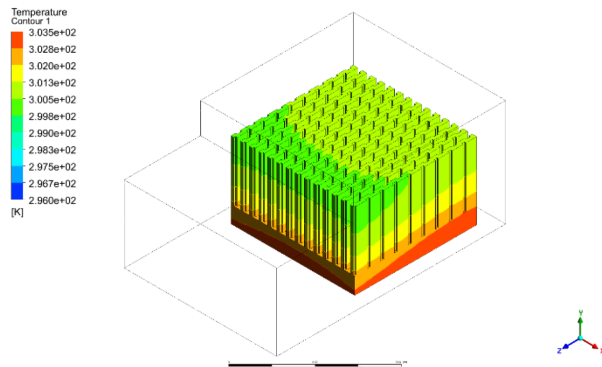


Fig 4.5 Temperature Distribution of Wavy fin Heat sink

The above figure represents the temperature distribution of staggered fin Heat sink where the maximum temperature of 303.5 K is seen near the heat flux wall due to air flow through the fins the temperature near the inlet side of the fin s 299.8 K approximately overall temperature distribution is represented in different color codes where the red colored region represents high temperature and the blue colored region represents the low temperature as shown in the left side of the picture.

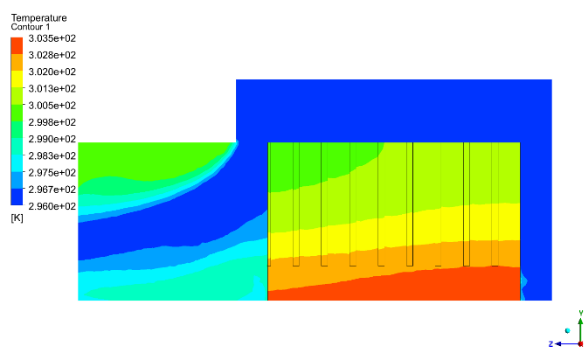


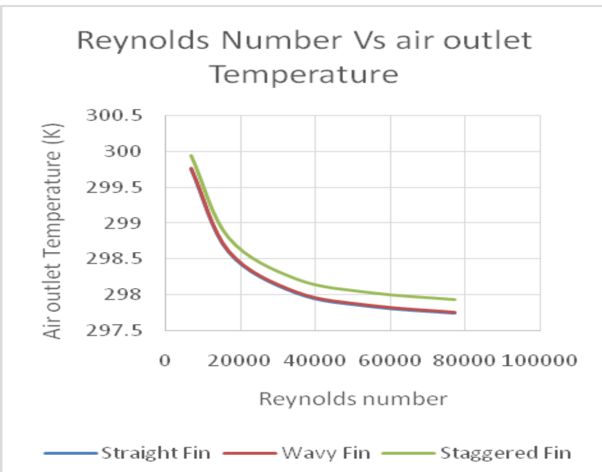
Fig 4.6 Temperature distribution of Staggered fin heat sink along the length of the air flow.

The above figure represents the temperature distribution along the air flow path it is observed that the maximum temperature in air flow is seen in the region of heat flux where the distribution of temperature is shown in figure 4.4 red colored region is the heat sink temperature and near blue regions are the air temperature where the inlet temperature of air is around 296 K.

Table 3 Wavy Fin Heat sink Results

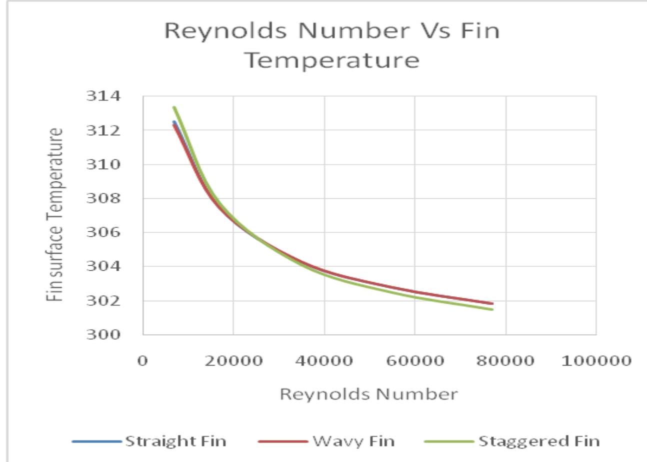
Staggered Fin				
S.No	Re	Air Temperature	Fin Temperature	Fin effectiveness
1	7000	299.944	313.374	0.227005871
2	17000	298.79	307.71	0.238257899
3	35000	298.214	304.098	0.27340084
4	55000	298.022	302.465	0.312761021
5	77000	297.927	301.459	0.352995054

Plots



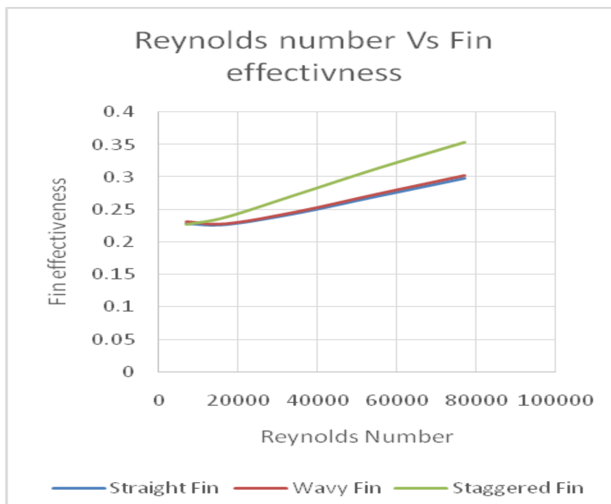
Plot 1 Reynolds number Vs air outlet temperature.

The above plots represents comparison between air outlet temperature when constant heat flux is given to the heat sink it is observed from the plot that staggered fin is dissipating more heat to the air than other types of fins increase in Reynolds number decrease in air outlet temperature.



Plot 2 Reynolds number Vs Fin Temperature.

The above plots represents comparison between Fin temperature when constant heat flux is given to the heat sink it is observed from the plot that staggered fin is dissipating more heat to the air than other types of fins increase in Reynolds number decrease in Fin temperature.



Plot 3 Reynolds number Vs Fin effectiveness.

The above plots represent comparison between Fin effectiveness when constant heat flux is given to the heat sink it is observed from the plot that staggered fin has more effectivness than other flat and wavy types of fins.

V. CONCLUSION

5.0 Conclusion

The simulation is done for 3 different heat sink models where details of this models is discussed in methodology the simulation is done for Reynolds number ranging from 7000 to 70000 a turbulent k-epsilon model is used from the results and discussions it is observed that 18% increase in fin effectiveness when staggered Fin is used.

REFERENCES

[1] Airflow Measurement Systems, "Airflow Systems," *http://www.fantester.com*

[2] Antonetti, V.W., "On the Use of Metallic Coatings to Enhance Thermal Contact Conductance," Ph.D. Dissertation, Department of Mechanical Engineering, University of Waterloo, Waterloo, ON, Canada, 1983.

[3] Ashiwake, N., Nakayama, W., Daikoku, T., and Kobayashi, F., "Forced Convection Heat Transfer from LSI Packages in an Air Cooled Wiring Card Array," *Heat Transfer in Electronic Equipment-193,* ASME HTD 28, 1983, pp. 35-42.

[4] Azar, K., McLeod, R.S., and Caron, R.E., "Narrow Channel Heat Sink for Cooling of High Powered Electronics Components," *Proceedings of the Eighth Annual IEEE Semiconductor Thermal Measurement and Management Symposium (SEMI-THERM VIII),* Austin, TX, 1992, pp. 12-19.

[5] Azar, K., "The History of Power Dissipation," *http://www.electronicssooling.com /html /2000 jan a2.html,* Southborough, MA: Electronics Cooling, 2000.

[6] Azar, K., "Managing Power Requirements in the Electronics Industry," *http: //www.electronics-cooling.com/html/2000 dec a1.html,* Southborough,MA: Electronics Cooling, 2000.

[7] Bar-Cohen, A., and Iyengar, M., "Least-Energy Optimization of Air-cooled Heat Sinks for Sustainable Development," *Proceedings of the Sixth Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems (ITHERM'98),* Seattle, WA, 1998, pp.
295-302.

[8] Bejan, A., *Entropy Generation Through Heat and Fluid Flow,* New York: Wiley, 1982.

[9] Bejan, A., and Morega, A.M., "Optimal Arrays of Pin Fins and Plate Fins in Laminar Forced Convection," *Journal of Heat Transfer,* Vol. 115, 1993, pp. 75-81.
136

[10] Bejan, A., Tsataronis, G., and Moran, K., *Thermal Design and Optimization,* New York: Wiley, 1996.

[11] Bejan, A., *Entropy Generation Minimization,* Boca Raton, FL: CRC Press, 1996.

[12] Bejan, A., *Advanced Engineering Thermodynamics,* New York: Wiley, 1997.

[13] Butterbaugh, M.A., and Kang, S.S., "Effect of Airflow Bypass on the Performance of Heat Sinks in Electronic Cooling," *ASME Advances in Electronic Packaging,* Vol. EEP-10-2, 1995, pp. 843-848.

[14] Churchil, S.W. and Usagi, R., "A General Expression for the Correlation of Rates of Transfer and Other Phenomena," *AIChE Journal,* Vol. 18, No. 6, 1972, pp. 1121 - 1128.

[15] Coetzer, C.B., and Visser, J.A., "Compact Modeling of Forced Flow in Longitudinal Fin Heat Sinks with Tip Bypass," *ASME Journal of Electronic Packaging,* Vol. 125, 2003, pp. 319-324.

[16] Copeland, D., "Optimization of Parallel Plate Heat Sinks for Forced Convection", *Proceedings of the Sixteenth Annual IEEE Symposium on Semiconductor Thermal Measurement and Management (SEMI-THERM XVI),* San Jose, CA, 2000, pp. 266-272.

[17] Culham, J.R., and Muzychka, Y.M., "Optimization of Plate Fin Heat Sinks Using Entropy Generation Minimization," *IEEE Transactions on Components and Packaging Technologies,* Vol. 24, No. 2, 2001, pp. 16-25.

[18] Flow Kinetics LLC., "Using a Pitot Static Tube for Velocity and Flow Rate Measurement," *http://www.flowmeterdirectory.com/flowmeter artc/flowmeter artc 02111201 .html*

[19] Goldberg, N., "Narrow Channel Forced Air Heat Sink," *IEEE Transactions on Components, Hybrids, and Manufacturing Technology,* Vol. CHMT-7, No. 1, 1984, pp. 154-159.

[20] Hirata, M., Kakita, Y., Yada, Y., Hirose, Y., Morikawa, T., and Enomot, H., "Temperature Distribution of Finned Integrated Circuits," *FUJITSU Scientific & Technical Journal*, December 1970.

[21] Holahan, M.F., Kang, S.S., and Bar-Cohen, A., "A Flow Stream Based Analytical Model for Design of Parallel Plate Heat Sinks," *ASME Proceedings of the* 31st *National Heat Transfer Conference (HTD-Vol. 329),* Houston, TX, Vol. 7, 1996, pp. 63-71.

[22] Incropera, P.F., and Dewitt, D.P., *Fundamentals of Heat and Mass Transfer,* New York: Wiley, 2002.

[23] Intel Corporation, "Moore's Law," *http://www.intel.com/research/silicon/ mooreslaw.htm,* Santa Clara, CA.

[24] Iwasaki, H., Sasaki, T., and Ishizuka, M., "Cooling Performance of Plate Fins for Multichip Modules," *IEEE Transactions on Components, Packaging and Manufacturing Technology Part A,* Vol. 18, No. 3, 1995, pp. 592-595.

[25] Iyengar, M., and Bar-Cohen, A., "Design for Manufacturability of SISE Parallel Plate Forced Convection Heat Sinks," *IEEE Transactions on Components and Packaging Technologies,* Vol. 24, No. 2, 2001, pp. 150-158.

[26] Jonsson, H., and Moshfegh, B., "Modeling of the Thermal and Hydraulic Performance of Plate Fin, Strip Fin, and Pin Fin Heat Sinks- Influence of Flow Bypass," *IEEE Transactions on Components and Packaging Technologies,* Vol. 24, No. 2, 2001, pp. 142-149.

[27] Kays, W.M., and London, A.L., *Compact Heat Exchangers,* New York: McGraw-Hill, 1984.

[28] Khan, W.A., Culham, J.R., and Yovanovich, M.M., "Performance of Shrouded Pin-Fin Heat Sinks in Forced Convection Cooling," *Proceedings of the 38th AIAA Themophysics Conference,* Toronto, Canada, 2005.

[29] Kim, D., and Kim, J.S., "Compact Modeling of Fluid Flow and Heat Transfer in Straight Fin Heat Sinks," *ASME Journal of Electronic Packaging,* Vol. 126, 2004, pp. 247-255.

[30] Knight, R.W., Goodling, J.S., and Gross, B.E., "Optimum Thermal Design of Air Cooled Forced Convection Finned Heat Sinks- Experimental Verification," *IEEE Transactions on Components, Hybrids, and Manufacturing Technology,* Vol. 15, No. 5, 1992, pp. 754-760.

[31] Lau, K.S., and Mahajan, R.L., "Effects of Tip Clearance and Fin Density on the Performance of Heat Sinks for VLSI Packages," *IEEE Transactions on Components, Hybrids, and Manufacturing Technology,* Vol. 12, No. 4, 1989, pp. 757-765.

[32] Lee, R.S., Huang, H.C., and Chen, W.Y., "A Thermal Characteristic Study of Extruded-Type Heat Sinks in Considering Air Flow Bypass Phenomena," *Proceedings of the Sixth Annual IEEE Symposium on Semiconductor Thermal and Temperature Measurement (SEMI-THERM VI),* Phoenix, AZ, 1990, pp. 95-102.

[33] Lee, S., "Optimum Design and Selection of Heat Sinks," *IEEE Transactions on Components, Packaging and Manufacturing Technologies,* Vol. 18, 1995, pp. 812-817.

[34] Lee, S., Song, S., Au, V., and Moran, K. P., "Constriction/Spreading Resistance Model for Electronics Packaging," *ASME/JSME Thermal Engineering Conference,* Vol.4, 1995, pp. 199206.

[35] Leonard, W., Teertstra, P., and Culham, J.R., "Characterization of Heat Sink Flow Bypass in Plate Fin Heat Sinks," *Proceedings of ASME International Mechanical Engineering Congress & Exposition (IMECE2002-39556),* New Orleans, Louisiana, 2002.

[36] Matsushima, H., Yanagida, T., and Kondo, Y., "Algorithm for Prediction of the Thermal Resistance of Finned LSI Packages Mounted on a Circuit Board," *Heat Transfer - Japanese Research,* Vol. 21, No. 5, 1992, pp. 504-517.

[37] Min, J.Y., Jang, S.P., and Kim, S.J., "Effect of Tip Clearance on the Cooling Performance of a Microchannel Heat Sink," *International Journal of Heat and Mass Transfer,* Vol. 47, 2004, pp. 1099-1103.

[38] Muzychka, Y.S., and Yovanovich, M.M., "Modeling Friction Factors in Non-circular Ducts for Developing Laminar Flow," *Proceedings of the Second AIAA Theoretical Fluid Mech. Meeting,* Albuquerque, NM, 1998.

[39] Narasimhan, S., and Bar-Cohen, A., "Flow and Pressure Field Characteristics in the Porous Block Compact Modelling of Parallel Plate Heat Sinks," *IEEE Transactions on Components, Packaging and Manufacturing Technologies,* Vol. 26, No. 1, 2003, pp. 147-157.

[40] Obinelo, I. F., "Characterization of Thermal Performance of Longitudinal Fin Heat Sinks for System Level Modeling Using CFD Methods," *Proceedings of the ASME International, Intersociety and Photonic Packaging Conference and Exhibition,* Hawaii, 1997.

[41] Omega Engineering, "Highly Accurate, Low-pressure Laboratory Transducer," *http://www.omega.com/Pressure/pdf/PX653.pdf*

[42] Prasher, R.S., "Surface Chemistry and Characteristics Based Model for the Thermal Contact Resistance of Fluidic Interstitial Thermal Interface Materials," *ASME Journal of Heat Transfer,* Vol. 123, 2001, pp. 969-975.