CFD Analysis of Heat Sink Using Trapezoidal Fins And Comparing With Rectangular Flat Plate

Md Imaaduddin Akrama Sagar[1], M. R. Nagaraj[2]
[1]Dept of Mechanical Engineering
[2]Associate Professor, Dept of Mechanical Engineering
[1, 2] P.D.A College of Engineering, Gulbarga, Karnataka.

Abstract- *The development of the digital computers and its usage day by day is rapidly increasing. But the reliability of electronic components is getting affected critically by the temperature at which the junction operates. As operating power and speed increases, and as the designers are forced to reduce overall systems dimensions, the problems of extracting heat and controlling temperature becomes crucial. In the last decade or so, CFD simulations have become more and more widely used in the studies of electronic cooling. In this paper the CFD simulation and Thermal analysis is carried out with a commercial package provided by ANSYS-FLUENT. The geometric parameters and design of heat sink for improving the thermal performance is experimented. The design is able to cool the chassis with heat sink attached to the CPU is adequate to cool the system. This paper considers the rectangular plate heat sink design with aluminium base plate and the same is done for the Trapezoidal Fin also.*

Keywords- Computational Fluid Dynamics (CFD), Central Processing Unit (CPU), Cooling, Trapezoidal Fins, Thermal Analysis.

I. INTRODUCTION

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 components at a satisfactory level.

1.1 Background

Heat removal from Integrated Circuits (ICs) now ranks among the major technical problems that needs to be solved to achieve higher power densities. For years, the IC industry has been trying to maintain the pace of Moore's prediction, the projection that the number of transistors on integrated circuits would double every eighteen months. Figure 1.1 shows the prediction of the growth of semiconductor transistor density, observed by Intel founder Dr. Gordon Moore. This remarkable rate of advancement has resulted in smaller feature sizes and improved manufacturing techniques, which allows for more efficient circuit designs and materials, which result in better circuit performance.

Figure 1.1: Time-line plot of transistor counts on Intel processors based on Moore's prediction [23]

As semiconductors become more complex and new milestones in transistor size and performance are achieved, power consumption and heat dissipation have emerged as limiting factors to the continued pace of new chip designs and manufacturing techniques. There are hundreds of millions, and even billions of smaller and faster transistors which are packed on to a processor, a single piece of silicon the size of a thumbnail. The power consumption and dissipation of heat generated in the processor core become significant technical challenges to the achievement of Moore's prediction. Power and heat have become the biggest technical issue of the decade while the semiconductor industry continues to strive to improve transistor speed and power efficiency.

Forced air cooling through the use of extended surfaces [Fig. 1.3] is being used as a viable technique for cooling microelectronic devices due to its inherent simplicity and cost effectiveness. Designs incorporating such surfaces typically take the form of finned heat sinks. In microelectronic applications, heat sinks are directly mounted on the cases that enclose microelectronic devices to provide extra surface area for heat transfer from the device to the cooling fluid [Fig. 1.2]. However, with the increase in component density within electronic enclosures, combined with ongoing increases in individual component power dissipation, it is apparent that the use of ducting for individual component cooling is not practical. If forced-air heat sinks are going to continue to be an effective means of cooling to these devices, the thermal engineer must examine more closely the relationship between thermal performance and fluid flow in and around the heat sink

Figure 1.2: Application of heat sinks in an electronic enclosure [5]

.

Heat sinks use a variety of fin arrangements to provide the extra surface area for heat transfer. The presence of closely spaced fins also creates an extra resistance for flow through the heat sinks. In many practical applications, heat sinks are mounted on circuit boards such that there are significant clearances around them [Fig. 1.2]. Because of the higher resistance to flow through a heat sink, the cooling fluid tends to bypass the heat sink and flow through the clearance zones. Since the temperature rise across the heat sink and the heat transfer coefficient depends on the velocity of the flow through the heat sink, the bypassing of the flow adversely affects the heat transfer performance of a heat sink. When the air velocity between the fins of such a heat sink can be well approximated, the thermal engineer can effectively predict the overall thermal resistance and viscous dissipation of the system. Approximating the fin velocity based on the upstream flow rate in the enclosure is often difficult, except in the case of fully shrouded heat sinks. All other ducting scenarios require consideration of flow bypass. Therefore, accurate system-specific analysis of the effect of bypass on the thermal performance of heat sinks is important for satisfying current thermal requirements as well as providing the capability for designing thermal solution for future generation of electronic hardware.

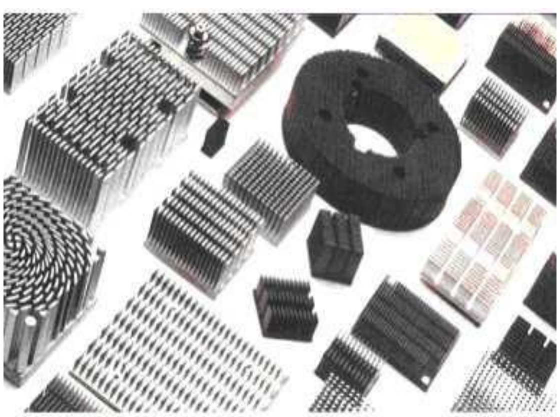


Figure 1.3: Different types of extended surfaces or heat sinks [6]

1.2 Problem Statement

The heat produced in an electronic device is conducted into the substrate and then transferred by some combination of thermal conduction, convection and radiation to the outer surface through numerous components such as thermal interface materials (TIMs), heat sinks, air etc. [Fig. 1.4]. Along this flow path, heat encounters various thermal resistances that cause a temperature rise inside the package. Therefore, careful design of heat sinks is extremely important in order to maintain operating temperatures at or below recommended limits.

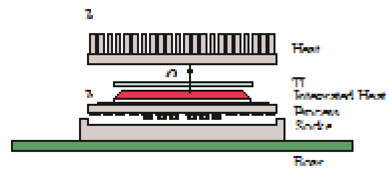


Figure 1.4: Microelectronics package with heat sink

When there is no clearance around the heat sink, the flow velocity through the fins is known from the duct flow mean approach velocity by applying conservation of mass. The use of such a duct is known to minimize the bypass of air around the heat sink but results in a considerable pressure drop penalty. A higher pressure drops and subsequently the higher pumping power required to push the air through the heat sink restricts the use of high performance heat sinks in industry. Though a higher pumping power may be achieved by using a high power fan, it is often difficult to have the necessary ducting in an electronic enclosure; therefore, use of ducting for individual component cooling is not practical. In addition, noise constraints associated with many electronics applications restrict approach flow velocities to a range of 8*m/s* or less.

Typically, the heat sink on an electronic module occupies only a fraction of the cross-section of the air flow channel of the card as shown in Fig. 1.7. The air flow areas that exist around the heat sink allow some of the oncoming air flow to bypass the heat sink.

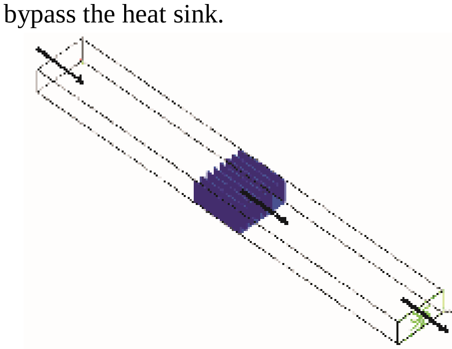


Figure 2.1: Fully shrouded heat sink

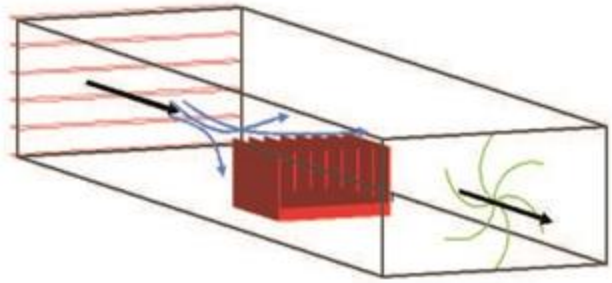


Figure 2.2: Heat sink with bypass

II. LITERATURE REVIEW

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.

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 ≤ 5*mm*) 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 >5*mm*) 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.

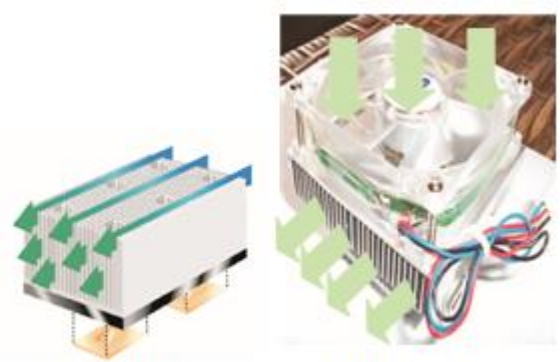


Figure 2.3: Different types of flow arrangement in heat sinks

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.

Least Energy Optimization

Bar-Cohen and Iyengar (2003) presented a methodology for the least-material and least-energy design of air-cooled heat sinks for the sustainable thermal management of electronic components. They tried to show that the energy invested in the formation and fabrication of such heat sinks can far exceed the pumping power dissipated by commonly used heat sinks. They also proposed a thermal Coefficient of Performance (COP_T) relating the cooling capability of a heat sink to the energy invested in its fabrication/formation (thermal energy) and operation (fluid or pumping energy). They used the technique of COP_T to determine the degree of sustainability of a specific heat sink design, and compared it to the entropy generation minimization methodology (EGM).Though the issue of bypass was first addressed in the late seventies; there are only a limited number of studies found in open literature. Most of these studies addressed the issue of tip clearance, only a few researchers addressed the issue of both tip and lateral clearance together.

Experimental, analytical and numerical studies of a heat sink with tip clearance are found in the literature.

Sata et al. (1997) carried out a numerical analysis for the flow and temperature fields around a plate fin array. Based on the knowledge of flow and thermal phenomena around the fin array, they proposed a new technique for predicting the cooling performance of the fins, in which inter-fin velocity is estimated by modelling the energy balances in the flow field around the fin array and between fins under the condition of constant pressure at its downstream edge. Their technique could predict inter-fin velocity with an error level below 20% and the cooling performance with an error level below 30% under practical conditions.

Optimization of the dimensions and performance of heat sinks with bypass are found only in the form of parametric optimization models. No literature is found to optimize the design of a heat sink under variable bypass conditions using multi-variable optimization or Entropy Generation Minimization or Least Energy Optimization techniques.

Parametric Optimization

Wirtz et al. (1994), during his experimental work, devised a set of expressions for determining the fin density for different fin geometries and flow conditions. He did not include the influence of bypass and spreading resistance in his optimization work.

Lee (1995) tried to optimize the performance of a plate fin heat sink by studying the parametric behavior of number of fins, fin length and approach velocity under fixed bypass conditions. The effect of bypass was not shown, and the effect of spreading resistance was not considered during the optimization.

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 turbine 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,

3.2 FIN INTRODUCTION

A fin is a surface used to produce lift and thrust or to steer while traveling in water, air, or other fluid media. Fins improve heat transfer in two ways. One way is by creating turbulent flow through fin geometry, which reduces the thermal resistance through the nearly stagnant film that forms when a fluid flows parallel to a solid surface. A second way is by increasing the fin density, which increases the heat transfer area that comes in contact with the fluid.

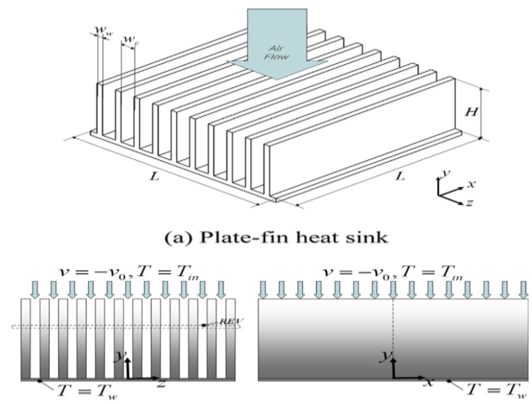
The plate fin heat exchanger is one of the most efficient designs used to transfer heat from one fluid to another. The best way to increase the efficiency of one of these devices is to alter the inner fin layout of the design. The fins are very important to the overall design because they add a large amount of surface area of contact between the fluid and the plate, which supplies heat. Altering the fin's geometry and other traits can drastically change the way a heat exchanger performs. Changing certain properties of a fin can have diminishing returns while others can have very beneficial effects. Effective ways to change a fin would be to change its height, length, the angle at which it is oriented with respect to the flow, and the amount of fins per unit length.

The fin angle is also an important factor to consider. Introducing different fin angles will create more vortices in the flow but will also create a lot more turbulence.

Vortices are good to have because the mixing swirling fluid is more efficient at transferring heat than a laminar boundary region attached to the surface. As the fluid hits the fin and passes over it, the wake region behind the fin sees almost no fluid velocity or mixing and has a large pressure drop.

3.3 PLATE FIN HEAT SINK:

The problem under consideration is an impinging flow through a plate-fin heat sink as shown in Fig. 1(a) and (b). The bottom surface is kept constant at a high temperature. Air impinges on the heat sink along the y-axis and then flows parallel to x-axis .Air, employed as a coolant, passes through the heat sink, thus removing the heat generated by the component attached at the bottom of the heat sink substrate. In analyzing this problem, the flow is assumed to be steady and laminar. In addition, all thermal properties are evaluated at the film temperature of fluid. In addition, it is assumed that the aspect ratio of the channel is higher than 1 and the solid conductivity is higher than the fluid conductivity.

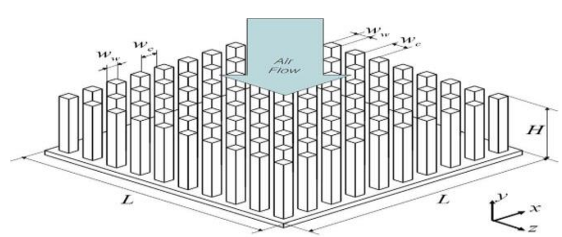


(a) Plate-fin heat sink

(b) Physical model

3.4 PIN FIN HEAT SINK:

The fluid flow in the pin-fin heat sink is axisymmetric in nature rather than two-dimensional. In the present study, the pin-fin heat sink is modeled as an equivalent porous cylinder which has the same porosity, wetted surface, and base area. The top view for the equivalent porous cylinder is shown in Fig. 2(b).



(a) Pin-fin heat sink

In early days the rectangular ducts are made up with straight array of plates. This type of rectangular duct has low surface conduct with air flowing inside the rectangular duct. This results in the reduced total pressure drop across the rectangular duct and also reduced heat transfer rate. Due to straight array of plates air passing in the straight direction there is no mixing between the airs flowing across the rectangular duct and there is no secondary flow formation.CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements.

Pre-processor
Solver
Post-processor

We briefly examine the function of each of these elements within the context of a CFD code.

3.5 PRE-PROCESSOR:

'Pre processing consists of the input of a flow problem by means of an operator friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activity at the pre-processing stage involves,

Definition of the geometry of the region of the interest: the computational domain.

Grid generation: the sub-division of the domain into a number of the smaller, non-overlapping sub-domains: a grid (or mesh) of cells (or control volumes or elements).

Selection of the physical and chemical phenomena that need to be modeled.

Definition of fluid properties.

Specification of appropriate boundary conditions at cells, which coincide with or touch the domain boundary.

The solution to a flow problem (velocity, pressure, temperature etc) is defined at nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells the better the solution accuracy. Both the accuracy of a solution and its cost in terms of necessary computer hardware and calculation time are dependent on the fineness of the grid. Optimal meshes are often non-uniform: finer in areas where large variations occur from point and coarser in regions with relatively little change. Efforts are under way to develop CFD codes with an adaptive meshing capability. Ultimately such programs will automatically refine the grid in area of rapid variations. A substantial amount of basic development work still needs to be done before these techniques are robust enough to be incorporated into commercial CFD codes. At present it is still up to the skills of the CFD user to design a grid that is a suitable compromise between desired accuracy and solution cost.

3.6 SOLVER:

There are three distinct streams of numerical solution techniques: finite difference, finite element and spectral methods. In outline the numerical methods that form the basis of the solver perform the following steps,

Approximation of the unknown flow variables by means of simple functions.

Diseretizations by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.

Solution of the algebraic equations.

The main differences between the three separate streams are associated with the way in which the flow variables are approximated and with the diseretizations processes.

3.7 Finite Difference Method:

Finite difference methods describe the unknown's of the flow problem by means of point samples at the node points of a grid co-ordinate lines. Truncated Taylor series expansions are often used to generate finite difference approximations of derivatives of F in terms of the point samples O at each grid point and its immediate neighbors. Those derivatives appearing in the governing equations are replaced by finite differences yielding an algebraic equation for the values of at each grid point.

3.8 Finite element method:

Finite element methods use simple piecewise functions (e.g. linear or quadratic) valid on elements to describe the local variations of unknown flow variables. The governing equation is precisely satisfied by the exact solution. If the piecewise approximating functions for are minimized in some sense by multiplying them by equations for the unknown coefficients of the approximating functions. The theory of finite elements has developed initially for structural analysis

3.9 Finite volume method:

The finite volume method was originally developed as a special finite difference formulation. It is central to four of the five main commercially available CFD codes:

PHOENICS, FLUENT, FLOW3D and STAR-CD. The numerical algorithm consists of the following steps,

Formal integration of the governing equations of fluid flow over all the (finite) control volumes of the solution domain.

Diseretizations involves the substitution of a variety of finite difference type approximations for the terms in the integrated equation representing flow processes such as convection, diffusion and sources. This converts the integral equations into a system of algebraic equations.

Solution of the algebraic equations by an iterative method.

The first step, the control volume integration, distinguishes the finite volume method from all other techniques. The resulting statements express the (exact) conversation of relevant properties for each finite size cell. This clear relationship between the numerical algorithm and the underlying physical conservation principle forms one of the main attractions of the finite volume method and makes its concepts much simple to understand by engineers than finite volume method and spectral methods. The conservation of a general flow variable, for example a velocity component or enthalpy, within a finite control volume can be expressed as a balance between the various processes tending to increase or decrease it.

CFD codes contain diseretizations techniques suitable for the treatment of the key transport phenomena, convection (transport due to fluid flow) and diffusion (transport due to variations of O from point to point) as well as for the source terms (associated with the creation of destruction of O) and the rate of change with respect to time. The underlying physical phenomena are complex and non-linear so an iterative solution approach is required. The most popular solution procedures are the TDMA line-by-line solver of the algebraic equations and the simple algorithm to ensure correct linkage between pressure and velocity.

3.10 POST PROCESSOR:

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. Owing to the increased popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualization tools. These include,

Domain geometry and grid display
Vector plots
Line and shaded contour plots
2D and 3D surface plots
Particle tracking
View manipulation (translation, rotation, scaling etc)
Color postscript output

More recently these facilities may also include animation for dynamic result display and in addition to graphics all codes produce trusty alphanumeric output and have data export facilities for further manipulation external to the code. As in many other branches of CAED, the graphics output capabilities of CFD codes have revolutionized the communication of ideas to non-specialist.

Computational domain and heat sink

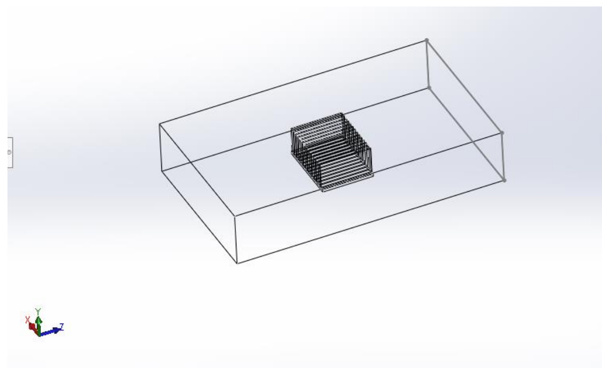


Fig 3.2 Computational domain and the heat sink

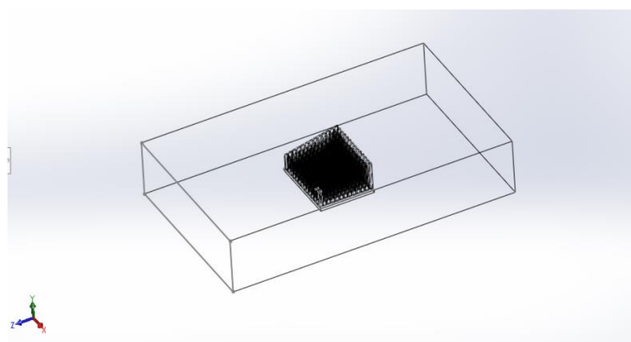


Fig 3.3 Computational domain and trapezoidal heat sink.

IV. RESULTS AND DISCUSSION

4.0 Introduction

In this Chapter the Discussion of CFD analysis Results of the Flat fin Heat sink and the Trapezoidal 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 conditions and setup:

Solver	Pressure Based solver
Viscous method	K-epsilon
Solution Method	SIMPLE
Inlet	Velocity inlet
outlet	Pressure Outlet
Velocity inlet	1m/s
Inlet Temperature	296.6K

Case 1: Flat fin heat sink

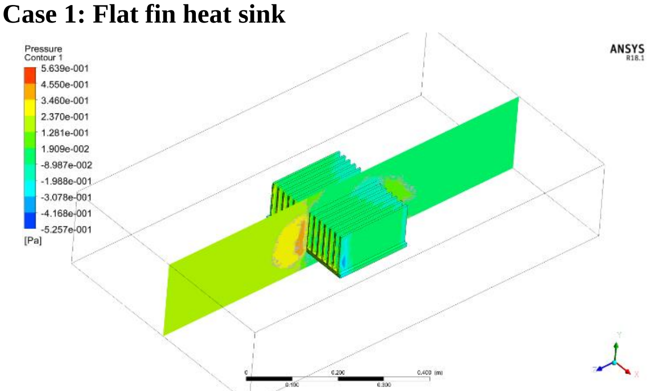


Fig 4.1 Pressure distribution of flat fin heat sink

The above Figure Represents the pressure Distribution from the domain along inlet to outlet where the red colored region represents the maximum pressure and the blue colored region represents the minimum pressure in between is the distribution the colored bar at the left side of the picture is legend subsequently with the color associated with the value.

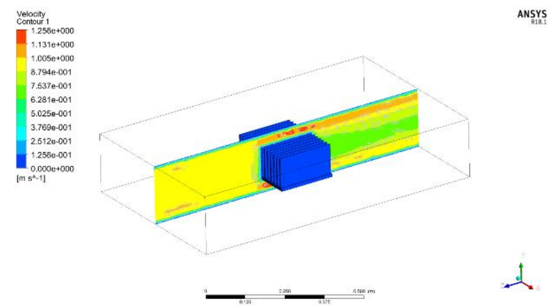


Fig 4.2 Velocity distribution of flat fin heat sink along the length of the domain.

The above Figure Represents the Velocity Distribution from the domain along inlet to outlet where the red colored region represents the maximum Velocity and the blue colored region represents the minimum Velocity in between is the distribution the colored bar at the left side of the picture is legend subsequently with the color associated with the value.

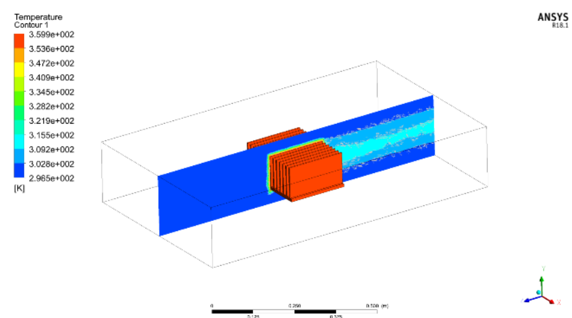


Fig 4.3 Temperature distribution of the flat fin along the length of the domain.

The above Figure Represents the Temperature Distribution from the domain along inlet to outlet where the red colored region represents the maximum Temperature and the blue colored region represents the minimum Temperature in between is the distribution the colored bar at the left side of the picture is legend subsequently with the color associated with the value.

Case 2: Trapezoidal fin heat sink

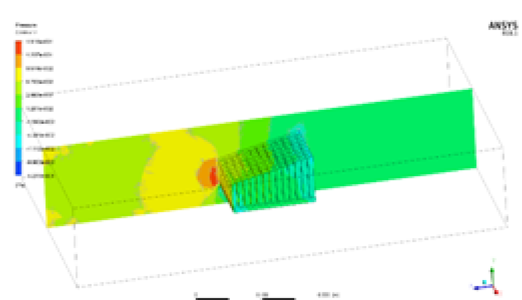


Fig 4.4 Pressure distribution of trapezoidal fin heat sink

The above Figure Represents the pressure Distribution from the domain along inlet to outlet where the red colored region represents the maximum pressure and the blue colored region represents the minimum pressure in between is the distribution the colored bar at the left side of the picture is legend subsequently with the color associated with the value.

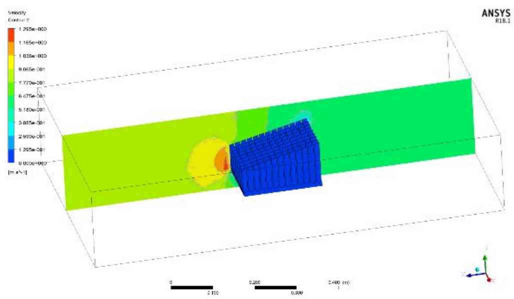


Fig 4.5 Velocity distribution of trapezoidal fin heat sink along the length of the domain.

The above Figure Represents the Velocity Distribution from the domain along inlet to outlet where the red colored region represents the maximum Velocity and the blue colored region represents the minimum Velocity in between is the distribution the colored bar at the left side of the picture is legend subsequently with the color associated with the value.

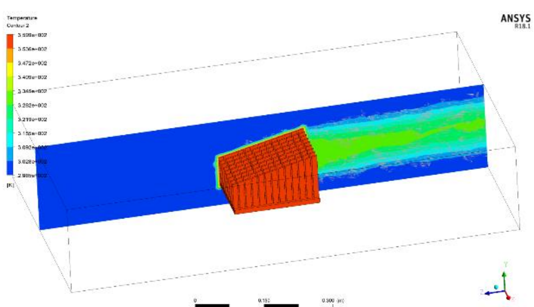
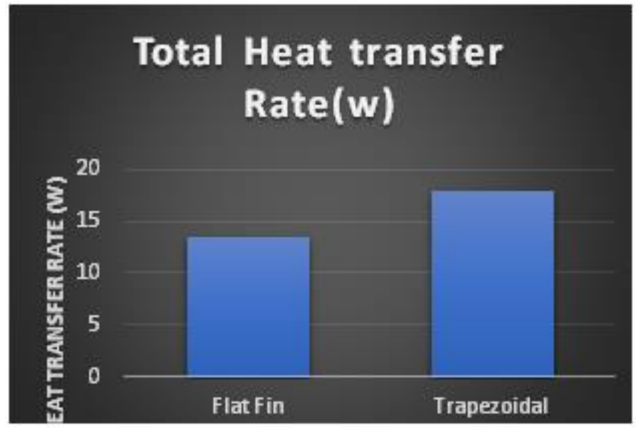


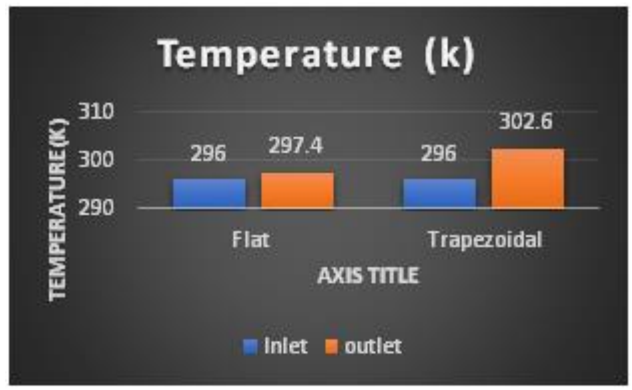
Fig 4.6 Temperature distribution of the trapezoidal fin along the length of the domain.

The above Figure Represents the Temperature Distribution from the domain along inlet to outlet where the red colored region represents the maximum Temperature and the blue colored region represents the minimum Temperature in between is the distribution the colored bar at the left side of the picture is legend subsequently with the color associated with the value.

Plot



Plot 1 Total heat transfer rate



Plot 2 Inlet and outlet temperature distribution

V. CONCLUSION

From the simulation done in computational fluid dynamics for both flat rectangular fin and trapezoidal fin, the heat gained by the fluid in the trapezoidal shaped fin more. Therefore, from the temperature distribution we can see the significant raise of outlet temperature. Hence the trapezoidal fin is efficient in transferring more heat.

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.

[10] 136Bejan, 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* 31*st 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,"