IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X
PP. 67-76
www.iosrjournals.org

"Numerical And Experimental Investigation of Effect of Dimples on Heat Transfer By Using Multi Air Jet Impingement On Pin Fin Heat Sink"

Mr. Sunil N. Tapase, Prof. Pravin Borade
M. Tech student, College Of Engineering Pune
Asst. Professor, Mechanical Department, JSPM NTC Narhe, Pune.
Corresponding Authors: Mr. Sunil N. Tapase

Abstract: *The work reported in this thesis is an attempt to enhance heat transfer in electronic devices with use of impinging multi air jet on pin fin heat sink with dimples. Increased miniaturization, higher power densities, and demands on system performance and reliability in electronic systems have necessitated more aggressive heat removal techniques in the thermal management of electronic components. Although the technology of cooling has greatly advanced, the main cause of malfunction of electronic devices remains overheating. The problem arises due to restriction of space and also due to high heat dissipation rates. To avoid the problem of overheating, work carried out in this thesis is computational and experimental investigation, using multi air jet impingement (3x3 nozzle array), impinging normally on pin fin heat sink(4x4 pin fin array) with dimples (3x3dimple array).*

Experimental and numerical analysis carried out on pin fin heat sink with dimples by varying Reynolds number ranging from 7000 to 11000, Distance between nozzle plate and target plate 6, 8, 10 times the nozzle diameter and Dimple depth 0, 1.25, 2.5 mm for constant heat supply of 30W. The flow and heat transfer characteristics of circular multi air jet, impinging vertically on pin fin heat sink with dimples, have been analyzed numerically using CFD commercial code ANSYS ICEM–CFD and ANSYS–CFX. Numerical simulation techniques and results are described.The results reveal that heat transfer enhancement is more for shallow dimple than the deeper dimple and at lower nozzle plate to target plate distance.

I. Introduction

The continuing increase of power densities in microelectronics and the simultaneous drive to reduce the size and weight of electronic products have led to the increased importance of thermal management issues in this industry. The temperature at the junction of an electronics package (chip temperature) has become the limiting factor determining the lifetime of the package. The most common method for cooling packages is the use of aluminum pin-fin heat sinks. These heat sinks provide a large surface area for the dissipation of heat and effectively reduce the thermal resistance of the package. They often take less space and contribute less to the weight and cost of the product. For these reasons, they are widely used in applications where heat loads are substantial and/or space is limited. They are also found to be useful in situations where the direction of the approaching flow is unknown or may change. They offer a low cost, convenient method for lowering the thermal resistance and in turn maintaining junction temperature at a safe level for long term, reliable operation. The overall performance of a pin-fin heat sink depends on a number of parameters including the dimensions of the base plate and pin fins, thermal joint resistance, location and concentration of heat sources. These parameters make the optimal design of a heat sink very difficult [1].

1.1 Introduction to Jet Impingement:

Jet impingement is one of the most efficient solutions of cooling hot objects in industrial processes as it produces a very high heat transfer rate of forced-convection. There is a large class of industrial processes in which jet impingement cooling is applied such as the cooling of blades/vanes in a gas turbine, the quench of products in the steel and glass industries and the enhancement of cooling efficiency in the electronic industry. Over the past 30 years, experimental and numerical investigations of flow and heat transfer characteristics under single or multiple impinging jets remain a very dynamic research area. Impinging fluid jets are frequently used for intensive cooling in various technological applications. The principle of impinging jets is to reduce the thickness of boundary layer and augment the convection. However achieving optimum heat transfer for a specific purpose particularly for pin fin array is still challenging because of large number of parameters involved such as nozzle shapes and their arrangements, impinging wall shape and its geometric parameters, distance between nozzle and impinging wall surface, the initial flow field and turbulence intensity in the jet and so on.

II. Literature Review

In the literature review we studied different papers related to the Pin fin heat sink Optimization/Experimental/Numerical (CFD analysis). An extensive literature survey of fluid flow and heat transfer from single isolated cylinders (circular and elliptical), cylinder/ pin-fin arrays and tube banks is conducted which is a critical step for developing fluid friction and heat transfer models for pin-fin heat sinks. A Pin fin heat sink study can provide insights into convection heat transfer effects due to boundary - layer separation, Reynolds number, and Prandtl number.

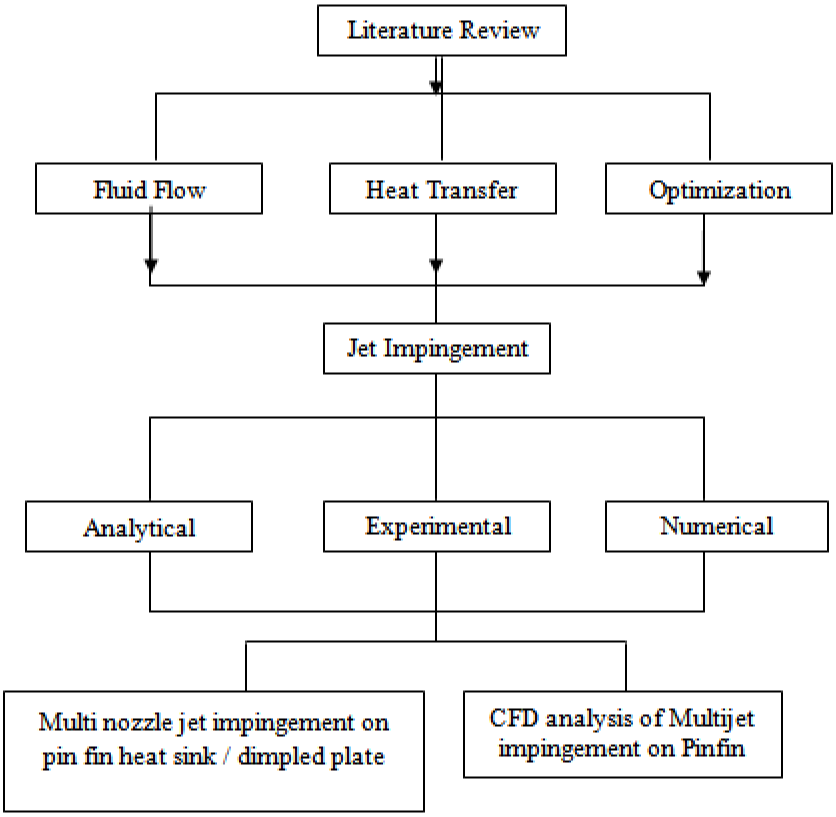

Fig.3.1 Flow Chart showing procedure for literature review.

The present literature review is categorized into two parts, namely literature on a) Experimentation and Optimization of Multi jet impingement on Pin fin and dimpled plate. b) CFD analysis of Multi jet impingement on Pin fin.

Experimentation and Optimization of Multi jet impingement on Pin fin heat sink and dimpled plate heat sink

The conventional method of electronic cooling uses steady jets. Several experiments and numerical studies have been performed to analyze the effectiveness of steady jet impingement cooling. Some of them are described as below.

Luis A. Brignoni and Suresh V. Garimella (1999)[6], Conducted a variety of nozzle configurations were tested to characterize and optimize the performance of confined impinging air jets used in conjunction with a pin-fin heat sink. Four single nozzles of different diameters and two multiple-nozzle arrays were studied at a fixed nozzle-to- target spacing, for different turbulent Reynolds numbers (5000<Re<20, 000). Variations in the output power level of the heat source and nozzle-to target spacing were found to have only modest effects on heat transfer at a fixed Reynolds number. Enhancement factors were computed for the heat sink relative to a bare surface, and were in the range of 2.8–9.7, with the largest value being obtained for the largest single nozzle (12.7 mm diameter). Average heat transfer coefficients and thermal resistance values are reported for the heat sink as a function of Reynolds number, air flow rate, pumping power, and pressure drop, to aid in optimizing the jet impingement configuration for given design constraints.

Jim G. Maveety et al (2002)[8] they have investigated on square pin fin heat sink under jet impingement cooling. They investigated on Experimental and numerical results are presented for heat transfer from a C4 mounted organic land grid array (OLGA) thermal test chip cooled by air impingement. Five heat sink geometries were investigated for Reynolds numbers ranging from 9,000 to 26,000. The dimensionless nozzle-to-heat sink vertical spacing was varied between 2 and 12. In this study, they investigate the interactions between heat sink geometry, flow conditions and nozzle setting and how they affect the convective heat transfer and overall cooling of the test chip as measured by total thermal resistance. Optimizing fin arrays by minimizing the overall heat sink thermal resistance instead of focusing solely on maximizing the heat transfer from the fins is shown to be a better design criterion.

Zeinab S. Abdel-Rehim (2007) in this work, the heat transfer and fluid flow analysis are employed to optimize the geometry of the pin-fin heat sinks. An entropy generation minimization (EGM) method is employed to optimize the overall thermal performance and behavior of pin-fin heat sinks. The performance of the heat sinks is determined by its thermal resistance and pressure drop since they significantly influence the thermal resistance during forced convection cooling. The optimum design of heat sink for in-line and staggered alignments with circular, square, rhombus, rectangular, and elliptical configurations are investigated and the thermal behavior is compared. The selected materials are; Plastic, Aluminumand copper. Thermal and hydrodynamic analysis of pin-fin heat sink are performed using parametric variation of each design variable including pin diameter, or side, pin height, approach velocity, number of pin-fins and thermal conductivity of the material. Optimization of heat sink designs and parametric behavior are presented and compared based on the selected pin-fin configurations, alignment and material property. The results indicate that geometries of circular and elliptical shapes provide more favorable condition for heat transfer than that of square, rectangular and rhombus shapes. Copper fin gives the best performance, and if the weight of the heat sink is a constraint, the Aluminum fin would be preferable [7].

W. A. Khan, J. R. Culham (2004) in this study, an entropy generation minimization, EGM, technique is applied as a unique measure to study the thermodynamic losses caused by heat transfer and pressure drop in cylindrical pin-fin heat sinks. The use of EGM allows the combined effect of thermal resistance and pressure drop to be assessed through the simultaneous interaction with the heat sink. Both in-line and staggered arrangements are studied and their relative performance is compared on the basis of equal overall volume of heat sinks. It is shown that all relevant design parameters for pin-fin heat sinks, including geometric parameters, material properties and flow conditions can be simultaneously optimized[9].

Hani A. El-Sheikh and Suresh V. Garimella (2000) the enhancement of heat transfer from a discrete heat source in confined air jet impingement were experimentally investigated. A variety of pin-fin heat sinks were mounted on the heat source and the resulting enhancement studied. Average heat transfer coefficients are presented for a range of jet Reynolds numbers (8000 Re 45000) and orifice diameters (12.7≤ d≤38 1 mm). Total fin effectiveness was computed for the pinned heat sinks relative to the unpinned ones, and was in the range of 2.4 to 9.2; the highest value was obtained for the largest nozzle diameter. Heat transfer rates from the bare heat source were increased by a factor of 7.5 to 72 due to the introduction of the heat sinks. Results for the average heat transfer coefficient were correlated in terms of Reynolds number, fluid properties and geometric parameters of the heat sinks[10].

a) Experimentation and Optimization of Multi jet impingement on Pin fin heat sink and dimpled plate:

Most of study is conducted on simple geometries like flat plate or smooth cylinders. These are for various Reynolds numbers and jet-to-plate distances Z/D ratios (target plate), heat flux supplied, heat sink geometry, temperature measurement [8,10].

1. It can be seen that, the geometry like pin fin with dimples has not been studied yet numerically or experimentally. By combining the parameters mentioned above, a significant augmentation in heat transfer can be expected with air as the cooling medium. This has wide range of applications in thermal management of electronic systems.
2. The shallow dimple gives high rates of heat transfer.
3. Narrow nozzle plate to target plate distance gives high rates of heat transfer.
4. There are potentials to improve heat transfer with air impinging jets provided an optimal combination of the governing parameters is to be identified.

b) CFD analysis of Multi jet impingement on Pin fin heat sink:

1. While the study on CFD analysis of multi jet impingement problem we didn't get the significant turbulence model so we go for k-ε, RNG k-ε, k-ω and SST k-ω and found perfect one.
2. To perform a preliminary study into mechanism providing heat transfer improvement in pin fin array under jet impingement, with emphasis on identifying modifications which best enhance mechanisms
3. To perform detailed parametric study of heat transfer improvement due to the modifications identified in part (2) using commercial CFD package CFX-12.0

4. Based on experimental and numerical simulations, optimal selection of heat sink configurations will be investigated for various operating conditions.

MULTI JET IMPINGEMENT COOLING

The heating or cooling of large areas with impinging jets requires arrays; however, the flow and geometrical parameters have to be carefully selected to provide both a sufficiently high average heat transfer coefficient and uniformity of the heat transfer over the impingement surface. The need for uniformity is important in applications such as drying of textile and paper, annealing and tempering of glass, cooling of turbojet engine structure to avoid local hot spots, and spot cooling of electrical apparatus. These and other applications motivated the research. The flow from arrays of impinging nozzle has the same three flow region free jet, stagnation, and wall jet- as the single impinging jet. However, there are some basic differences in the fluid mechanics of single and multiple jets that complicate the use of single jet heat transfer results for the design of multiple jet systems.

The multiple jets systems can be subdivided into three different kinds of arrays. (I)Round jets from free tube (e.g. In-line and staggered), (II) round or slot jets from perforated plate with or without spent air holes, and (III) rows from hole channels which can be considered as mixture of perforated plate and free jet. Many additional factors influence the heat transfer in multiple jets impingement systems. These factors include separation distance, jet-to-jet spacing distance, kind of array, and geometry of jet, diameter of jet and impingement surface form. For arrays perforated plate impingement jets, a cross flow is formed by the spent air from the impinging jets in a confined space, and the amount of cross flow increases as the flow moves downstream. Turbulent intensity of impinging jets is increased because the cross flow disturbs impinging jets at downstream region. Therefore, the local heat transfer rate around the stagnation region is enhanced. However, at the mid-way region, the heat/mass transfer is decreased because the spent fluid upstream jets in an array can sweep away the downstream jets and delay impingement. Also the thermal boundary layer is developed in the cross flow at this region. Therefore, the heat/mass transfer coefficient is non-uniform over the overall impingement surface.

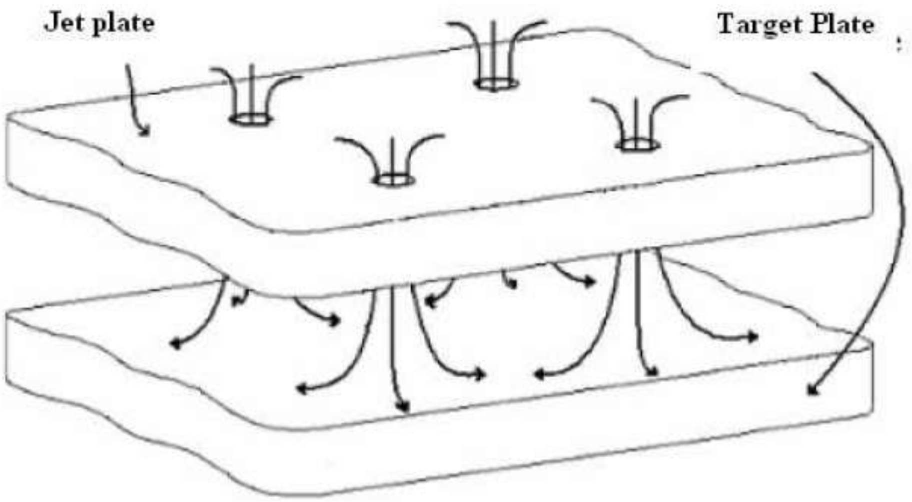

Fig.2.1 Multi jet system on target plate

PIN FIN HEAT SINK PARAMETER SELECTION

Pin fins are widely used to increase the rate of heat transfer from a wall. They come in many shapes and forms, a few of them are shown in Fig. 4.1(Types of fins). The selection of suitable fin geometry requires a compromise among the available space, weight, cost, and the pressure drop of the fluid as well as the heat transfer characteristics of the fin surface. The fins increase the hA product and hence decrease the convective thermal resistance 1/hA, where h is the convective heat transfer coefficient and A is the surface area of the fin.

Assuming constant thermo physical properties, heat transfer coefficient, thermal conductivity and homogeneous and isotropic material for the fin, an integral approach of the boundary layer analysis is employed to derive closed-form expressions for the calculation of local and average heat transfer from single isolated pins (circular and elliptical), and cylindrical pin-fin arrays.

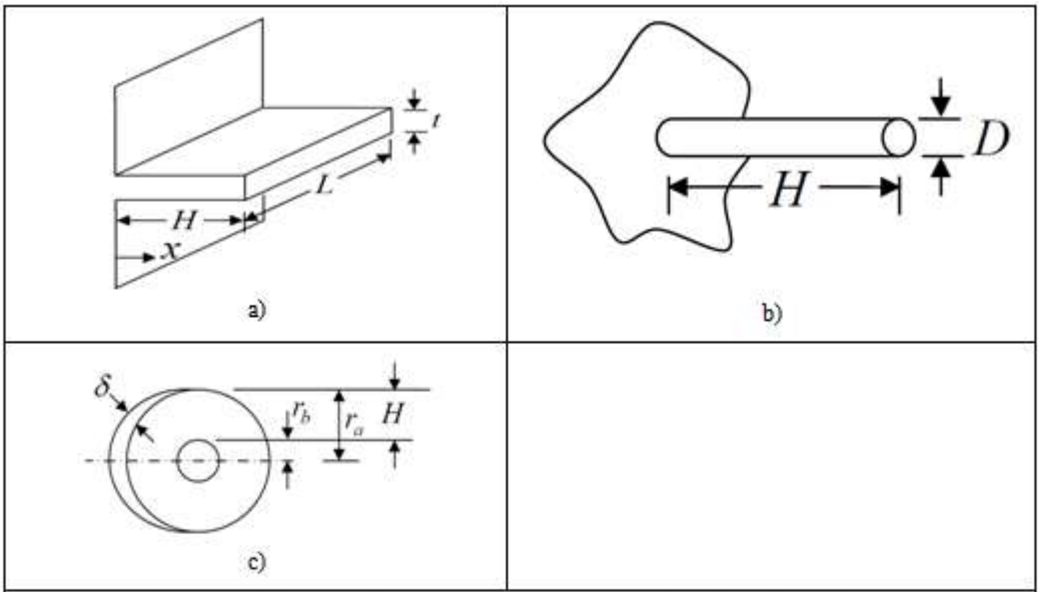

Fig4.1:Types of Fins :a)Longitudinal Fin b)Cylindrical Fin C)Radial Fins

EXPERIMENTAL APPARATUS AND SETUP

A schematic of the experimental test setup is as shown in fig.(5.1). The experimental apparatus mainly consisted of Air Flow Bench, Jet assembly: Nozzles (5mm), Top jet plate, Bottom jet plate, pin fin heat sink, Heater (100W, 230V), mica sheets and other accessories. The air was supplied by the air flow bench through its Centrifugal Blower. A Honeycomb structure is used inside the plenum chamber in order to provide streamlined flow prior to entering the jet plate and a butterfly valve used in order to regulate the discharge. (The specifications of Air Flow Bench are mentioned in appendix).Photograph of Experimental test setup is provided in Appendix.

5.1 Experimental test Setup Assembly:

Experimental test set up consists of nozzle jets plate assembly, pin fin heat sink, heater, asbestos sheet, supporting bars, bottom plates, nuts etc. By considering the Weight and transparent property, aesthetic etc we use to select material for the manufacturing of jet plates, we use the Acrylic material for the manufacturing of set up. Considering the air flow bench exit dimensions (100×50mm) and alignment holes at exits of flow bench. We designed the experimental test set up nozzle jet plate, bottom plate, heat sink cavity in bottom plate we selected the dimensions for the nozzle jet plate and base plate as 220×110mm.

The pin fin heat sink of aluminum is of dimension 60x60x5 mm. The top and the bottom jet plates were separated by cylindrical aluminum nozzles (jet tubes) of jet diameter 5mm. The ratio of distance between the Nozzles and the target plate (Z/d) was adjusted to be in ratio 6, 8, 10.

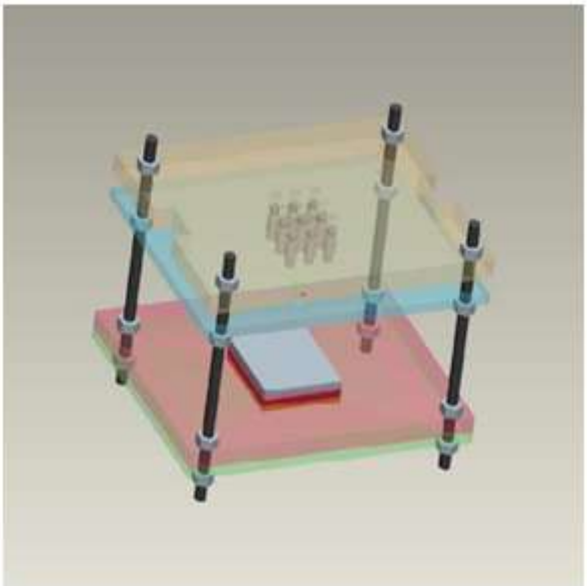

Fig.5.1 Final assembly of experimental test set up

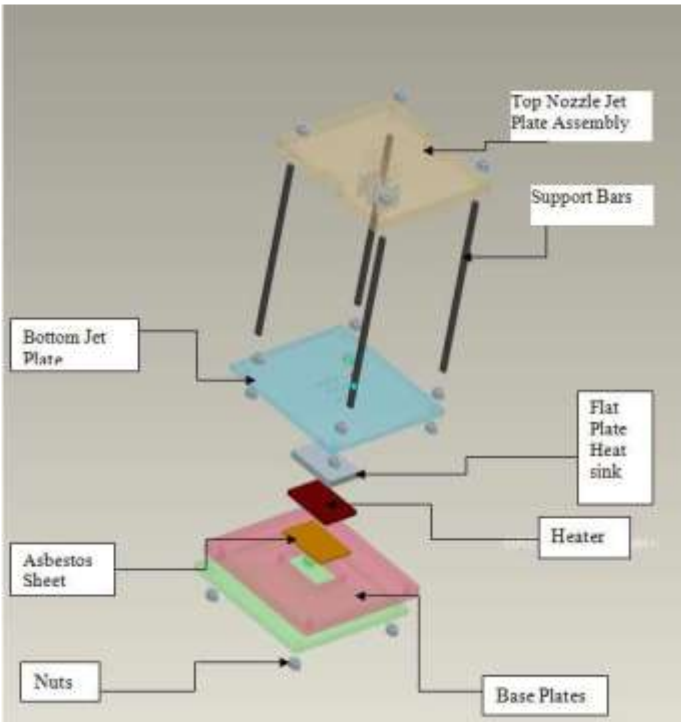

Fig. 5.2 Exploded view of assembly of experimental test set up

CFD MODELING AND SIMULATION

Computational Fluid Dynamics (CFD) has advanced sufficiently to enable such heat transfer and fluid flow calculations to be made and potentially with good confidence levels. It is a powerful numerical tool which is becoming widely used to simulate many thermal systems and fluid flow system. The accuracy of the calculations depend on a number of parameters including the choice of turbulence model and the treatment of the boundary layer since this largely dictates the nature of the heat transfer.

In CFD calculations, there are three main steps Pre-Processing, Solver Execution Post-Processing. Pre-Processing is the step where the modeling goals are determined and computational grid is createdWhen solver is terminated, the results are examined which is the post processing part. In this work the flow field is numerically examined by using ANSYS CFX 12.0[4,25]. The over view of the problem computational domain is as shown in fig (6.1).

Geometry Creation and Meshing

As geometry was small enough and we had sufficient computational power so here we considered the complete geometry for the CFD analysis and created in ANSYS ICEM CFD Ver. 13.0. All geometrical dimensions are similar as that of test set up. The opening boundary was sufficiently away from the actual physics so that flow becomes stable and actual flow phenomenon at impinging surface was captured. The unstructured mesh was created within this domain by using ANSYS ICEM CFD Ver. 12.0 with the option of part meshing. The determinant, more properly defined as the relative determinant, is the ratio of the smallest determinant of the Jacobian matrix divided by the largest determinant of the Jacobian matrix, where each determinant is computed at each node of the element. The Determinant can be found for all linear hexahedral, quadrahedral, and pyramidal elements. A Determinant value of 1 would indicate a perfectly regular mesh element, 0 would indicate an element degenerate in one or more edges, and negative values would indicate inverted elements. Generally if quality of mesh above 0.3 then it is acceptable. Here in this work the quality of all the meshing for all geometries ($6 \leq H/D \geq 10$) falls in the range of 0.35 to 1.0[25,26]

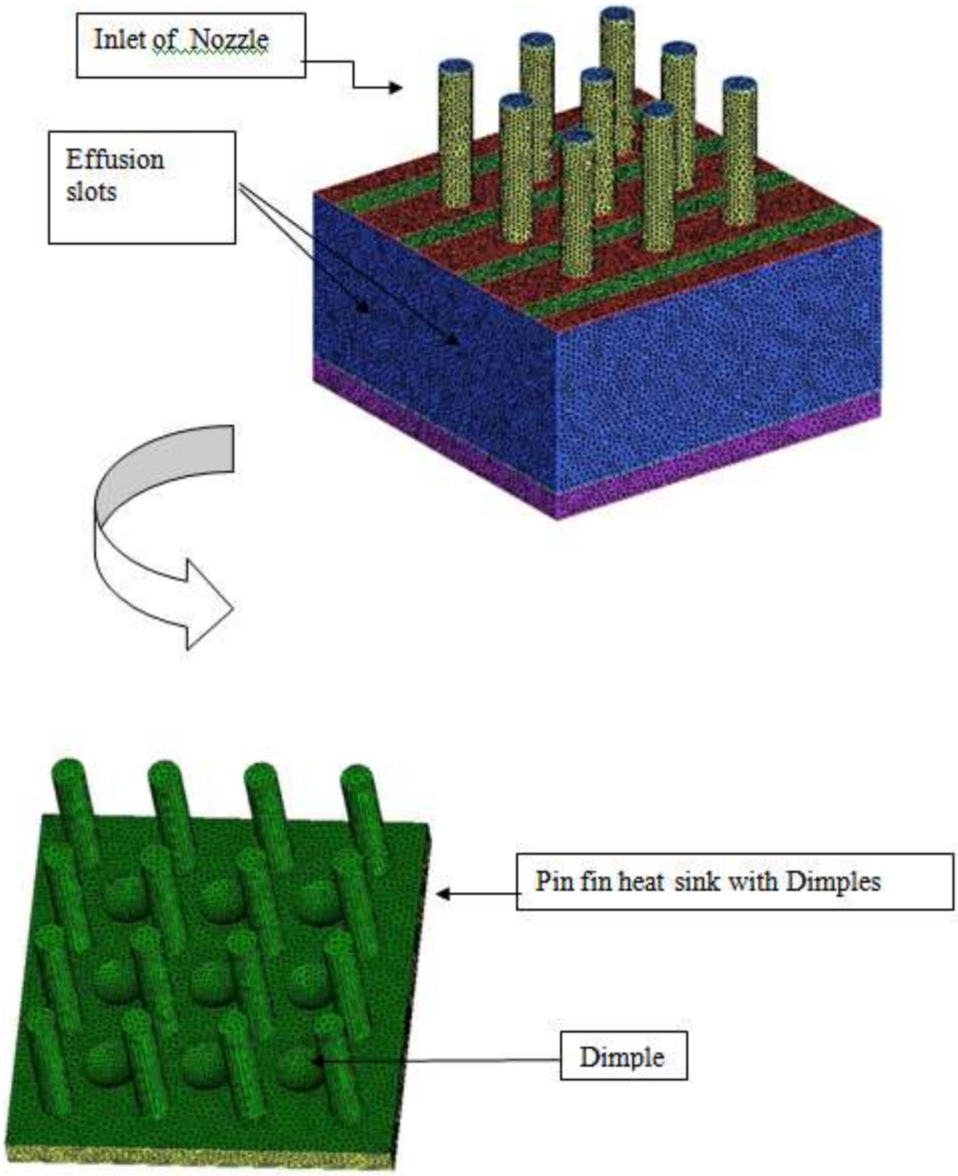

Fig 6.1- Computational Domain with Grid

III. Results And Discussions

The tests were carried out in turbulent range for Reynolds numbers ranging from 7000-11000. It was observed that, there is appreciable rise in heat transfer coefficient with increase in Reynolds number for both cases, pin fin heat sink with dimple and without dimple.

Ultimately, all CFD results were validated by comparison to reliable experimental results and to determine overall model error in predicting the real situation. Obviously, the model would match the experimental conditions, including all of the geometry, fluid entry, exit conditions, and target surface properties. This matching must include not only the domain boundary average velocities, pressures, and temperatures, but also their turbulent components.

Here in this study both experimental and CFD analysis results are presented at a time, with their comparison graph between CFD and experimental results. Also post processing contours results are given below.

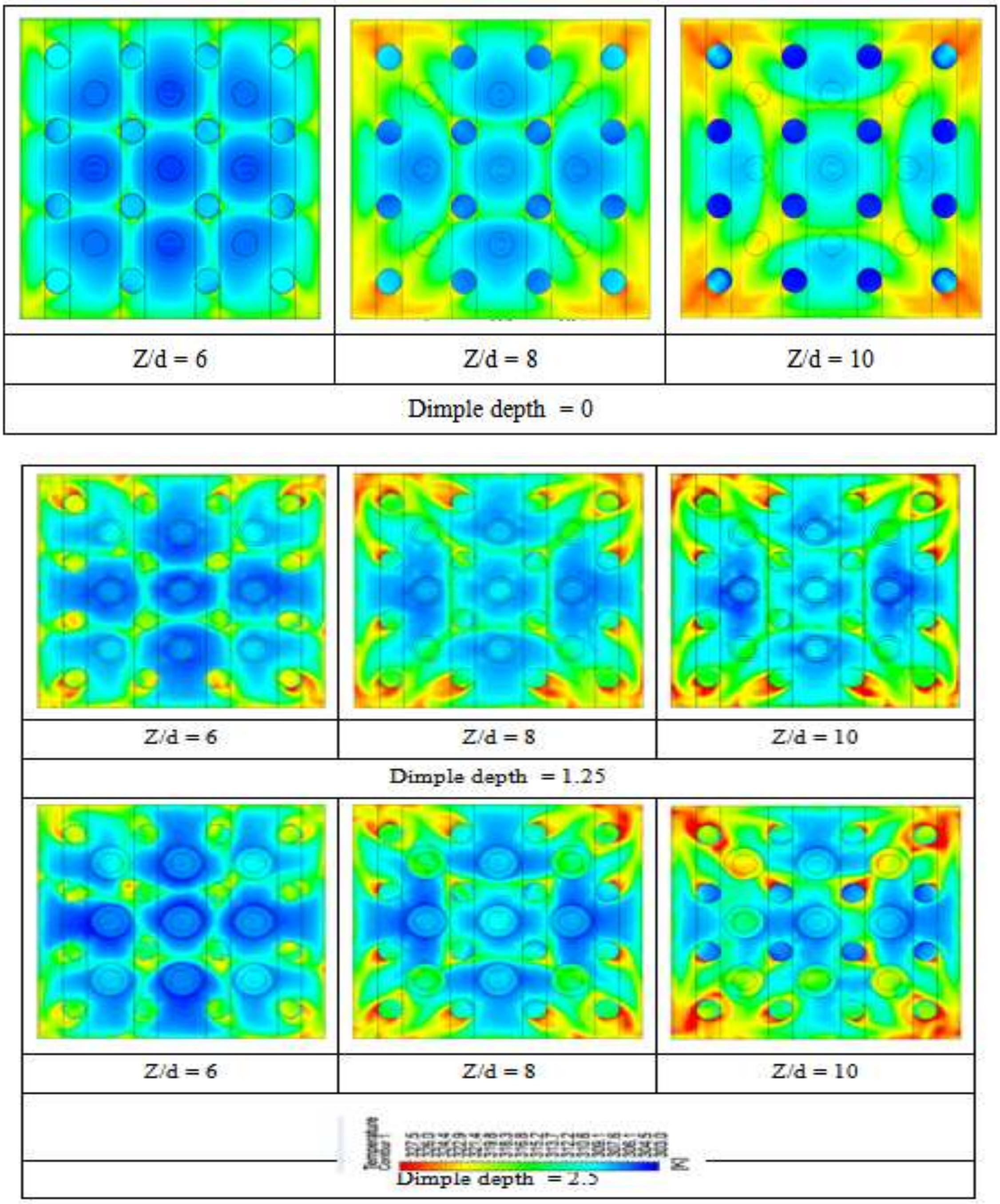

Fig 7.1- Temperature contours at Reynolds number (Re 11000)

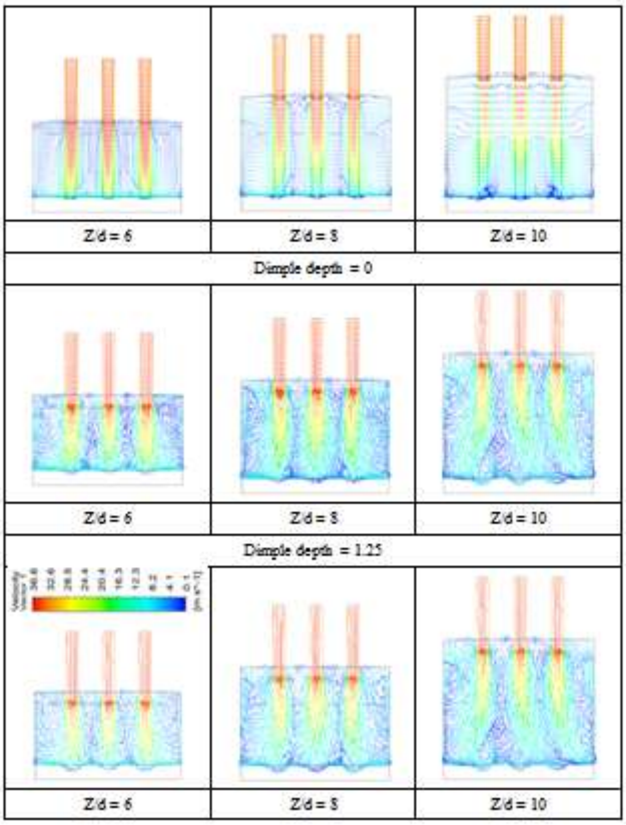

Fig 7.2- Velocity vector plots for Reynolds number(Re 11000)

IV. Conclusions

The work presented in this thesis is research on cooling performance in electronic devices using jet impingement cooling via experimental as well as numerical simulation using ANSYS CFX-12.0. The major finding of this study chronicled below.

1. The nozzle Jet plate to pin fin heat sink spacing (Z/d ratios) had a significant impact on heat transfer on a pin fin heat sink with dimples and without dimples which can be observed from the graphs.
2. As Reynolds number increases the Nusselts number also increases for both geometry i.e. pin fin heat sink with dimple and pin fin heat sink without dimple.
3. As the dimple depth increases cooling performance decreases. More amount of cooling is observed for pin fin without dimple.
4. The area surrounding to dimple gets cooled more, because of breaking of thermal boundary layer.
5. Longer core jet is observed for lower Z/d.
6. Vortices are generated in between dimples which enhances the heat transfer.
7. Nusselts number is higher for shallow dimple than the deep dimple.

References

[1]. Waqar Ahmad Khan," Modeling of fluid flow and heat transfer for optimization of pin Fin heat sinks",ASME,2004.
[2]. Philipe Marty,Vaclav Sobolik,Experiemental Study of pulsating round impinging jet 16 Dec-2002.
[3]. Anuradha Sanyal, "Numerical study of heat transfer from pin fin heat sink using steady and pulsating jets", IISc Bangalore, April 2009.
[4]. ANSYS CFX-12.0,User Manual.
[5]. Akin Burak Etemoglu, "A brief survey and economical analysis of air cooling for electronic equipments", International Communications in Heat and Mass transfer,34(2007)103-113.
[6]. Luis A. Brignoni And Suresh V. Garimella,"Experimental optimization of confined air jet impingement on pin fin heat sink, IEEE Transactions on components and packaging Technology.Vol.22,No 3,September.
[7]. Zeinab S.Abdel Rehim, "Optimisation and thermal performance assessment of pin fin heat sinks", Journal of Applied Sciences Research,3(3),227-235,2007.
[8]. Jim G. Maveety, Henry H. Jung, "Heat Transfer from square pin fin heat sinks using air impinging cooling",Vol 25,No 3, September 2002.
[9]. W.A. Khan, J.R. Culhamn and M.M. Yovanowich,"Optimisation of pin fin heat sinks using entropy generation Minimization"2004 Inter society Conference on thermal phenomena.
[10]. H.A. El-Sheikh, S.V.Garimella,"Heat transfer from pin fin heat sinks under multipleimpinging jets",IEEE Transactions on components and packaging,Vol.23, No 1, February 2000.

[11]. C.J. Kobus, T. Oshio,"Development of theoretical model for predicting the thermal convection with impinging flow" , International journal of Heat and Mass transfer48(2005)1053-1063.
[12]. M.Can," Experimental optimizatio9n of jet impinging on continuously moving flatplate",Heat and Mass transfer 39(2003)509-517.
[13]. K. Kanokjaruvijit, R.F. Martinez-Botas, Jet impingement on a dimpled surface with different crossflow schemes, Int. J. Heat Mass Transfer 48 (2005) 161–170
[14]. G.S. Azad, Y. Huang, J.C. Han, Impingement heat transfer on dimpled surfaces using atransient liquid crystal technique, J. Thermophys. Heat Transfer 14 (2000) 186–193
[15]. Yunfei Xing, Bernhard Weigand, "Experimental and Numerical Investigation of Heat Transfer Characteristics of Inline and Staggered Arrays of Impinging Jets", Journal of HeatTransfer(2010)September 2010, Vol. 132 / 092201-1.
[16]. Dong-Ho Rhee, Pil-Hyun Yoon, Hyung Hee Cho," Local heat/mass transfer and flow characteristics of array impinging jets with effusion holes ejecting spent air", International Journal of Heat and Mass Transfer 46 (2003) 1049–1061.
[17]. J.Badra,A.R.Marsi,M.Freeman, andM.Fitch,"Enhanced heat transfer from Arrays of jets impinging on plates",16th Australian Fluid Mechanics conference, Gold coast, Australia(2007),1187-1192.
[18]. Y.Q.Zu,Y.Y.Yan,J.D.Malston," CFD prediction for confined impingement heat transfer Using different turbulence models ", School of the Built environment, University of Nottingham NG 2RD, UK.
[19]. J.Badra,A.R.Marsi,andM.Fitch," Calculation of transient heat transfer from arrays of circular jets',16th Australian Fluid Mechanics conference, Gold coast, Australia(2007),150-163.
[20]. Kadem,A.Mataoui,A.SalemandR.Younsi," Numerical Simulation of Heat Transfer in an Axisymmetric turbulent jet impinging on flat plate", Advanced Modeling Optimisation, No 9,204-217, Vol 2,2007.
[21]. Yue-TzuYang*,Huan-SenPeng," Numerical study of the heat sink with uniform fin width designs", International journal of Heat and Mass transfer,52 (2009)3473-3480.
[22]. H.Jung and J.G. Maveety,"Pin fin Heat sink modeling and characterization", IEEE 2000.
[23]. Colin Glynn," Jet impingement cooling", Journal of Thermal Science 2000.
[24]. Srinath V. Ekkad , David Kontrovitz," Jet impingement heat transfer on dimpled surfaces", International Journal of heat and fluid flow 23(2002)22-28.
[25]. ANSYS-ICEM-CFD 12.0, User manual.
[26]. Kenichi funazaki and Hamidon Binsalleh."Extensive studies on Internal and external heat transfer characteristics of integrated impingement cooling structures for HP turbines", Proceedings of ASME Turbo expo 2008.
[27]. Neil Zukerman and Noam Lior," Impingement heat transfer correlations and numerical modeling", Journal of heat transfer by ASME, Vol 127,May 2005.