Performance Evaluation of Plate-fin and Pin-fin Heat Sinks for The Application of Oil Immersion Cooling and Design Optimization of Dynamic Cold Plate for The Application of Warm Water cooling

by

Parth Soni

Presented to the Faculty of the Graduate School of

The University of Texas at Arlington in Partial Fulfillment

Of the Requirements

For the Degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON

MAY 2016

Acknowledgements

I would first like to thank my thesis advisor Dr. Dereje Agonafer of the Mechanical Engineering at The University of Texas at Arlington. The door to Prof. Agonafer's office was always open whenever I ran into a trouble spot or had a question about my research or writing. He consistently allowed this to be my own work, but steered me in the right the direction whenever he thought I needed it.

I would also like to thank Dr A. Haji-Sheikh and Dr Miguel Amaya for serving as my thesis committee and honoring me with valuable comments. And special thanks to Ruturaj Kokate and Shivang Desai for kind help.

I would also like to acknowledge Ms. Sally Thompson of the administrative department for the help in administrative matters and other study matters. Our conversion always are stress relievers and for all the kindness for my well-being.

Finally, I must express my very profound gratitude to my parents and to my Team members. For helping through my research.

Parth Soni

05/06/2016

Abstract

Performance Evaluation of Plate-fin and Pin-fin Heat Sinks for The Application of Oil Immersion Cooling and Design Optimization of Dynamic Cold Plate for The Application of Warm Water cooling

Parth Soni, MS

The University of Texas at Arlington, 2016

Supervising Professor: Dr. Dereje Agonafer

In IT industries, to keep server rooms and data centers cool is a constant challenge for hardware developers and the cooling solution providers. Air cooling is the most widely used approach for thermal management of servers in data centers from many years. However, the scenario is changing gradually as the computational ability of data center is increasing exponentially. Because of increasing computational ability, the heat generated by the servers is increased in such amount that air cooling is not as efficient as it was before. Hence, alternative cooling system is required. In 1970's the idea of liquid cooling was introduced. Liquid cooling can be decided in two different parts.

First is water cooling and oil cooling. The study in this thesis is based on both the system. Firstly, for warm water cooling, this study presents the optimization of the cold plate cover. By optimizing this design, the efficiency of the cooling system can be improvised.

Secondly, in oil immersed cooling, the purpose of the study of these heat sinks is to document the performance characteristic of both the heat sink as it was not studied before for the oil immersed cooling application.

CONTENTS

List of illustrations

List of Tables

Chapter 1 Introduction

Keeping temperature of IT servers in the operating range is the constant challenge for hardware developers and cooling providers**Error! Reference source not found.**. Initially solution to this problem was air cooling. In air coolling the fans blow cold air over the heating component of the server. The cold air will carry heat outside the server. This method was totally based on the pure convection. This method is widely used because of nonconductive behavior and cost effectiveness of the system. The major component used in the systems are fans, controlled and power source. The fan speed can be controlled according to the load carried by the heat generating components. Hence, air cooling was considered as the reliable and efficient way of the cooling of the server for some time.

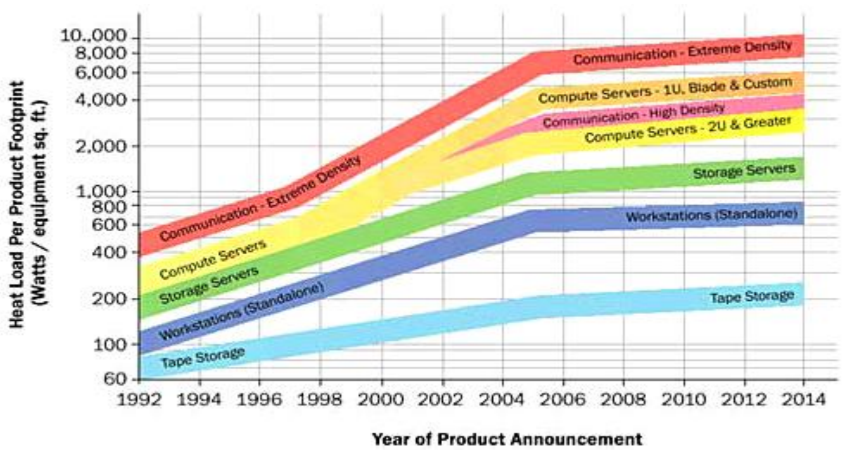

Figure 1-1: Power trend in Data centers

However, air cooling requires the pathway for the air to blow. So, it requires duct to direct the blowing air over the heat-generating component. And ducting results in space consumption. For an efficient cooling ducting optimized number of fans for each and every server required. Apart from that, power used in filtration and cooling is increasing as the heat generated from the servers increase. Lastly, considering the power generation trend of the data center, data center developers and users will require some more efficient and reliable technology for the data center cooling is required.

In 1970 the concept of liquid cooling was introduced as an objective of the air cooling system. But, such technology was very advanced at the time of introduction. Because electronics industry at that time had started to grow. So, the air cooling systems got popular as they are handy and reliable technology. But, the scenario has changed as data centers require more effective technology because of the incapability air cooling system. Hence, liquid cooling has emerged as an alternative to the air cooling. Because heat caring capacity of the liquid is far better than air. Apart from that, as these techniques based on conduction type of heat transfer rather than convection, these techniques shows better results than convection type of heat transfer techniques. In addition to that, these techniques are less space consuming techniques then air cooling. Likewise, the coolant used in these techniques can be recycled for the same application for a long time. Also, oil cooling system can be designed in such a way that it has to travel less distance because of coolant's reusability. However, in an air cooling system such system cannot be designed as a part of air has to be replaced for the further cooling. Hence, oil immersed cooling system is required to be studied.

Liquid cooling was introduced in late 60's, but it came into consideration in current years. The two techniques of liquid cooling are presented in this research named warm water cooling and oil immersed cooling. It is evident for in the name of the techniques that oil immersed cooling is using oil as the coolant and warm water uses warm water as a cooling agent. In these both techniques, liquids are carrying the heat from the heat carrying components out of the servers. This heat is extracted in the chiller or another cooling system. The principle reason behind to use these cooling agents is cost effectiveness and heat carrying capacity of oil and warm water. Apart from that, such technologies can handle current power trends in the data centers. But, without proper documentation of the data and lack of experimental results of technology prevents Data center developers from using such efficient technology.

What is oil immersion cooling?

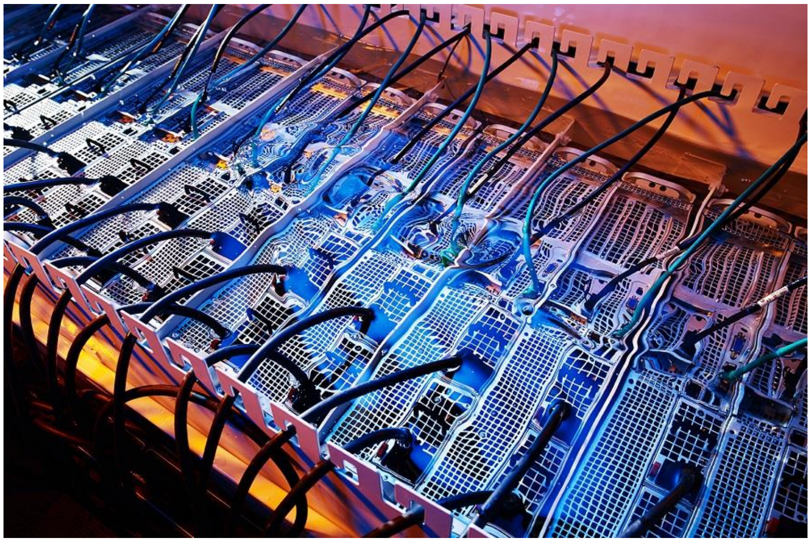

Figure 1-2: oil immersed servers

Oil immersion cooling is one of the immerging technology for cooling of data centers. In this system, the server is totally immersed in the oil tank and oil acts as a coolant. Oil carries heat from the server to radiator. Most widely used oil for the application are 3M novac and white mineral oil. By comparing both the coolants it can be concluded that both can be used for different applications.

Firstly, novac oil can be used for the 2 phase cooling as the boiling point of this oil is 34 to 49°c. Whereas mineral oil is having the boiling point way higher than the application so this oil is used for the single phase conductive cooling. Secondly, the other significant difference is, novac oil is nonflammable and mineral oil is flammable. And the last difference is, the mineral oil is having very low price compare to novac oil. Hence, mineral oil is used in this study. Because the price of this is low.

Figure 1-3: Face open compute generation-1 server

The purpose of presented research is to document the required data regarding the performance of the pin fin and plate fin heat sinks for the application of oil immersed cooling. For presented research, Facebook open compute generation one server is used as baseline model. This server was immersed and experimented for a certain time for comparison of the performance of the oil immersed cooling system. This study is using CFD modeling in ANSYS ICEPAK for study. The experimental model of from the paper [1] is used for validation of CFD model. This validated model was further used for the study of parallel plate, plate fin and pin fin heat sink which are presented in the figure 1-3 and the performance of the both heat sinks is evaluated for documentation through the parametric study.

Parallel plate heat sink Pin fin heat sink Plate fin heat sink

Figure 1-4: Different types of heat sinks

Heat sinks are passive heat exchanging devices which transfer heat generated from electronic components to fluid in motion. The principle job of the heat sink is to maximize contact surface between fluid and heat exchanging device. The performance of the heat sink can be improved using the heat spreader on heat generating device. Usually, aluminum and the copper heat sinks are available in the market. The different type of heat sinks are parallel plate, pin fin and plate fin heat sinks are easily available in the market. The parallel plate heat sink is already studied so, in this study pin fin and plate fin heat sinks are studied and optimized. Properties considered for the research are fin orientation, fin dimension, hydraulic diameter and friction coefficient. Apart from this from the inlet condition Reynolds number and Nusselt number are also calculated.

This research is based on the The experimental results from the paper. The principle goal of the experiment was to compare the performance of the server with the air cooling application [1]. The experiment shows significant results which guide towards the further study. Heat sink optimization is the initial part of the study. CFD model of the server is considered for the further heat sink studies. Firstly, for this research the CFD model was validated with the experimental data. This validation was presented in the other thesis. This validated model was then used for study of the plate fin and pin fin inline heat sinks. For the study, the original heat sink was replaced by the external heat sinks.

What is warm water cooling?

Warm water cooling is one of the technologies which is capable of replacing the air cooling technique. This system is totally different from air cooling and oil immersed cooling in terms of coolant. As “water is a good conductor of electricity” is very well known fact. But, the performance of the water cooling system can be remarkable as water is very good conductor heat. However, water cannot flow from the server in the same manner as air and oil. To use water as a coolant different kind of cooling system is required

As a result of the idea of water as a coolant for the servers. The cold plate was introduced which is demonstrated in the figure 1-5. The cold plate is a passive heat transferring device which works just like heat sinks. This cold plate is fixed on top of heat generating chip and the contact between those can be filled with heat spreading components. Heat generating chip transfer heat to the copper base of the cold plate and that copper plate transfer the heat to water flowing inside the cold plate. Then the water carries the heat to the chiller for extracting heat and filtering for next same cycle.

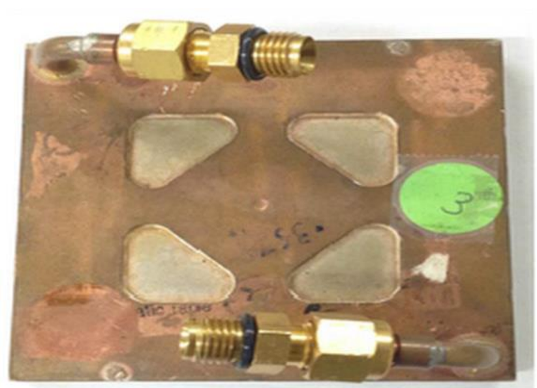

Figure 1-5: Top view of the original cold plate

Figure 1-5 shows the original cold plate which can be used for the warm water cooling. Firstly, it is evident that cold plate is having one inlet and one outlet. In between there is a serpentine path for the water to flow. While flowing through those path water absorbs heat and carries out of the cold plate. The main advantage of cold plate is to compare to heat sinks it requires less place to fit in. This cold plate requires less power than it required for the air cooling system. Hence, this system is wrath studding.

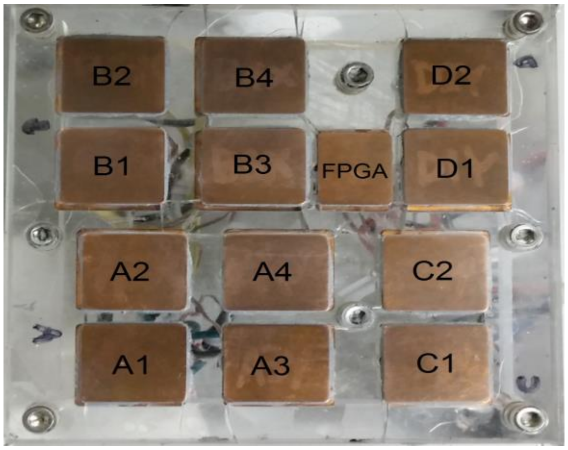

Figure 1-6: Multi-chip module (MCM)

This cold plate can be installed in place of each and every heat sinks. Water flows from this cold plate without any regards of heat load on the chip. As figure 1-6 shows the test setup prepared for the experiment In addition to that, for multichip module shown in figure 1-6. Here, if an original cold plate is used then without considering chip load, the water flow from the cold plate will be can be considered as the waste of power. So, if the design of this cold plate can be optimized then power loss from this flow. Hence, as a part of optimization, the dynamic cold plate was introduced. The dynamic cold plate can be considered as a replacement of the original cold plate.

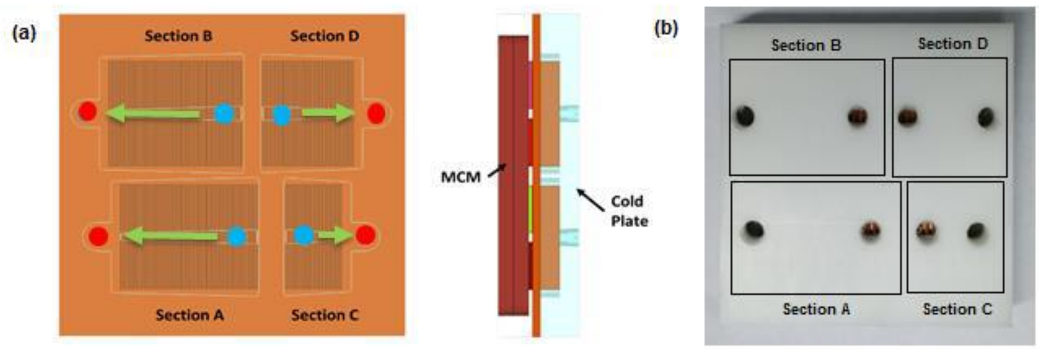

Figure 1-7: CFD and Experimental model of dynamic cold plate

Here, figure 1-7 shows the experimental model of the dynamic cold plate. In which, the cold plate is divided into four isolated compartment named A, B, C, D as shown in the figure 1-6. Here, the idea behind the compartments is to provide the distributed flow to chips. Through distributed flow the cooling of the chip can be optimized and power used for the cooling can be minimized. Apart from that, if we can control the flow inside each compartment then power can more be optimized. In addition to that, if the flow pattern is optimized then this plate can show significant results. This flow pattern can be optimized using parametric study of the cold plate.

For the optimization of the flow pattern inside the cold plate, CFD model can be the important module. Firstly, the model DCP was prepared in SolidWorks. Secondly, this model was imported in the ANSYS Workbench for the further analysis. After that, the model was transferred to the ANSYS ICEM CFD for the meshing. Lastly, after meshing the model imported in the fluent for the validation and analysis.

The model is validated with experimental data [4]. In this experiment, the flow to every compartment is supplied through different pumps at 4 lpm flow rate. For the experimental setup, only chip no-B1 is supplied at 40W of power and other are at 5w (ideal). The experiment is showing significant results as the distributed flow is provided for each compartment. Here, the total pumping power is reduced by 28% and temperature of the chip is reduced by 9%. However, the design used in this experiment can be further optimized for better results. As, while the experiment the problem of water trap between fins encountered. Likewise, even distribution of flow at the inlet was not there in the initial model. Hence, the further optimization of the design using CFD modeling is required.

Chapter 2 Literature review

2.1 Oil immersion cooling

"With the increase of circuit density and power dissipation of integrated circuit chips and other microelectronic devices, electronic packages have underlined the need for employing effective cooling devices and cooling methods to maintain the operating temperatures of electronics components at a safe and satisfactory level" D. Soodhakdee [9]. "Both direct and indirect forms of liquid cooling offer many advantages over conventional air cooling such as higher heat capacities and lower transport energy requirements" R. Eiland [10]. "As thermal design moves from the final stages of packaging development to an initial consideration, the space claim of the heatsink becomes a critical factor in board layout and chassis design "D. Copeland [11].

2.2 Dynamic Cold plate

Thermal management of microelectronics components, microsystems, and systems has been of incrementing paramount in the past few decades and will perpetuate to be paramount in the near future with a perpetuated push for performance. Liquid cooling is growing in concurrence with coolling novel, high-powered microelectronic contrivances [2]. However, the equipollent paramount field of high-power electronic contrivances has been experiencing a major paradigm shift from air cooling to liquid cooling over the last decade. For example, multiple 250-W insulated-gate bipolar transistors utilized in a potency drive for a 7000-HP motor utilized in pumping or in locomotive traction contrivances would not be amply cooled with air-cooling techniques. Another example is a "hockey puck" SCR of 63 mm diameter used to drive an electric motor that could dissipate over 1500 W and is arduous to cool with air because of the shape of the contrivance. Other contrivances include radio-frequency engenderers, industrial battery chargers, printing press thermal and sultriness control equipment, traction contrivances, mining contrivances, crude oil extraction equipment, magnetic resonance imaging, and railroad engines. This article relegates the cold plates into four types: composed tube cold plate, deep drilled cold plate, machined channel cold plate, and pocketed folded-fin cold plate. [1]

Water-cooling has sundry advantages like more preponderant heat carrying capacity, targeted distribution and lowers convey power over air cooling. Additionally, the servers are more energy- efficient when operating at higher utilization [3] making liquids more felicitous for heat transfer while maintaining desired operating temperatures. There is a comprehensive review of cold plates employed for thermal management of high-density servers in [4]. Application of CFD analysis to target amendments in subsisting cold plate designs is not unorthodox. A methodology for multi-design variable optimization of a water-cooled cold plate is mentioned by Fernandes et al. [5], which employed user defined functions to fine-tune the pumping puissance. A multi variable design optimization to minimize the thermal resistance was evaluated by varying width and height of the serpentine channels in an IBM ES/9000 cold plate. The numerical model was validated by comparison with published experimental data [6] and the soothsaid thermal resistance (baseline design) was found to be in excellent acquiescent. Fernandes et al. [7] previewed novel designs to elongate performance of static cold plates. Remsburg [8] reported a solution that utilized impingement heat transfer along with non-linear fin patterns for thermal management of a 1080W IGBT/diode assembly. When compared to conventional designs composed tube designs to machined pin fins the proposed solution reported visibly lower maximum temperatures for a series of finetuned flow rates. This was attributed to a accumulation of ameliorated heat transfer due to impingement and reduced pressure drop through optimized fin design.

From all the literature survey, they had analyzed static nature of cold plates without focusing on the capability to respond to non-uniform heat dissipation by modulating the cooling resource accordingly. In this report, the proposed parametric study on “dynamic cold plate” concept aims to ameliorate on conventional static designs through implementation of sensing and control, by redistributing the liquid across its body to contravene varying power dissipation across the contrivance being cooled.

Chapter 3 Modeling and Meshing

3.1 Modeling of experimental setup (Oil immersion cooling)

Validated CFD model of the Facebook open compute generation-1 server is used for the research presented. Initially, parallel plate heat sink is already installed in the server for the application of air cooling application. Parallel plate sink has been studied and optimized for the application of oil cooling. Results from study shows significant reduction in power as well as material consumption. However, results also show oil trap inside the wall of parallel plate heat sink. Hence, in the same model parallel plate heat sink was replaced by the plate fin and pin fin heat sink for further analysis.

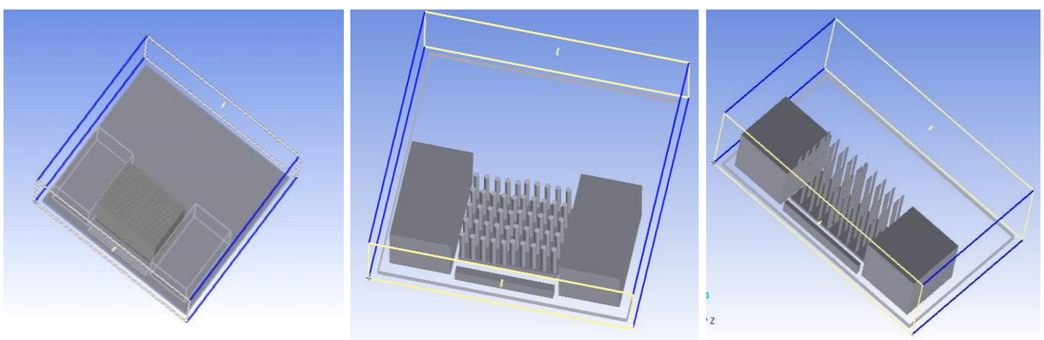

Figure 3-1: Validated and modified model of Facebook open compute Generation-1 Server

The validated CFD model of server is shown in figure 3-1, Models demonstrates the components of the servers which affects the heat transfer from the heat generating component such as RAM, Capacitors, heat sinks, PCB, and heat sources. The reason behind adding the components is to generate exact environment as an experiment. Although ram and capacitor are not affecting heat generation of the server, they affecting the flow. The simulation was also performed without considering those components. The results of the simulation show that oil is taking the least resistance path and travels from the sides of heatsinks so the results were not satisfying. However after adding those components in the model the temperature resistance curve was showing less than 10% deviation from the experimental results. Hence, the model can be concluded as a validated model and can be used for the further study.

Table 3-1: CFD model dimensions

Part	Dimension (cm)	Material
Cabinet	35×35×4	-
PCB	33×33×0.15	FR-4
Ram unit	20×5×3.4	FR-4
Capacitor unit	1×13×0.83	FR-4

Table 3-2: Dimensions of heat sinks

	PLATE FIN	PIN FIN
Foot print	10 cm × 7 cm	
Base height	0.6 cm	
Total height	3.1 cm	
Number of fin	25	
Fin radius	-	0.6 cm
Fin foot print	1cm × 0.4	-

ANSYS ICEPAK 16.2 software is used for the CFD analysis. Dimensions of the model are listed in the table no 3-1 and 3-2. The dimensions are taken form the experimental model. However, cabinet of model is smaller than actual cabinet because original cabinet was designed for the air cooling application. Apart for that, the rems and capacitors are considered as a block. Because, the block dimensions are same as total rem dimensions. Heat sinks and sources are having exact dimension as in actual experimental model. Aluminum and copper are the material commonly used for the heat sinks. However, for presented research aluminum is used as heat sink material.

3.2 Meshing of the model

The meshing of whole model is done by default mesh from ICEPAK for the further analysis. ICEPAK provides the facility of conformal and non-conformal meshing for the analysis. Conformal meshing means meshing the whole model at once. In this meshing the boundary layer is between components, in this type of meshing there no hanging nodes. The space is completely filled with solid elements. Whereas, in non-conformal mesh meshing is done separately for each components. So, there is possibility of hanging nodes and incomplete elements.

The application here is having big geometries which can be captured through conformal meshing, hence conformal meshing is used. As the default mesh generator in ICEPEK works on the conformal meshing, the meshing of the model is easy for the application. Apart from that, the conformal mesh for the model can be useful for the parametric runs of the model. Because, default mesh generator generates mesh automatically for each trial. Hence the parametric study can be easier for the application and optimization of the heat sink can save time.

3.3 Modeling of Dynamic cold plate (DCP)

CFD model of the DCP is needed to be developed for design and flow pattern optimization. The goal of optimization and validation can be achieved with steps listed below. Firstly, the design of the model in modeling software according to the experimental setup. Secondly, the same model needed to be imported in simulation software. Moreover, the model needed to be meshed for the further simulation.

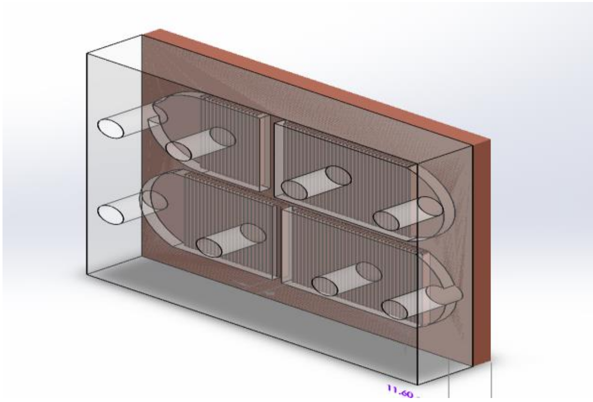

Figure 3-2: CAD model of Dynamic Cold Plate

For the optimization, Design of the model is prepared in modeling software SolidWorks. The dimensions of the model is listed in table 3-3. Parts, such as copperplate, cover plate, fins, and heated plates are modeled. These separately modeled parts are then assembled using the assembly feature of the software. The software is not capable of simulation. Hence, modeled assembly has to transfer to other software named ANSYS Workbench. In addition to that, file format has to be compatible with the simulation software. .Igs and .step are the file format which support in ANSYS Workbench. Hence, .step file format was used to export the model to other software.

Table 3-3: Dimensions for DCP

Part name	Dimension (mm)	Material
Coper plate	90 × 90 × 5	Copper
Cold plate cover	90 × 90 × 15	Acrylic
Fins	29 × 0.5 ×2	Copper
Inlet / outlet Ø	7mm	-

3.4 Meshing of Dynamic Cold plate model (DCP)

ANSYS Workbench then used for the simulation of the model. Firstly, the model was imported into the software as a geometry. This geometry is then transferred to mashing software named ICEM CFD. Here, we have to transfer the model for the meshing because for the model simulation meshing critical thing. This geometry of the model is carrying the shape smaller than 1mm. Hence, unstructured tetra element mesh is used for the simulation.

ICEM CFD is dedicated software for the meshing in ANSYS packages. The main feature of the software is to be able to generate structured and unstructured. Hence, this software is used for the meshing. Firstly, for the meshing the presented model of DCP inlet and outlet surfaces needed from an analysis point of view. ICEM CFD is providing the feature of generating thin 2D surface. Firstly, the surface was generated. After generating surface, the model needed to be mesh.

Firstly, ICEM CFD provides two approaches for meshing named Robust (Octree) and Quick (Delaunay). In the given application Robust (Octree) algorithm is used for meshing. Robust (Octree) algorithm adapts top-down approach for the meshing. It is evident from the name that top-down approach means line geometries will mesh first. Surface elements will be meshed after line elements and surface elements will be meshed last. Apart from the explanation, the key benefit of the algorithm is, unlike another algorithm, initial mesh on the surface is not required for this algorithm.

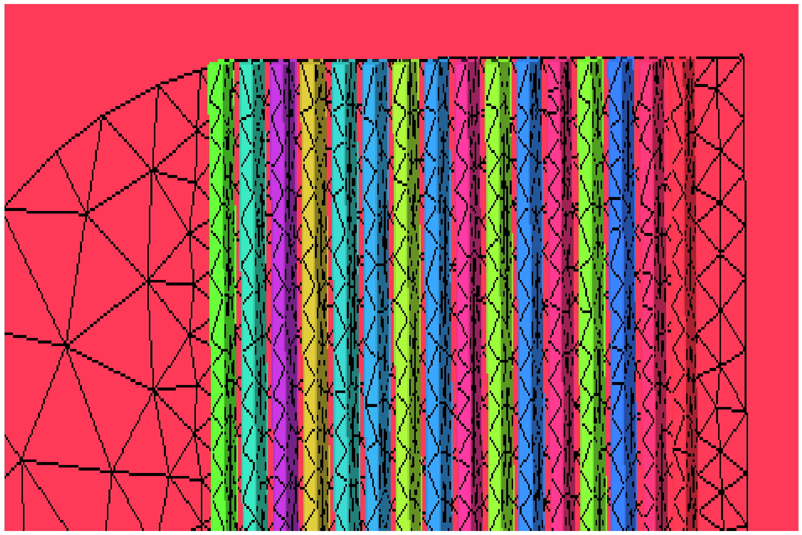

Figure 3-3: Mesh fineness of the DCP model

The meshing of the model starts with the global scale factor which is 1 for present model. Apart from that, same mesh for the whole model will not work. Because of the very small geometry of fins. Hence, surface mesh factor can be changed in ICEM CFD for the accurate mesh. Surface mesh factor is the factor for the software which decides the maximum length of a side of the element. Surface mesh factor for fins, heated plates, inlet and outlets is 0.001m. The reason behind keeping the factor as 0.001 is, to capture smallest side of the fin which is of 0.5cm. Copper plate and cover are having surface factor 1 because to capture the geometry of these two small elements of the size less than 0.001 is not needed. Apart from that, small size of the element implies more number of elements. And number of elements leads to more time of computation. The number of cells generated in the given specification was 11,000,000. Which is also quite high number for given model. If we use same surface fineness for whole model then number of cell would be very high.

Chapter 4 Boundary condition and Simulation

3.1 Boundary conditions for Oil immersion model.

The preceding part CFD analysis after meshing is boundary condition and simulation. Firstly, boundary conditions are the imperative part of CFD analysis. To generate the exact environment as the experiment, boundary conditions are needed. For the simulation of the model, the boundary condition can be given in ICEPAK software.

Fix mass flow rate and fix pumping power are two approaches can be used for the study. Flow rate is constant throughout the study in fix mass flow rate concept. Where in fin pumping power, pumping power is constant through the study. Pumping power for the can be computed from equation 1. Here, pressure drop is the major factor. The pressure drop for the air cooling system was 4 pa. Whereas in oil cooling parallel plate heat sink pressure drop was of 0.9 pa. However, for the pin fin and plate fin heat sinks the pressure drop is of the range of 0.04 pa. Hence, the simulation would give same result for both techniques.

The fix mass flow rate method is used for the given application because of technique's simplicity. In addition to that, ICEPAK gives facility to insert the flow rate. So, the method does not require any further calculation.

Table 4-1: Material Specification of oil immersion setup

Material name	**properties**
FR-4	Density: 1900 kg/m^3 Thermal conductivity: 0.17 W/m k Specific Heat : 749 J/kg k

Aluminum	Density: 2700 kg/m^3 Thermal conductivity: 218W/m k Specific Heat : 900 J/kg k
Mineral oil	Density: 1680 kg/m^3 Thermal conductivity: 0.13 W/m k Specific Heat : 1680 J/kg k

Firstly, as a part of boundary condition, parts of the model need to be provided with exact same material properties as an original model. ICEPAK facilities user to create the material and define the exact properties of the material. Apart from that, there are two openings in the model as per the experiment. As oil is interring the tank from the bottom and exits the tank towards the radiator from the top of the tank so there are two openings inlet and outlet as shown in the Figure 4-1. The oil is entering the tank initially at the rate of 1lpm. Hence, the velocity of the oil is calculated from the equation 4-1 and it came out to be 0.00157 m/s. Moreover, the oil is at an ambient temperature of 30°c. The pressure of the oil at inlet is 6psi. Furthermore, Reynolds number for the experiment is 13.6 from equation

$$mass\ flow\ rate = \ section\ area\ of\ tank \times velocity \tag{3-1}$$

$$Re = \frac{\rho v D}{\mu} \tag{3-2}$$

Re= Reynolds Number

P=Density

V=inlet velocity

(3-3)

$$D = \frac{2gh}{g + h}$$

g=channel width

h=fin Height

3.2 Simulation

Simulation is the last part of the CFD analysis after applying boundary conditions. According to this boundary condition, ICEPAK runs the calculations of the naiver-stocks equation for the values on each node and at last the software shows the values as a result. Basically, two parts of the naiver-stocks equation based on which calculation are conducted.

ANSYS ICEPAK provides the facility of parametric study as well. Parametric study means to measure the result by changing different parameters. Parametric study of the heat sinks can be done in the ANSYS ICEPAK. Parameters such as fin height, fin thickness, base height and number of fins were computed for the given research. Such simulation can give us optimized results for the selected parameter.

3.3 Boundary conditions (Warm water cooling)

Analysis of the mesh model of DCP is possible in ANSYS Fluent software. Firstly, the model has to be imported in Fluent. The first step for the simulation is checking mesh quality and report the mesh quality. This is trial and error method. If mesh quality is fluent then again model needed to be meshed in the ICEM CFD. However, for the mesh quality for the model at 11,000,000 cells was passed mesh check. Hence, the model and mesh are ready for the simulation.

Table 4-2: Power Specification of test set up of DCP

Component	Quantity	Power(W)
Base	1	-
ASIC	1(B1)	40
ASIC	11	5
FPGA	1	5
LICA	137	0

Firstly, fluent is fluid base software and all the meshed zones are considered as fluid zones initially. So, the first step is to separate fluid and solid zones in the model. After separation of the zones, boundary conditions are needed for the simulation. Secondly, boundary conditions are needed to be set for the model. The first boundary condition is defining inlet and outlet. In this model, the given inlet is velocity inlet as we r giving the mass flow rate to the inlet which is 4lpm. And inlet pressure is about 1000 pa. Whereas, the outlet of the model is pressure outlet because for the water to come out pressure difference is required between inlet and outlet. The pressure of the pressure outlet is 3000 pa. Lastly, the temperature of the inlet water is ambient. In addition to that, same as experimental model the power to the heat plates also provided. Are mentioned in table no 5-2. The power of the chip B1 is 40 w and other chips on the test board are at ideal state of 5 w. After applying all this boundary conditions. Our model is ready for the simulation set up.

3.4 Simulation set up for DCP

Model of the DCP is set for the simulation, as meshing and boundary conditions step is already done. So, for the simulation we need to first generate the exact environment in fluent for the accurate results. Heat transfer in our model is pure conduction, as heat generated heat plates / packages will be transferred to the copper base plate. Copper base plate will transfer the heat to fin and fin will transfer to flowing water. Hence, here we need to use Naiver-Stocks energy equation, momentum as well as continuity equations which are presented in as equations 4-4, 4-5 and 4-6

The energy equation used by the fluent can be represented as follow.

$$\frac{\partial}{\partial t}(\rho E)+\nabla \cdot (\vec{\vartheta} \cdot (\rho E + p)) = (\nabla \cdot \left(k_{eff}\right)\nabla T - \sum_j h_j \vec{J}_j + \bar{\bar{T}}_{eff}) + S_f \quad (3\text{-}4)$$

Here,

ρ = Density of the quantity

E= Energy

p= Pressure

$\vec{\vartheta}$ = Velocity vector

k_{eff}= $k + k_t$(effective conductivity; k_t is depending on the turbulent model)

$\vec{J}_j$= Diffusion flux

$\bar{\bar{T}}_{eff}$= Temperature

h_j=$\int_{T_{ref}}^{T} C_{p \cdot j}\, DT$

K-epsilon model used as the turbulent model in this simulation. This model is two equation model in which first equation is used for the finding turbulent kinetic energy k and another equation is used for the dissipation ε. The equations for the model are shown here.

Equation for turbulent kinetic energy k

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial t}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + S_k \qquad (4\text{-}5)$$

For dissipation ε

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial t}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + C_{1E}\frac{\epsilon}{k}(P_k + C_{3b}P_b) - C_{2_\epsilon \rho}\frac{\epsilon^2}{k} + S_e \qquad (4\text{-}6)$$

Here,

ρ=density of the fluid

k=kinetin energy

ϵ= total energy

u_i=velocity vector

μ_t=viscosity at temp t= $\rho C_\mu \frac{k^2}{\epsilon}$

S_k=rate of strain tensor

Reynolds number from the calculation came out to be 3057. And also because of complex geometry turbulence was evident in the model. Hence, rather than taking the simple laminar model here K-epsilon model was used.

The model is now ready for the simulation in the fluent. And it can be further used for the results and optimizations.

Chapter 5 Results

5.1 Results of Oil immersion model

Firstly, as explained earlier the validation of the model can be displaced by the graph at the ambient temperature. The validation of the model was studied in the previous thesis .

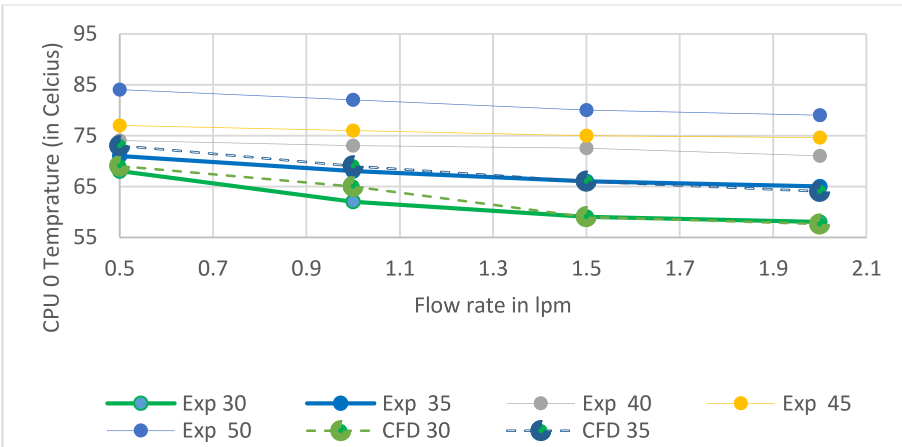

Figure 5-1: Validation with experimental data

The graph is of CPU temperature vs flow rate in lpm for the experimental and CFD analysis from the graph we can say that they are in the goodl agreement with each other. Hence, for the other results as well the same model was used. The other results can be presented as follow.

Figure 5-2 represents the greed independent study of the models in which both the model shows almost the same trend as near 1.5 million nodes the given models shows the accurate and constant results. The graph shows the number of nodes vs thermal resistance. For every study, thermal resistance is used as the standard for future use.

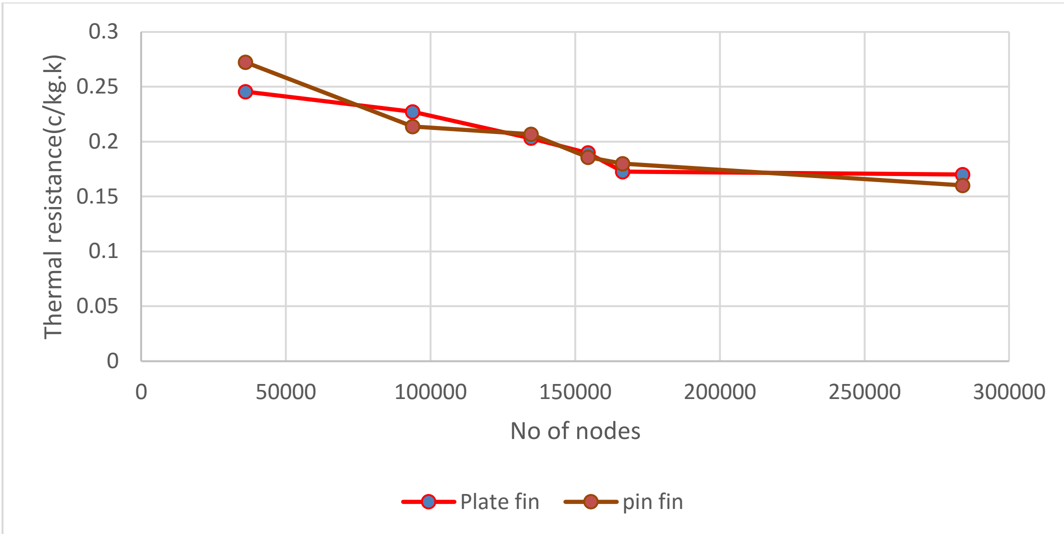

Figure 5-2: Grid independent study

Figure 5-3 shows the graph of flow rate vs thermal resistance. The purpose of the study for the model is to evaluate the optimum flow rate for the optimization of the heat sinks. From the graph that, after 1 lpm the graph of the thermal resistance is going towards constant hence, for the rest of the study 1lpm flow rate was kept constant.

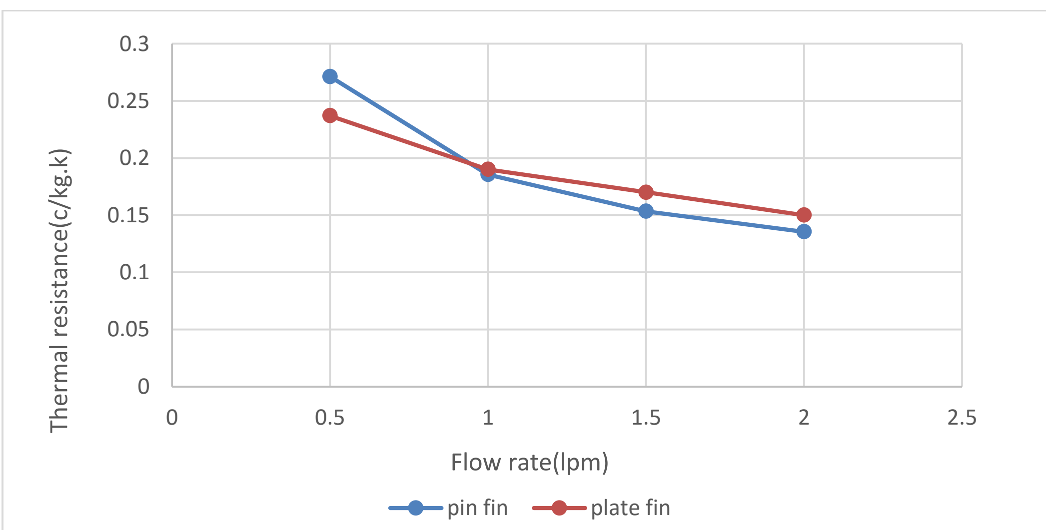

Figure 5-3: Flow rate vs Thermal resistance

The optimum range of the flow rate and a number of nodes would allow as to simulate the optimization of the heat sink for the further study. In the figure 5-4 it can be seen that thickness of the fin is optimum in the range of the 0.4 to 0.6 cm whereas height of the fin for the plate fin heat sink is optimum between 2.5 to 3.5 cm. Reason behind such small height is the nature of the oil of taking least resistance way. In higher than optimum range oil takes path over the REM and very less oil pass through the heat sinks as a result temperature of the chip is going high rather than going low.

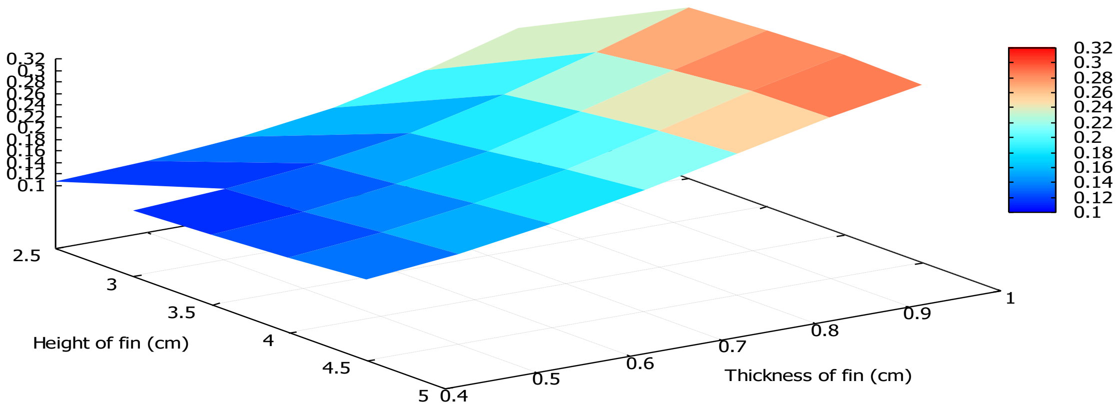

Figure 5-4: Geometrical Optimization of plate fin heat sink

Figure 5-5 shows the optimum range for the height and the diameter for the heat sink. Here, the optimum range of for the fin height is also the same as plate fin heat sinks which is 2.5 to 3.5 and redius of the fin can be changed in between the range of the 0.35 to 0.4. which can give the optimum result for the givan server.

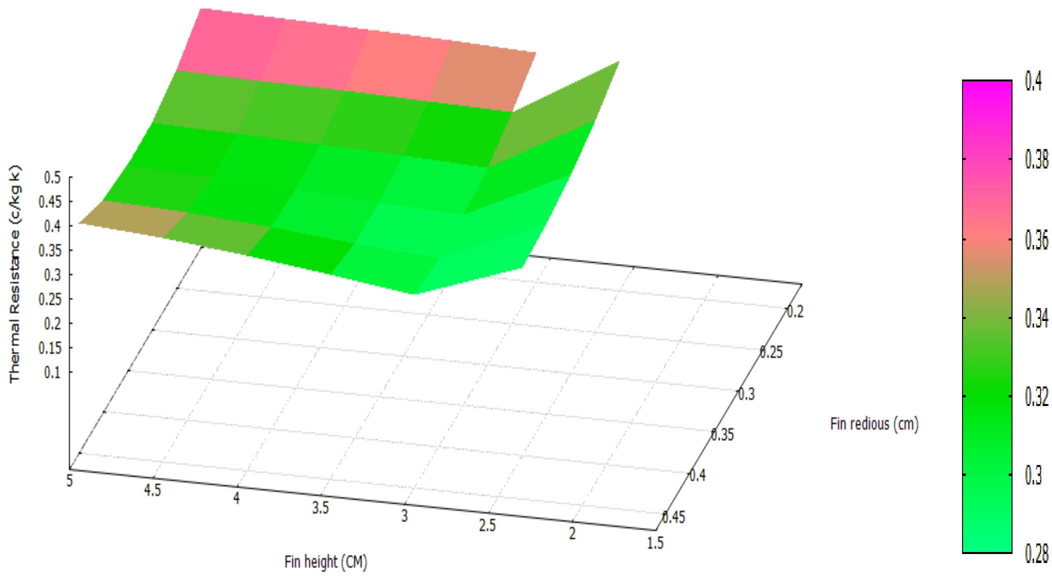

Figure 5-5: Geometrical optimization of pin fin heat sink

5.1 Results for DCP analysis

After simulating the model for the iteration around 500. Results are starting to converge, hence such results can be considered for the validations with the experimental results. The experimental results can be considered from the previous thesis.

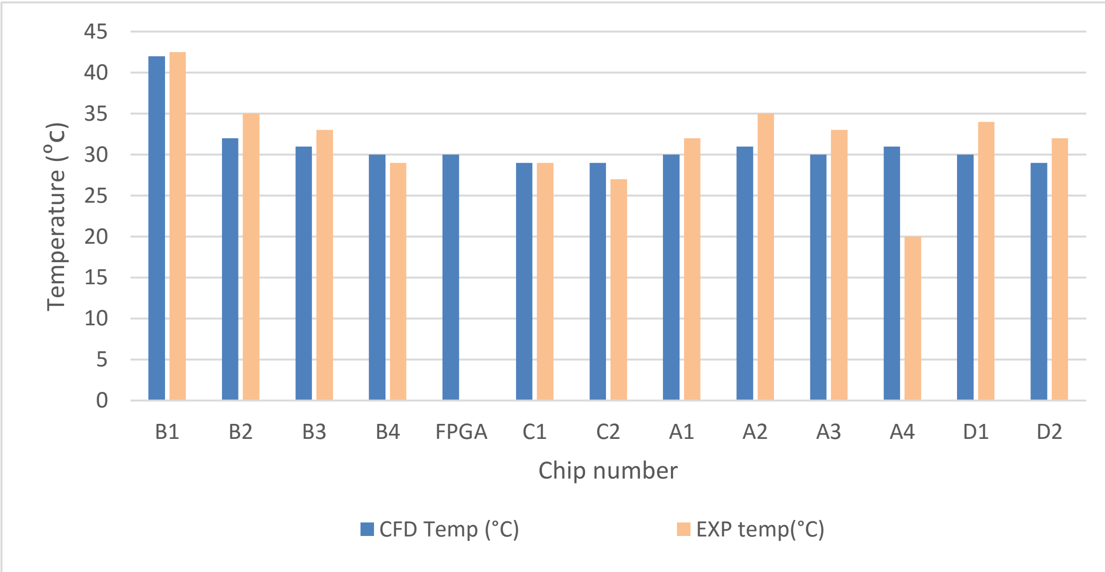

Figure 5-6: Validation of Dynamic cold plate model

Here figure 5-6 shows the temperature measurement for the experimental results as well as CFD model results. From the graph the results are in agreement. Because, all the ideal chips are nearly at the same temperature for both the cases. Chip B1 which is at high power is showing some remarkable result. As the temperature for the different cases for the chips are 42°c and 44.7°c.

Ideal chips are showing some uneven graph because of some manufacturing defects such as surface flatness of the copper plate, inter compartment leakage and less fins on the FPGA package. Hence, if such defects are rectified then it can be the exact same graph for both the cases. Moreover, figure 5-7 shows the temperature contour. Which also support the credibility of the model.

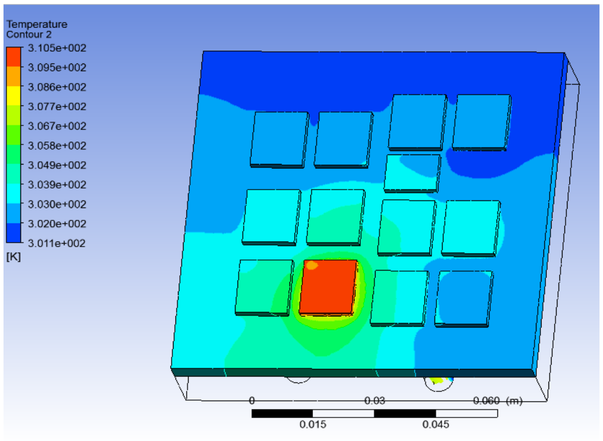

Figure 5-7: Temperature conture of DCP model

Hence, the conclusion of the study can be, This validated model can be used for the further optimization of the dynamic cold plate.

Chapter 6 Conclusion

6.1 Oil Immersion Setup

In oil immersion application, this research suggest for the server identical to Facebook open compute generation-1 the Range of the heat sink dimension can be in the range suggested. Apart from that, the research also contribute to the future researchers a model which can be used for the further study of the optimization of such server. Such guideline can be used for the documentation of the data for further use of the datacenter developers.

6.2 Dynamic Cold Plate setup

The research presented is the initial Step towards the optimization of the DCP design for the better performance. Apart from that, the guideline presented here can also be useful for researchers investigating the same model as presented here.

ferences

1. R. Mahajan, C.-P. Chiu and G. Chrysler, "Cooling a microprocessor chip," Proceedings of the IEEE, vol. 94, no. 8, pp. 1476-1486, 2006.

2. B. Agostini, M. Fabbri, J. E. Park, L. Wojtan, J. R. Thome and B. Michel, "State of the art of high heat flux cooling technologies," Heat Transfer Engineering, vol. 28, no. 4, pp. 258-281, 2007.

3. L. A. Barroso and U. Holzle, "The case for energy-proportional computing," Computer, vol. 40, no. 12, pp. 33-37, 2007.

4. D. Copeland, "Review of Low Profile Cold Plate Technology for High Density Servers," 1 May 2005. [Online]. Available: http://www.electronics-cooling.com/2005/05/review-of-low-profile-cold-plate-technology-for-high-density-servers/. [Accessed 23 January 2015].

5. J. Fernandes, S. Ghalambor, D. Agonafer, V. Kamath and R. Schmidt, "Mutli-Design Variable Optimization for a Fixed Pumping Power of a Water-Cooled Cold Plate for High Power Electronics Applications," in IEEE Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems, San Diego, CA, USA, 2012.

6. U. Hwang, K. Moran and R. Kemink, "Cold Plate Design for IBM ES/9000 TCM Electronic Modules," Advances in Electronic Packaging, pp. 75-81, 1992.

7. J. Fernandes, S. Ghalambor, A. Docca, C. Aldham, D. Agonafer, E. Chenelly, B. Chan and M. Ellsworth, "Combining Computational Fluid Dynamics (CFD) and Flow Network Modeling (FNM) for Design of a Multi-Chip Module (MCM) Cold Plate," in ASME International Electronic Packaging Technical Conference and Exhibition, Burlingame, CA, USA, 2013.

8. R. Remsburg, "Nonlinear Fin Patterns Keep Cold Plates Cooler," 1 February 2007. [Online]. Available: http://powerelectronics.com/thermal-management/nonlinear-fin-patterns-keep-cold-plates-cooler. [Accessed 23 January 2015].

9. D. Soodphakdee, M. Behnia, and D. Copeland, “A Comparison of Fin Geometries for Heatsinks in Laminar Forced Convection: Part I - Round, Elliptical, and Plate Fins in Staggered and In-Line Configurations” *The International Journal of Microcircuits and Electronic Packaging*, vol.24, p 68-76, 2001

10. D. Copeland, “Fundamental Performance Limits of Heatsinks”, Journal of Electronic Packaging, Vol 125, p. 221-225, Available: http://electronicpackaging.asmedigitalcollection.asme.org/ on 08/03/2014 Terms of Use: http://asme.org/terms

11. R. Eiland, J. Fernandes, M. Vallejo, D. Agonafer, V. Mulai, “Flow Rate and Inlet Temperature Considerations for Direct Immersion of a Single Server in Mineral Oil”, 14th IEEE ITHERM Conference, 2014, p. 706.

Biographical Information

Parth Jayeshkumar Soni received his bachelor's degree in Mechatronics Engineering from Gujarat Technological University Ahmedabad 2013. In august 2014, he began his masters at The University of Texas at Arlington, after working for the Parag Engineering, Ahmedabad for 1 Year. Parth Joined EMNSPC team for the research on thermal management for the data center and IT equipment. He completed his MSc degree in Mechanical Engineering from the University of Texas at Arlington in May 2016