CFD MODELLING OF WET COOLING MEDIA AND DESIGN OPTIMIZATION OF HEAT SINK FOR DATA CENTER APPLICATION

By

SHIVANG YOGESHKUMAR DESAI

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 like to thank my supervising professor Dr. Dereje Agonafer for his support throughout my research work. He gave me a platform to improve my skills and abilities in computational as well as in experimental areas of this field. He has been a great teacher in molding my expertise into the professional engineer by giving me an opportunity to work and present to great people of the industry.

I would like to thank Dr. Haji Sheikh and Dr. Miguel Amaya for serving on my committee with valuable comments and inputs. I am thankful to Ms. Sally Thompson who has been a great help in educational and administrative matters. I would like to thank my friend Parth Soni and Mansi Prajapati in assisting me from exploring stuff to making presentations. I shared valuable moments of my journey at UTA with them and I appreciate them for being there with me. I am also thankful to my project mates Jimil Shah and Ashwin Siddarth and other EMNSPC team members for their support.

I am obliged to MAE faculty members who have been a large pool of knowledge and resources. I am thankful to Ms. Debi Barton for her support in educational matters from day one at this university.

Special thanks to my parents Mr. Yogeshkumar Desai and Mrs. Santoshben Desai for giving me immense support emotionally and bringing best out of me. Finally, I would acknowledge each and every friend who have made my journey here a memorable one.

March 21, 2016

Abstract

CFD MODELLING OF WET COOLING MEDIA

AND DESIGN OPTIMIZATION OF HEAT

SINK FOR DATA CENTER

APPLICATION

Shivang Yogeshkumar Desai, MS

The University of Texas at Arlington, 2016

Supervising Professor: Dereje Agonafer

Air cooling is most widely used and economical method of currently available cooling technologies in which heat exchanger plays a vital role in facility designs. Evaporative cooling can be energy efficient than mechanical cooling because it uses water's enthalpy of vaporization. Wet cooling pads can be used as the direct evaporative cooling system in which pad is made of cellulose fibers those gets wet when water injected. When hot and dry air passes through pad surfaces, water evaporates in the air which cools down air adiabatically increasing humidity. These type of cooling units can be installed in these type of climates for a use of modular data centers. Wet cooling pads are studies for greenhouse and wind tower application using system resistance model or equivalent representation of media using the box. This study focuses on compact CFD model of wet cooling pad representing an actual model of the pad surfaces. Surfaces are represented with the equivalent porous medium. These type of modelling can be used free of pad dimensions and flute angles for fixed material.

The other part of this study focuses on heat sink customization for application of oil immersion cooling in data centers. Oil immersion cooling can be promising cooling

technology for high thermal loads and heat capacitance. At server level, heat sink is critical part for cooling efficiency and it is been studied widely for air cooling systems. There is a lack of literature work in studies involving heat sink for the oil immersion cooling application. This work is to provide an efficient range of operation for the heat sink with numerical modelling for first generation open compute server. Customization of the heat sink can open up to more heat load per rack. A parametric study is conducted on fixed pumping power and thermal efficiency of the heat sink for oil immersion cooling has been optimized to a cost saving value of 44%.

Table of Contents

List of Illustrations

List of Tables

NOMENCLATURE

CFD	Computational Fluid Dynamics
DPM	Discrete Phase Model

CHAPTER 1 THESIS ORGANIZATION

This research work is dedicated to cooling performance enhancement for application of data center. Data centers are equipped with air cooling from a long time. Air cooling is most traditional and continent method amongst all cooling technologies. Another side of the industry is trending towards high-performance computing where heat load and power demand is very high. This kind of thought is mapping onto future. For very high capacitance air cooling becomes a costly and inconvenient technique. In these sides of industry liquid cooling is becoming an important player of the cooling market.

The research area of this thesis is focused on to two distinguished areas of cooling technology. The title of the thesis is made up of two parts. The first part is CFD modeling of wet cooling media for application of evaporative heat exchangers. The second part of research work is design optimization of parallel plate heat sink for oil immersion cooling application.

1.1 CFD Modelling of Wet Cooling Media

The first part of the thesis is focused on CFD modelling of wet cooling media. Wet cooling media is used in evaporative cooling towers. CFD modelling is complex work in modelling wet cooling media because they involve two-phase evaporation modelling and material modelling having fiber type of confirmation. This research gives modelling approaches and validation against experimental data.

Second chapter is dedicated to introduction of wet cooling media. This gives an idea of evaporative cooling systems and use of these in data center against conventional chiller plants. This section also covers the difference between direct and indirect evaporation cooling system and how to get the maximum benefit of both the systems in a single unit. Third chapter focuses on literature review of wet cooling media applications and CFD modeling practices for those. Fourth chapter focuses on CFD methodology and discussion of research phases. Fifth chapter is focused on design consideration and CFD results validation. Sixth chapter discusses conclusion and future work that can be extracted using this modelling approach.

1.2 Design optimization of Parallel plate heat sink for oil immersion cooling application

The second part of the thesis is heat sink optimization of parallel plate design for oil immersed server. This study is crucial where server technology is needed to be retrofitted from air cooling to oil immersion cooling. Oil immersion cooling is the new trend in server cooling. Heat sinks have not been previously studied for this type of cooling technology. Seventh chapter discuses on oil immersion cooling, use of mineral oil and the importance of heat sink in server design. The eighth chapter gives an idea of literature survey in this area. Ninth chapter is focused on CFD modelling and validation along with grid independence study. The tenth chapter discusses on optimization of the heat sink by fixed volume flow rate and fixed pumping power. It also gives an idea of an orientation of the server and differences. Eleventh chapter is focused on the conclusion and future work of the work.

CHAPTER 2 INTRODUCTION TO DIRECT EVAPORATION COOLING

The direct evaporation cooling technology is a part of evaporative cooling as a method of cooling and it is very different from adiabatic cooling. Evaporation cooling word is self-explanatory as it uses evaporation process in order to cool the system down. Vaporization is a phase transition process in which phase of liquid is changed to vapor by some process. Boiling and evaporation comes into this category of phase transition processes. Two processes are different in the sense of vaporization at the surface or at the volume level. Evaporation is phase change from liquid to vapor like boiling, but it acts on the surface. Vapor is a state in which substance stays below its critical temperature and pressure. In the atmospheric air, there is always water vapor molecules are present. So, atmospheric air pressure is a combination of the partial pressure of air molecules and partial pressure of water vapor molecules. This partial pressure of water vapor in the air is an important factor in consideration of evaporation phenomena. When equilibrium vapor pressure is more than the partial vapor pressure of air, then evaporation process occurs in the liquid phase. One example of evaporation is water in glass to surroundings. Some of the water molecules have energy that is enough to escape from water or it is vice versa. But it is dependent upon saturation of air. If it is not 100% then water will escape from water to the air. The same process happens in a hot cup of tea, when water particles will escape from surface and evaporate into the air because of difference in partial pressure of air. Water droplets due to evaporation are shown in figure 1. Evaporation cooling falls into the broad category of adiabatic cooling where heat and mass transfer is the implication of work done by the system. In real environment adiabatic process is impossible, but if the process is very rapid then it can be said to be adiabatic. In evaporation cooling air and water are called one system in which if air is cooling through water evaporation, then air pressure reduces. This pressure reduction of air allows it to increase volume and decrease in internal energy. Adiabatic cooling is shown from P-V diagram which is shown in figure 2. This evaporation concept is used in the cooling system called as a swamp cooler. Earliest use of swamp cooler can be dated back to Egypt and Persia thousand years ago.

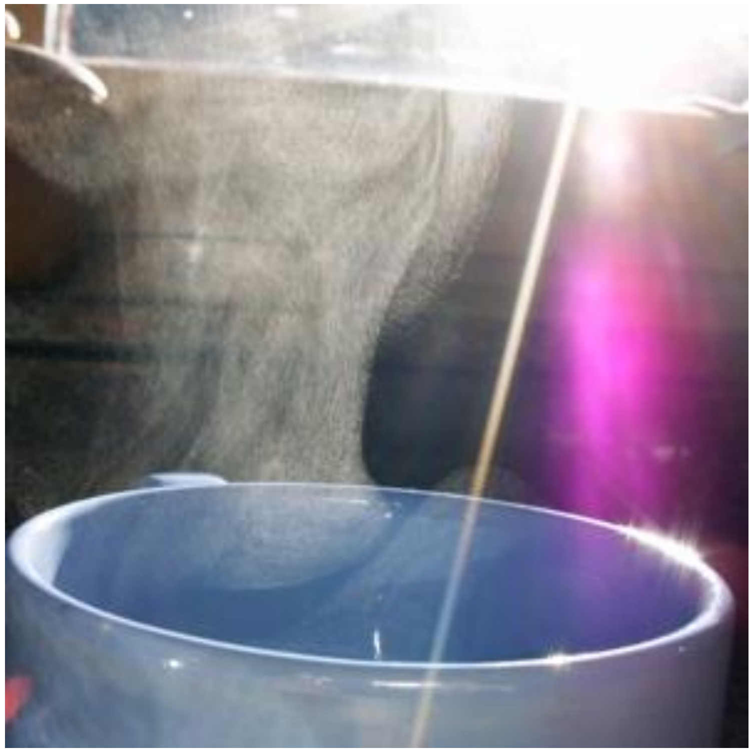

Figure 1 Evaporation of water droplets above hot tea cup [19]

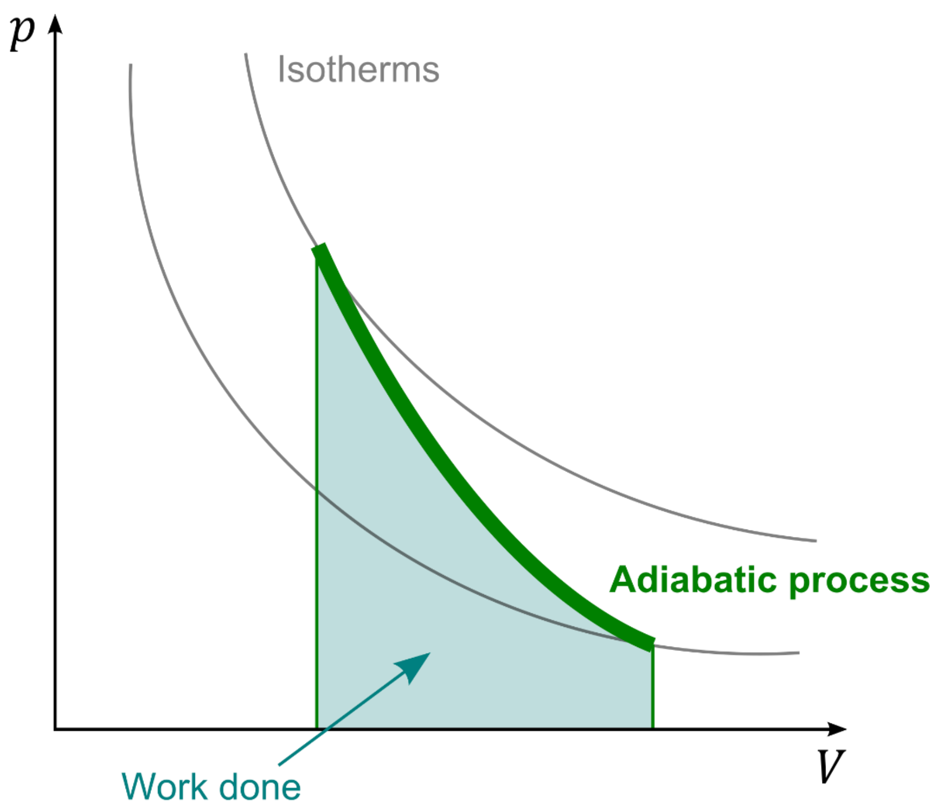

Figure 2 P-V diagram of adiabatic cooling process [20]

This type of coolers did not use pad configuration instead it used water reservoir called Qunat. Air from roof allowed to pass from Qanat and then it cools down adiabatically from evaporation process. This type of evaporative cooling in ancient times is shown from figure 3.

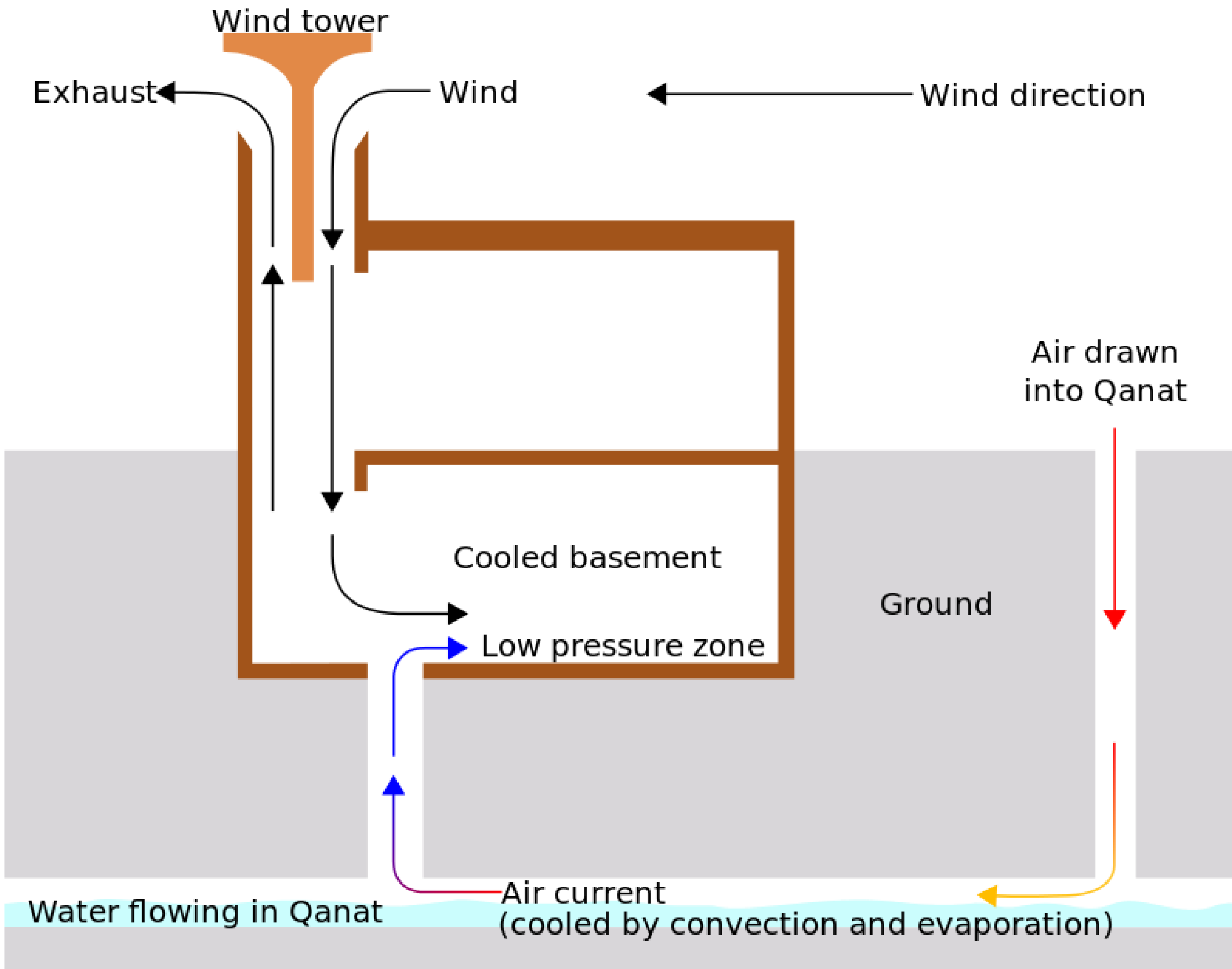

Figure 3 Ancient evaporative cooling system [21]

In recent times, evaporative cooling is used by two methods.

- Direct evaporation
- Indirect evaporation

Direct evaporation is the system which used by ancient Iran and Egypt civilizations. This type of evaporation occurs when air comes into direct contact with water. Nowadays direct evaporation cooling system is implemented by wooden pads. This type of pad is called wet cooling pad which is shown I figure 4.

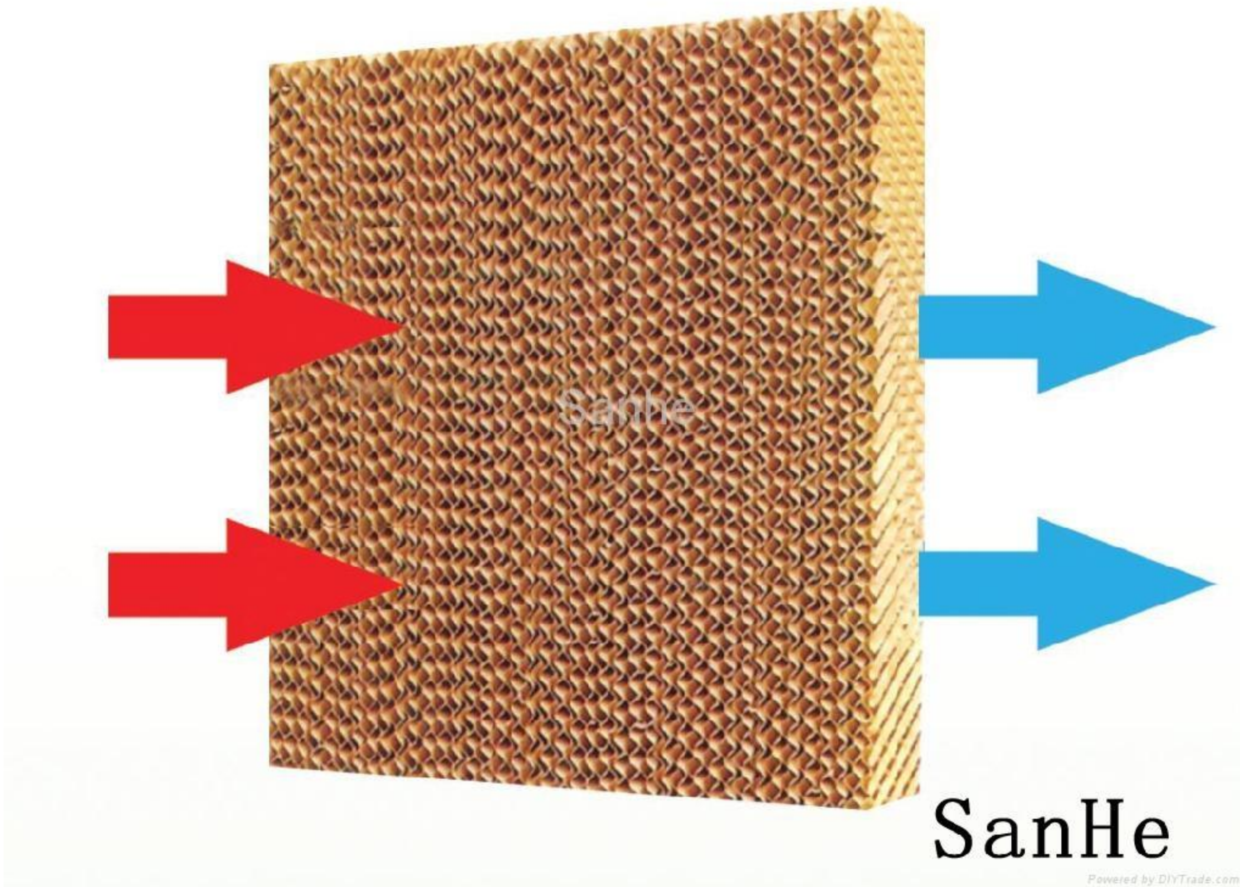

Figure 4 wet cooling pad [22]

Wet cooling pads are used in swamp cooler system. Water is injected from the top of the cooling pad by a pump which wets the pad. Water is stored in the reservoir and it is controlled by the float valve. The centrifugal pump draws the ambient air from wet pads towards the building. Swamp coolers can be useful in hot and arid areas like Indian subcontinent and Latin America. Direct evaporation cooling uses evaporation of water into the air. It is different from vapor-compression refrigeration mechanism. Refrigeration cooling also uses evaporation to cool down hot air. But coolant fluid does not evaporate into surroundings instead it is being reused again into the system by compression and condenser cycles. This type of mechanism is controlled in a manner of temperature and it is being used in refrigerators and air-conditioners. This mechanism of cooling provides high performance in humidity and less condensation of water particles inside facility air. Data center cooling is most essential part of energy consumption chart. Cooling technologies are challenging task to design and implement. Current cooling provision of data center ranges from indoor CRAC units to outdoor chillers and cooling towers. These cooling systems can be equipped with air, water or glycol. Chiller cooling is frequently used which uses chilled water as coolant fluid. Chiller uses same mechanical cooling mechanism as explained earlier. This system involves complex ducting, compressor, pump and condenser configuration and they consume lots of power. Evaporative cooling can be a promising cooling solution for IT equipment heat removal. Evaporative cooling uses fresh incoming air as coolant fluid rather than costly and pollutant refrigerants. Evaporative cooling uses the pump, fan and water reservoir system which is easy to maintain, lower electricity consumption and easy installation even can be used in the modular data center as well as a large data center. The evaporative cooling technique should be weather smart means it should adapt the nature of the weather condition like temperature and relative humidity. Thus, evaporative cooling restricts as per weather condition and the system should cool down air to operate IT facility in operating envelope over the psychometric chart. Water is most commonly used coolant fluid in evaporation cooling because it has the highest enthalpy of vaporization.

When the climate is hot and dry than wet cooling pad can be used to cool down fresh air for a use of IT equipment. Cooling of It equipment by fresh air is restricted by temperature and humidity conditions. These conditions are recommended by psychometric chart provided by ASHRAE which is shown in figure 5 [9]. These envelopes are operating conditions for data center when using evaporative cooling. When humidity in atmosphere is relatively high then critical value then direct evaporation may not be used. Indirect evaporation cooling is heat exchanger coils which cool down air without increasing its humidity. This type of evaporation system can be combined to get optimal cooling in operating envelope. Advantages of evaporative cooling are easy installation and cheap maintenance. The increase in humidity is advantageous for human comfort and reduces static electricity in IT equipment.

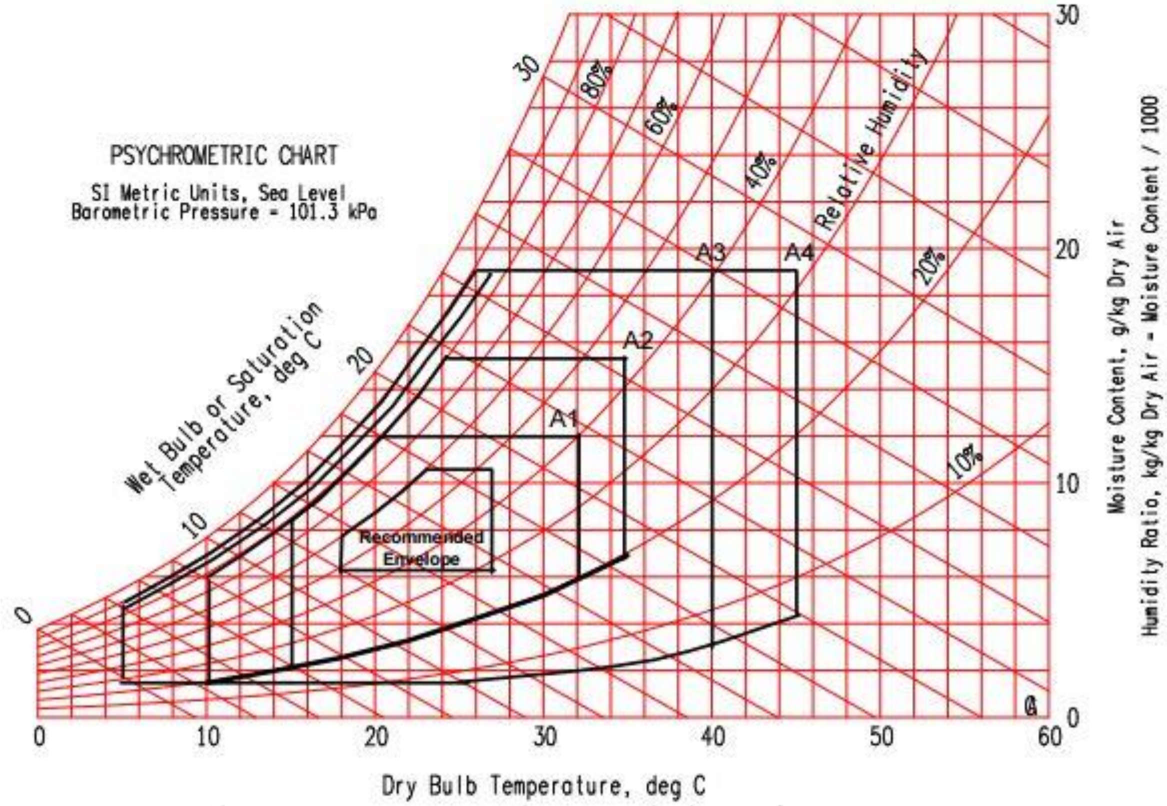

Figure 5 Psychometric chart recommended envelope [9]

CHAPTER 3 LITERATURE SURVEY

Wet cooling pads are being used in greenhouses along with fans to cool air for crop harvesting [1]. These cooling systems are installed in dry and hot climates. This system includes an exhaust fan at the end of a greenhouse which draws air through cooling pads. Cooling pads are supplied with water through water distribution pipe at the top of the pad. Water is collected in a sump at the bottom of the pad which is used to recirculate by a pump. If evaporative pad efficiency is known then the temperature of out coming air can be calculated from following equation 1 [23].

$$Efficiency = \frac{Tcool - Tdry}{Twet - Tdry} \tag{1}$$

Where:

$Tcool$ = temperature of air exiting pad

$Tdry$ = dry bulb temperature of incoming air

$Twet$ = wet bulb temperature of incoming air

CFD modelling of greenhouse cooling system is simplified by using source terms in modelling [2]. Direct evaporation is the transformation of latent heat to sensible heat. Thus, pads are taken as the mass source term in CFD modelling. According to given efficiency and latent to sensible heat ratio, output air temperature and pressure drop can be calculated in the model [2]. In this approach ventilation rate is provided in advance. But this lacks water usage calculation.

Water droplets in wind tower CFD model can be modelled with dispersed droplet phase using Lagrangian equation [3]. Two phase modelling is challenging in evaporative cooling and a numerical model is a critical factor in the calculation. Discrete droplets require the velocity of droplet, drag coefficient, droplet diameter, and droplet density. Droplet particles are tracked with trajectory equation within the continuous phase. The model works very well for two-dimensional heat transfer within the cooling tower.

Wet cooling media is modeled using equivalent porous material approach [4]. The temperature of air from pads can be calculated from saturation efficiency. The porous material is modeled as black box giving it physical characterization. Permeability and porosity are parameters that needed in the model. Permeability is a ratio of pore volume to total volume. This type modelling is suitable for estimation of pressure drop and air temperature simultaneously. Inlet conditions are velocity, the temperature of air and water.

CHAPTER 4 CFD MODELLIG METHODOLOGY FOR WET COOLING MEDIA

CFD modelling of wet cooling media involves modelling of water which is two-phase flow field. In this research work commercial CFD software ANSYS Fluent is used for modelling approach. Two phase modelling involves evaporation modelling. There are models of evaporation available in CFD. This work gives validation of evaporation model with the present analytical model. By further going material of fibre representation is also a complex part. For which porous medium concept is used. This section covers the porous material property calculation and modelling practices. Due to the involvement of heat transfer/ mass transfer concept along with material complexity this modelling is divided into three parts. Three parts are as follows.

- Part-1 Study of evaporation and validation of models
- Part-2 Porous media approach and property calculation with simple model
- Part-3 Real world model of media and validation with experimental data

This chapter is dedicated to part-1 and part-2 of the research work.

4.1 Study of evaporation models

Evaporation involves heat and mass transfer between two components. Water changes phase from liquid to vapor and mass of vapor is transferred into incoming air. Meanwhile liquid water absorbs energy from air and reduces its latent heat. Reduction of enthalpy causes temperature reduction in the air and heat transfer occurs between two components. Definition of evaporation can be stated as: “evaporation is the process in which a liquid phase is changed to the gaseous state at the free surface of liquid state, below the boiling point through the transfer of heat energy”. In the evaporation process rate of evaporation is depend on vapor pressure which is shown in following equation 2 [24].

$$E = C(e_w - e_a) \quad (2)$$

E=Rate of evaporation

e_w= Saturation vapor pressure of air at water temperature

e_a=Actual vapor pressure in air

C =Constant

Other factors for evaporation are temperature, air speed and atmospheric pressure. Thus analytical rate of evaporation can be calculated for simple systems. Analytical model for evaporation rate is taken from Shah's model [5]. Shah's model for evaporation gives estimate of evaporation from free water surface of swimming pool into air. Equation of evaporation rate of this analytical model is as follows in equation 3.

$$E = C\rho_w(\rho_a - \rho_w)^{\frac{1}{3}}(W_w - W_a) \quad (3)$$

$Where, E$=evaporation rate

ρ_w=Density of saturated air at water temperature

ρ_a=Density of air at current condition

W_w=Specific humidity of saturated air at water temperature

W_a=Specific humidity of air at current condition

Wet cooling media represents same as swimming pool model. The material of the media fills up with water particle in wetting phenomena and water film forms on a surface which represents as free surface of the water in swimming pool. The air passes over and inside fibers come in interaction with the free water film and evaporation rate from Shah's model can be used implementing following assumptions.

I. One of the many media surface is considered
II. Air flow is parallel to the air flute angle (in this case 15° upward)
III. Material is solid and flow passes only over, not below or inside

Accounting these assumption for wet cooling media, we can use Shah's model for validation purpose in this research. This model is for quite a flow over water surfaces. So laminar model can be used for CFD modelling.

4.2 CFD models for evaporation

There are four models available in the Fluent for multiphase modelling. The list of the models are given and why they cannot be used in this work has been discussed as follows.

I. VOF model

VOF stands for volume of fluid. VOF model is used for application of two immiscible fluid interacting together in the system. This model does not provide heat and mass transfer. So, it can't be used for modelling evaporation.

II. Mixture model

Mixture model is best suitable for particle laden flows where one phase is solid. This model does not provide species transport in modelling.

III. Eulerian model with continuous phases

Eulerian model in ANSYS Fluent is full scale multiphase model with every features. It treats every component in the system as continuous phase. Thus it can't track the water phase inside pores of the material. In this research pores are also critical part of the study and it is modelled as porous medium at local level which is discussed further in this chapter. Eulerian model can't be used in this study for this reason.

IV. Discrete Dense Model

Discrete dense model uses Euler-Lagrange approach which treats air as continuous phase and water as discrete droplets. Equation of the particles are solved in Lagrangian frame and it uses force balance to solve particle trajectory.

Trajectory equation is as follows in equation 4.

$$\frac{du_p}{dt} = F_D(u - u_p) + \frac{g_x(\rho_p - \rho)}{\rho_p} + F_x \tag{4}$$

$where, u$=Fluid phase velocity

u_p=particle velocity

$F_D(u - u_p)$=Drag force

F_x=Additional acceleration

According to this equation velocity of particle is calculated and trajectory is calculated. Droplets are modelled using spherical law. Droplets are introduced in the system using droplet injection method. They are injected through surface. Evaporating species are taken as h_2o in the model. Evaporation of 2nd

phase is accounted using law 2 of heat and mass transfer. Law 2 is called law of vaporization in which droplet evaporates until temperature is lower than vaporization temperature. Mass transfer is governed by convection-diffusion controlled model.

Advantages of DDM:

- Can be used with porous medium
- Tracks the particle splashing
- Species modeling available

4.3 Swimming pool test case and Flow conditions

To validate the discrete dense model, a test case with simple geometry is considered here. Swimming pool type configuration has been taken into account [6]. Here large swimming pool is considered with respect to inlet and outlet.

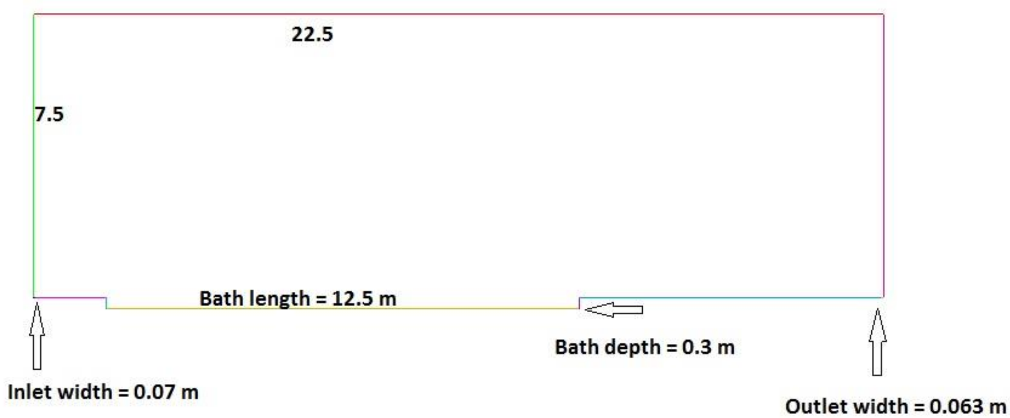

Figure 6 Swimming pool test case

Geometry with all dimensions of the test case is shown in figure 6. All dimensions are given in m. Bath represents the swimming pool surface. Flow conditions are given in table 1.

Table 1 Swimming pool test case Flow conditions

INLET	
Temperature	35° C
Relative Humidity	28%
Mass fraction of water	0.00979 kg
Velocity	1.86 m/s
Swimming pool (water)	
Water	Droplets are injected at pool boundary
Temperature	27° C
Mass flow rate	0.01 kg/s

Discrete dense model is used here for multiphase modelling. Injection of this model is used for water in swimming pool.

4.4 Governing equations and Gibbs phase rule

ANSYS Fluent solves continuity, momentum and energy equation along with species transport and discrete particle tracking from DDM. Equation 5 represents continuity equation used for this case [25].

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho \vec{V}\right) = S_{DPM} + S_{Other} \tag{5}$$

Where,

ρ = density of fluid phase

$\vec{V}$=velocity in vector form

S_{DPM}=Discrete phase model source

S_{other}=additional mass source

Momentum equation is shown in equation 6 for this case. Here acceleration due to droplet particle and viscous forces are considered [25].

$$\frac{\partial \rho \vec{V}}{\partial t} + \nabla .\left(\rho \vec{V}\vec{V}\right) = -\nabla p + \nabla . \tau + \rho \vec{g} + \vec{F}_{DPM} \tag{6}$$

Where,

p=Static pressure

τ=stress tensor

$\vec{g}$=gravitational acceleration

$\vec{F}_{DPM}$= DPM force acceleration

Water vapor in the air is modelled using species transport in ANSYS Fluent. Species transport equation uses convection-diffusion phenomena to transport water vapor in the air. The general equation is shown in equation 7 [25].

$$\frac{\partial \rho Y_i}{\partial t} + \nabla .\left(\rho \vec{v} Y_i\right) = -\nabla . \vec{J}_i + S_i \tag{7}$$

Where,

Y_i=local mass fraction of each species

$\vec{J}_i$=diffusive flux

S_i=creation of species by DPM

Heat transfer is governed by energy equation which is shown in equation 8 [25].

$$\frac{\partial \rho E}{\partial t} + \nabla .\left(\vec{v}(\rho E + p)\right) = \nabla .\left(k_{eff} \nabla T - \sum_j h_j \vec{J}_j + \bar{\bar{\tau}}_{eff} . \vec{v}\right) \tag{8}$$

Where,

E=enthalpy

k_{eff}=effective thermal conductivity

$\vec{J}_j$=diffusive flux due to species

When dealing with the multiphase flow it is needed to determine known variables to solve the system. Gibbs phase rule shows a number of variable needs in the system by equation 9. According to this equation, variance needs to solve the system are 3 [26].

$$f = c - p + 2 \tag{9}$$

Where,

f=intensive degrees of freedom

c=components

p=number of phases

4.5 Evaporation model validation and results

Using above governing equations for the flow simulation with enabling species transport and DPM model, steady state analysis has been carried out. Evaporation rate has been calculated from the discrete dense model. In figure 7 ANSYS Fluent window is shown where discrete particles status is shown by red line. As a sample, 69 water droplets are injected from pool boundary and all particles are evaporated.

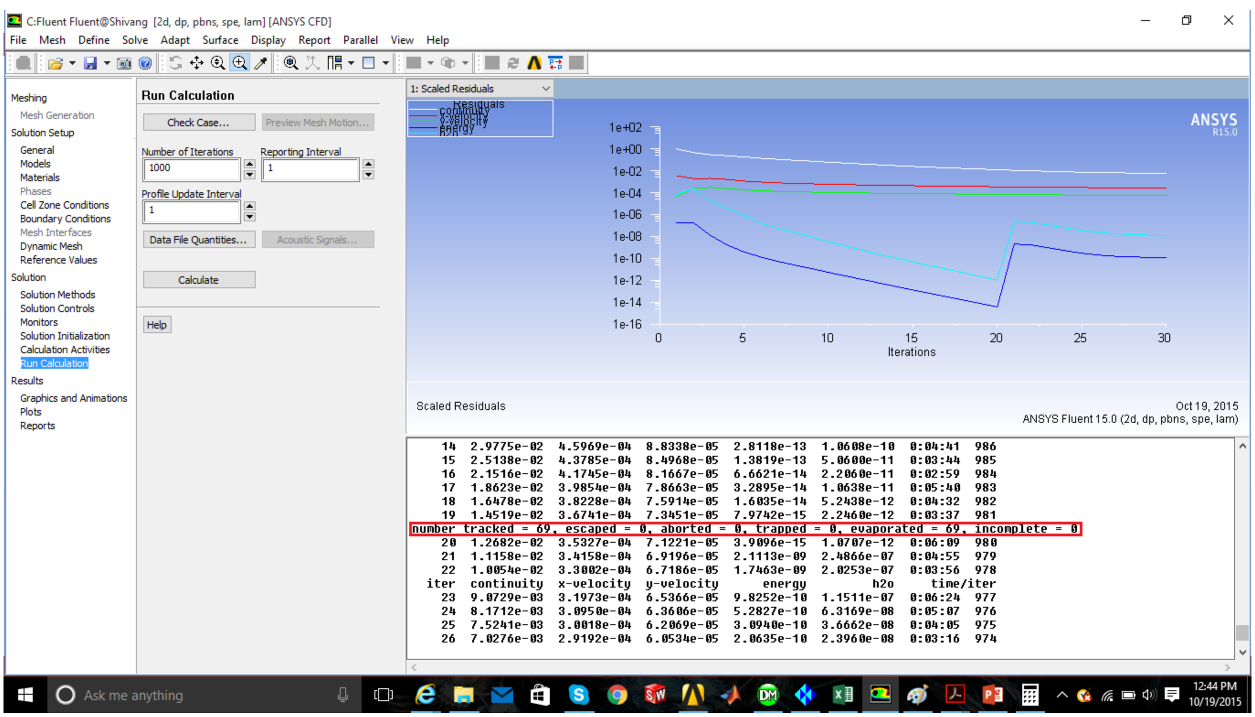

Figure 7 DPM particle status in ANSYS Fluent window

From h_2o mass calculation per unit square meter surface of the pool in unit time, evaporation rate can be calculated. From equation 3 analytical evaporation rate has been calculated and compared with

CFD in table 2. The error between CFD and analytical is 3.2%. Which is less than 10% threshold for validation purposes. DPM model can be used for evaporation modelling for this research work.

Table 2 Evaporation rate comparison for swimming pool case

Method	Evaporation Rate in $\frac{kg}{m^2 s}$
Analytical	0.00296
CFD	0.00304
Error%= $\frac{Analytical-CFD}{Analytical}100\%$=3.2%	

Temperature contour of a test case is shown in figure 8 below. Inlet air temperature is reduced from 308 K to 303 K due to evaporative cooling over a pool. After the middle of the pool weak recirculation is occurring and mixing of vapor in this area is not significant.

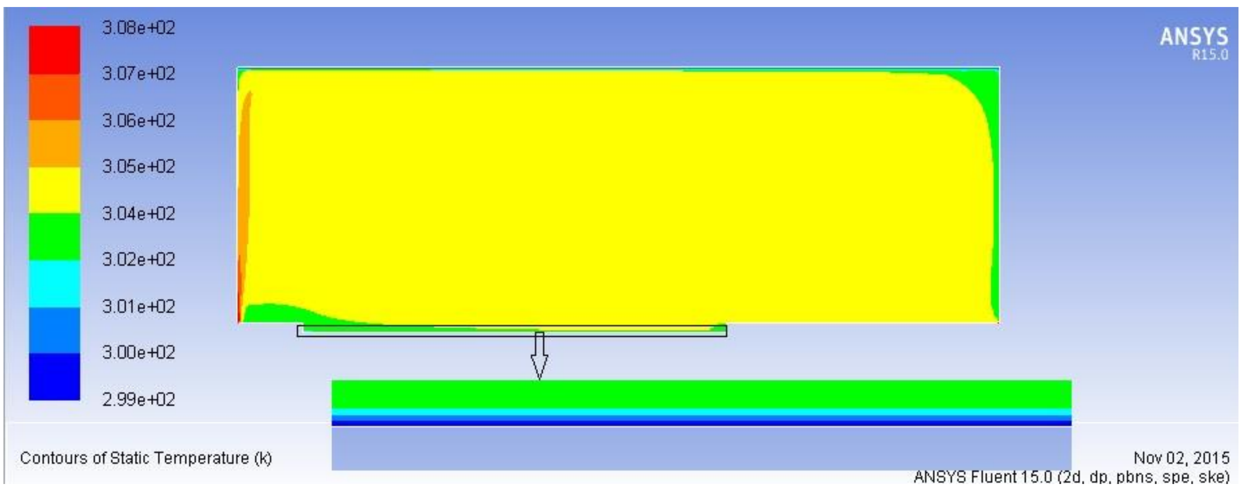

Figure 8 Temperature contour of Swimming pool test case

Likewise, Relative humidity contour is shown in figure 9. It indicates that water vapor is evaporating above pool surface. Relative humidity of air is increasing from 28% to 53 % which can be seen at outlet.

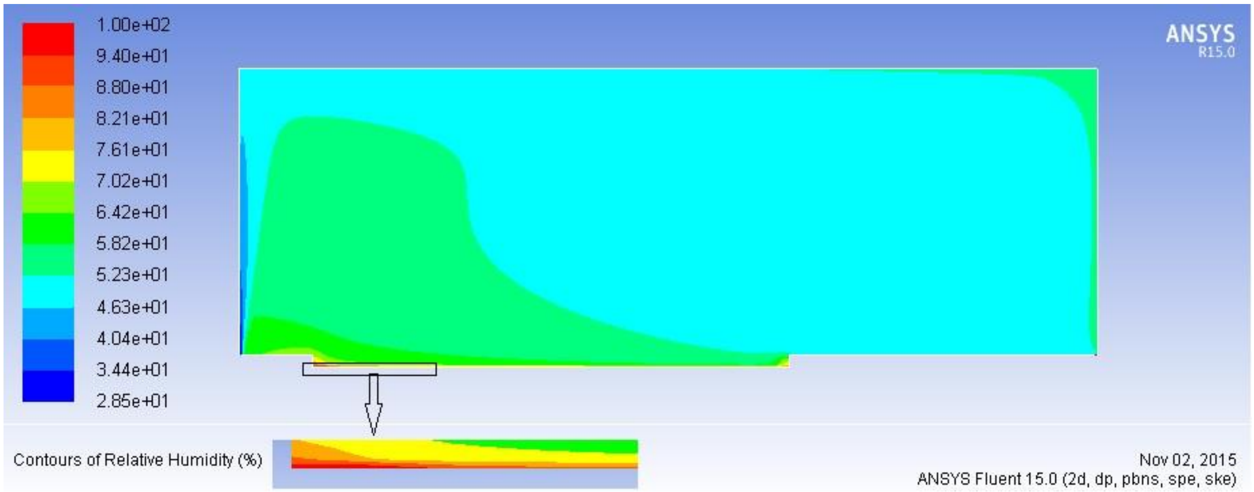

Figure 9 Contour of relative humidity of swimming pool test case

4.6 Porous medium approach

The porous medium is the material which is having hollow voids inside of it. Wet cooling media has material of such a kind which has pores. These micro pores trap water particles inside of it. If voids are saturated with water droplets than it is called wetting of the material. Figure 10 shows leaf which is solid without any voids and it forms water film or separate drops on the surface without letting it pass through it. Figure 11 shows cloth material having visible voids inside a material. This cloth soaks water droplets within this pores. Wet cooling media behaves same as figure 11 kind of surface.

Figure 10 Wetting of leaf

Figure 11 Wetting of cloth

The approach considered modelling micro pores in the material here is a porous medium approach. The porous medium has been mathematically derived first by Darcy in 1856 [7]. Basic volumetric flow through porous media can be derived from equation 10.

$$q = \frac{\kappa}{\mu} \nabla p \tag{10}$$

Where,

κ= permeability of the medium

μ= viscosity

The velocity of a porous medium is derived from volumetric flux calculated from equation 9 using equation 11.

$$v = \frac{q}{\emptyset} \tag{11}$$

$\emptyset$=porosity which is volume fraction of voids to total volume

4.7 Governing equation of porous medium

ANSYS Fluent uses basic Darcy's law for modelling porous media along with it uses inertial resistance. Porous media is considered as sink term in the momentum equation. In equation 5 additional sink term is enabled which acts on the whole volume of the cell. Equation 12 represents porous media sink term.

$$S_i = -(\sum_{j=1}^{3} D_{ij}\mu v_j + \sum_{j=1}^{3} C_{ij}\frac{1}{2}\rho|v|v_j) \tag{12}$$

Where,

D_{ij}= inverse permeability matrix

C_{ij}= inertial resistance matrix

i=momentum equation vectors

These parameters solely depend upon material properties and behavior in a fluid. For the unit surface of material these parameters can be calculated from pressure drop and porosity data.

4.8 Calculation of porous media parameters

Wet cooling media has cellulose, fiberglass, wool, wood and many more materials in use. Cellulose material is taken into consideration for this research work. Inertial resistance is calculated from equation 13 using existing pressure drop and superficial velocity data for the unit surface of a cellulose material. From this equation inertial resistance calculated for cellulose is 9000 m^{-1}.

$$C_2 = \frac{2\Delta p}{thickness\ v_{100\%open}^2} \tag{13}$$

Another factor is permeability which is calculated from equation 14 and for cellulose it is coming out to b 2.0 E +11.

$$\Delta p = \frac{\mu}{\alpha} thickness \tag{14}$$

4.9 2-D test case of porous media

The simple test case includes porous cell zone between fluid zones. The porous zone is kept dry without any water injection. The geometry of the test case is shown in figure 12. Air having relative humidity and water droplets are condensing due to pressure drop in the pores of the structure.

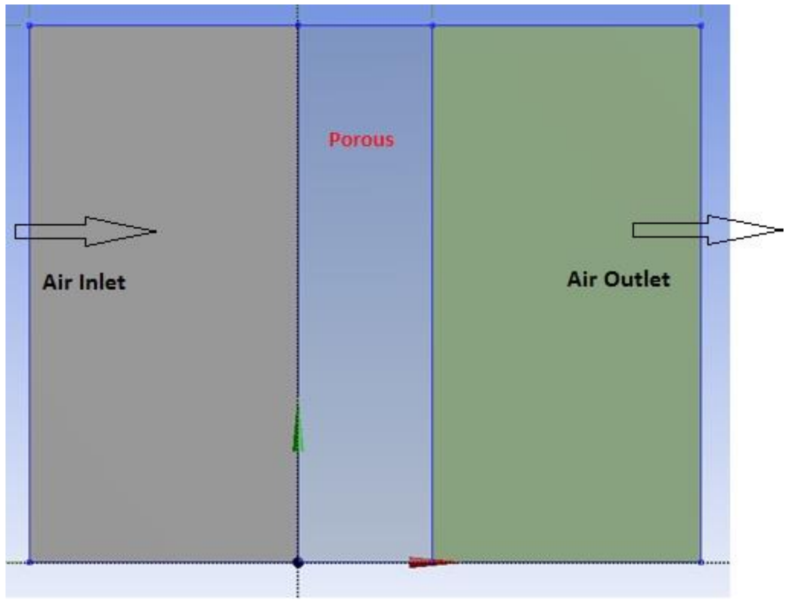

Figure 12 Porous media test case

Flow conditions for this test case is shown in table 3 below.

Table 3 Porous media test case flow conditions

Inlet	
Temperature	32° C
Relative Humidity	33.92 %
Velocity	$1 \frac{m}{s}$
Vapor mass source	0.0097 kg
Porous cell zone	
Permeability	2.0 E+11 m^{-2}
Inertial resistance	9000 m^{-1}

After steady state analysis, pressure contour of the test case is shown in figure 13. Pressure is dropping over porous cell zone. Pressure drop around 900 Pa is there in the porous zone.

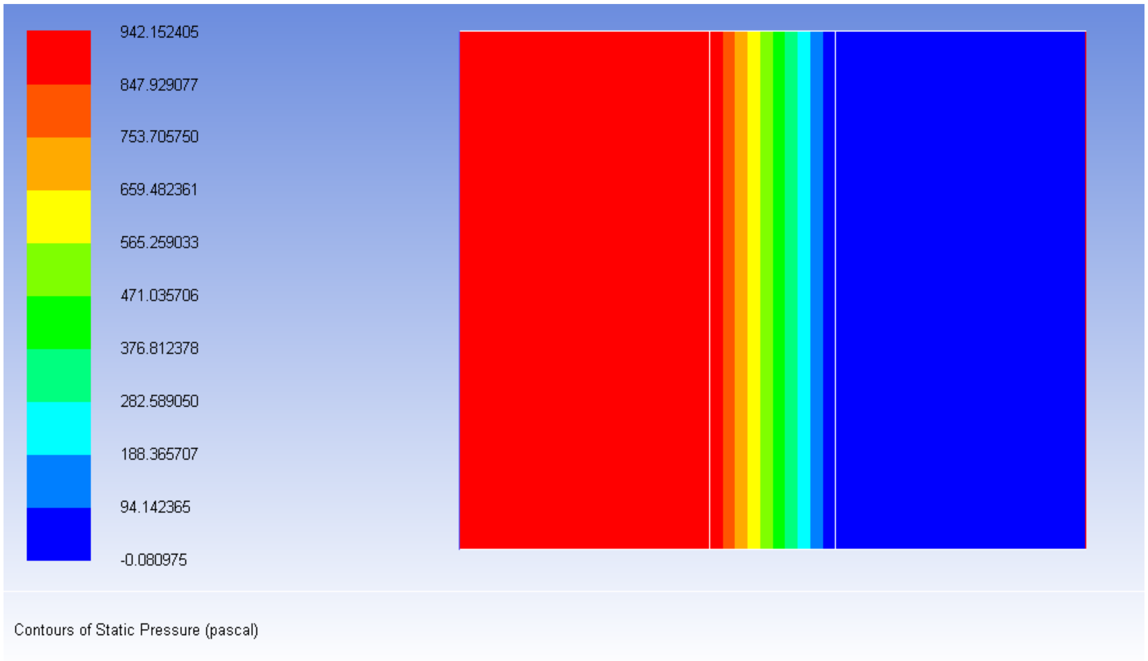

Figure 13 Pressure contour of porous test case

Figure 14 shows relative humidity contour of the test case. It is decreasing from 33.92% to 33.60%.

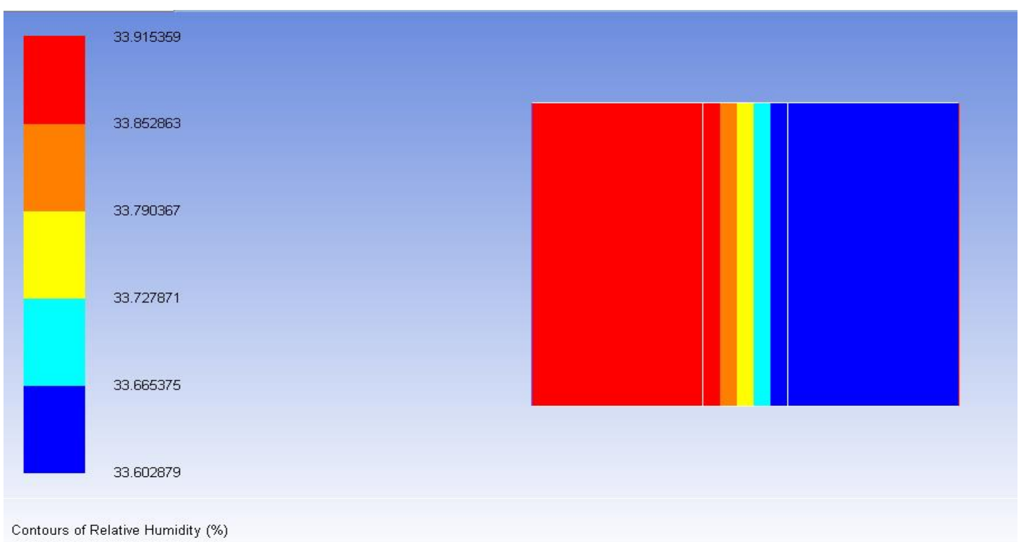

Figure 14 Relative Humidity contour of porous test case

With this modelling approach, this research work implements in wet cooling media CFD simulations.

CHAPTER 5 WET COOLING MEDIA: RESULTS AND VALIDATION

5.1 Flow domain and Conditions

Wet cooling media has been designed in PTC-Creo CAD software using solid surfaces which are shown in figure 15. These surfaces of media have flute angles which alternate as per vertical surfaces. One surface has flute angle of 45° negative in a downward direction. This angled surfaces guides the water droplets in downward direction. Another surface next to it has 15° upward angle. This surface guides air horizontal direction. Both comes into contact, but angles are put for flow guides.

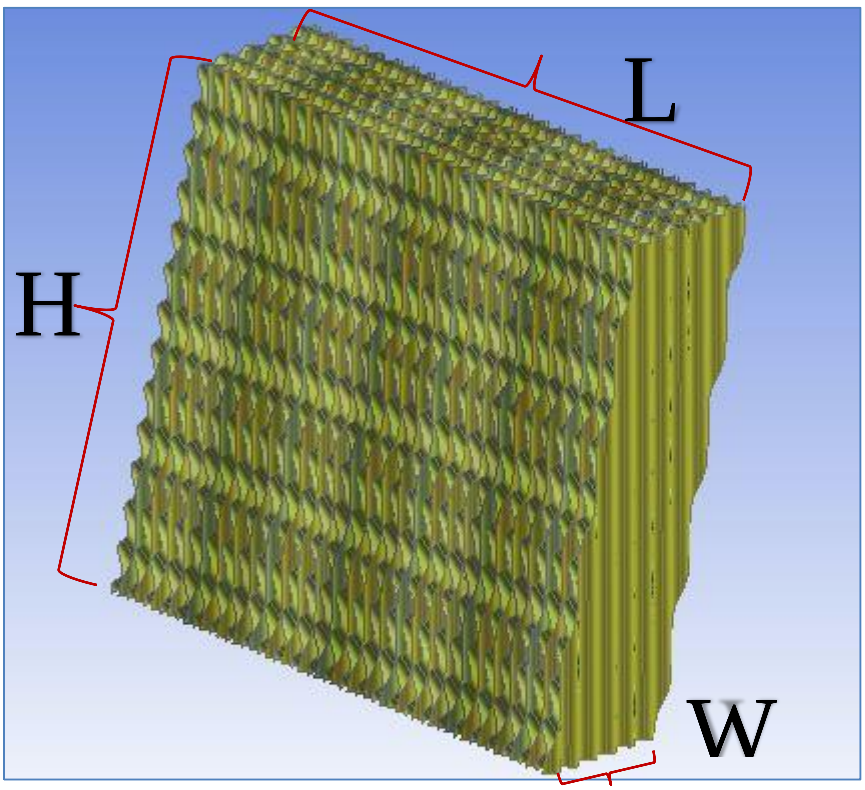

Figure 15 CAD model of wet cooling media

Geometry details of length, width and height are shown in table 4. Gap is maximum distance between alternate surfaces which is fixed for the model.

Table 4 Dimensions of wet cooling media

L	65 inch
W	12 inch
H	80 inch
Gap	9.8 mm

5.2 Flow domain and conditions

The rectangular domain is considered for flow simulation which is shown in figure 16. Enough provision is given at outlet for pressure recovery. This model is initially for dry simulation only without any water injection to validate the porous medium approach.

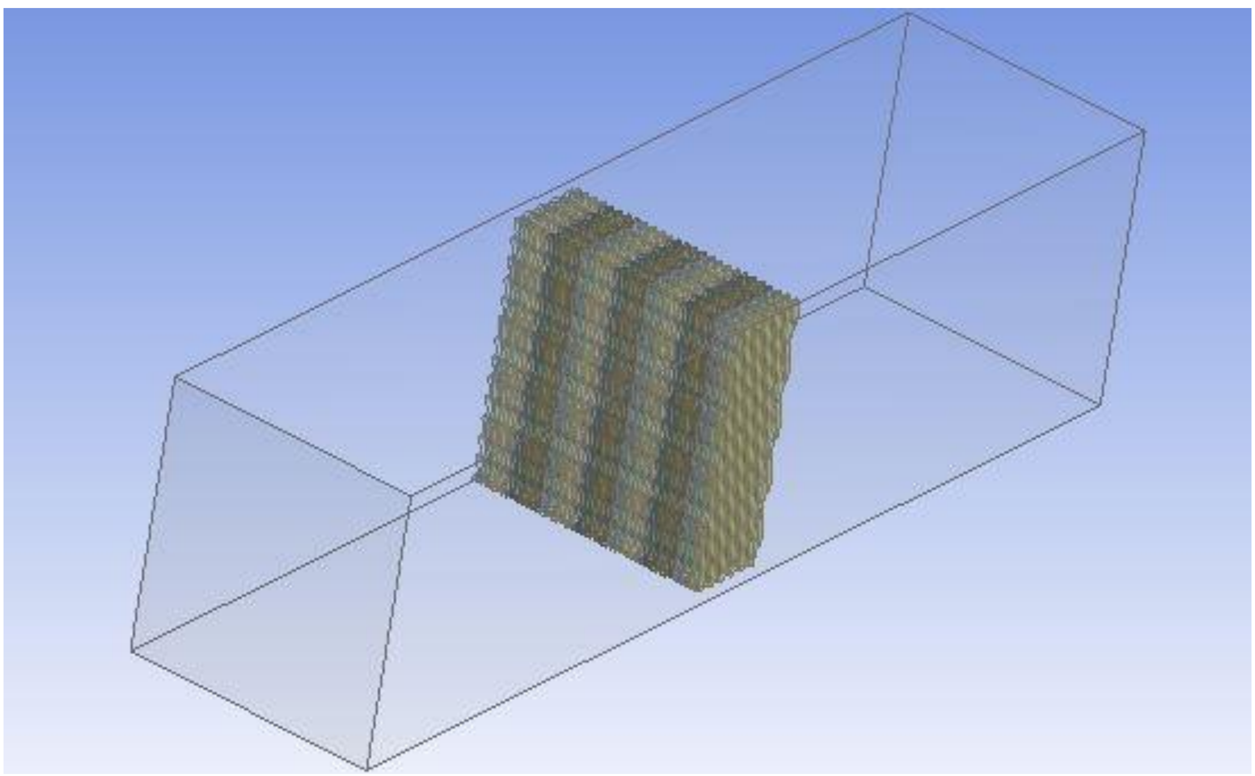

Figure 16 Flow domain of wet cooling media

Inlet flow and boundary conditions are shown in table 5. These conditions are kept same to validate with reference [8] results.

Table 5 Flow conditions for wet cooling media

Velocity	350 fpm
Reynolds Number	43000
Pressure	407 in of h_2o

The pressure drop of air over wet cooling media is shown in figure 17. The model has an upward deflection of air after the pad and turbulence is encountering in the domain. Spalart–Allmaras model are used to predict turbulence in the flow.

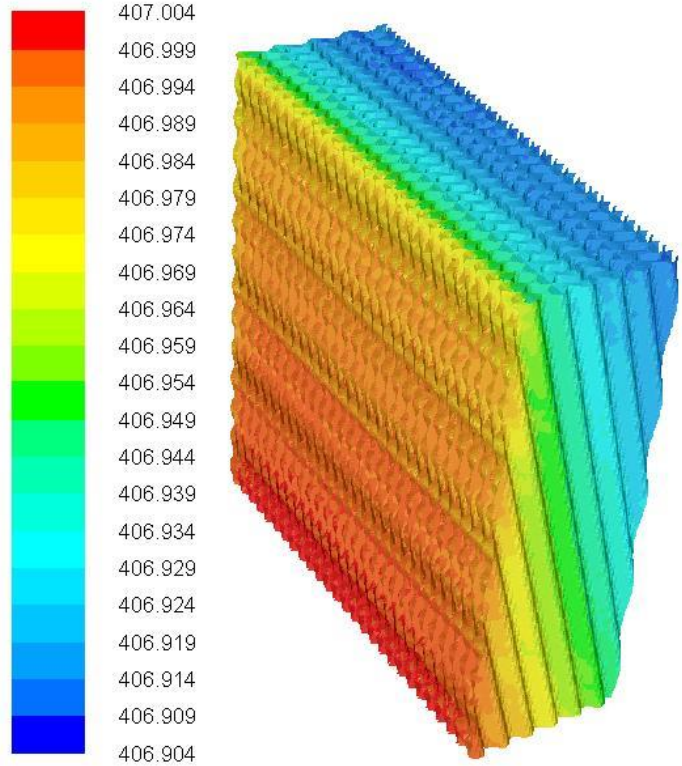

Figure 17 Contour of pressure drop across wet cooling pad (unit in ofh_2 o)

The pressure drop of 0.1 in of water is occurring over the pad at 350 fpm velocities. Figure 18 is a plot of pressure drop at various inlet velocities. Dotted line is experimental data which is compared with CFD results [8]. CFD results are in good agreement with experimental data. <u>So porous medium approach can be used for modelling wet cooling media.</u>

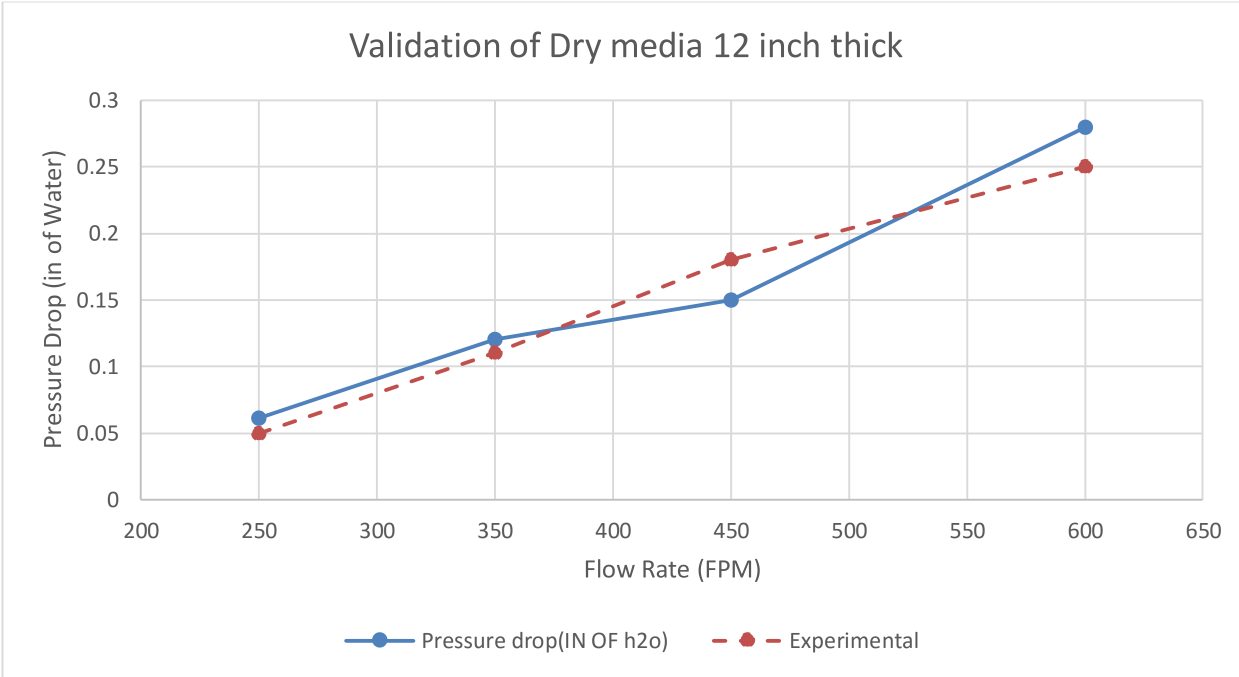

Figure 18 Validation of CFD pressure drop results with experimental data [8]

5.3 Water injection model

Water injection model has water inlet and an outlet at top and bottom of the pad respectively with the same top footprint of the pad surface which is shown in figure 19.

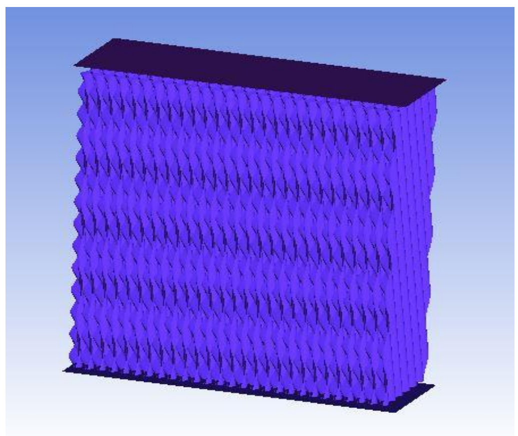

Figure 19 water injection model geometry

Water inlet conditions and inlet temperature conditions are shown in table 6.

Table 6 Water inlet condition for wet cooling pad

Air	
Temperature	305 K (85° F)
Relative Humidity	23%
Water	
Temperature	288K
Diameter of droplet	0.001 m
Mass flow rate	0.12 $\frac{kg}{s}$

After steady state analysis results obtained for the temperature at an outlet for air is 297 K (75° F). Figure 20 is experimental data of the temperature measured upstream and downstream of wet cooling media [8]. The average upstream temperature of the air is 85° F which is taken in CFD model. The downstream temperature of the air is 75° F average which is getting same in CFD model. Thus, model with water injection is validated and DPM can be used for future study.

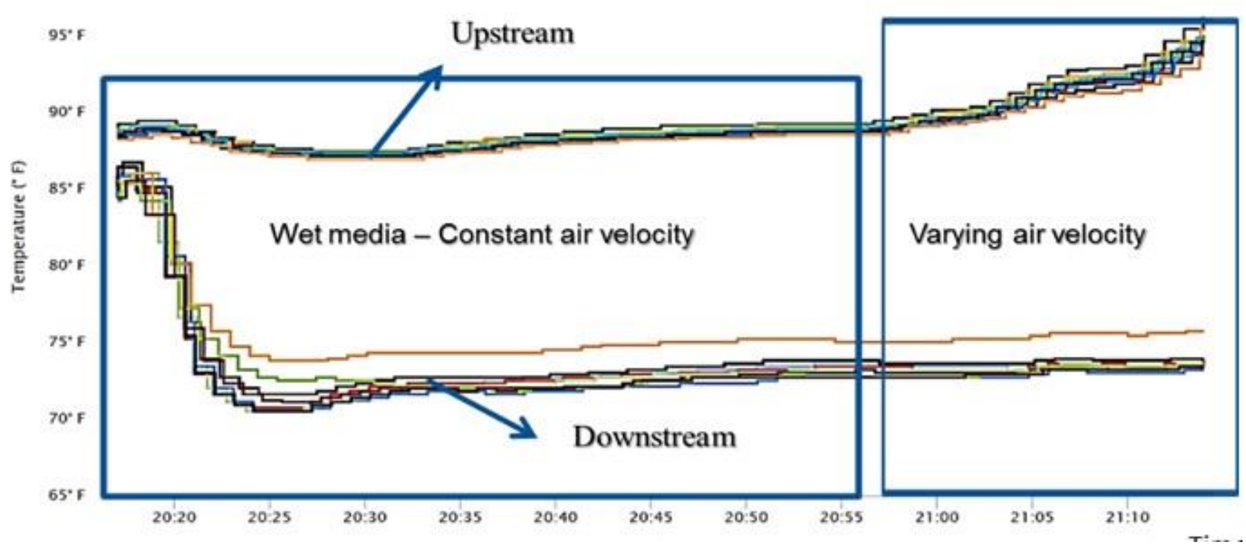

Figure 20 Temperature experimental data of wet cooling pad [8]

CHAPTER 6 CONCLUSION OF WET COOLING MEDIA AND FUTURE SCOPE

6.1 Conclusion

- ✓ CFD model is validated against experimental results for dry and wet both the cases.
- ✓ To model wet cooling media surfaces, a porous medium approach is successful and validated.
- ✓ DPM model is Lagrangian based model in which water particles can be tracked and water usage can be calculated.
- ✓ This model can be used free of dimensions and flute angles because porous parameters work for unit surface of the media material.

6.2 Future Scope

[1] Study of the flute angles and the different combination is the important factor in wet cooling media. In this research 45×15, combination is used. Another combination is 30×30 angle which can be studied for the effectiveness of the saturation.

[2] In this study, cellulose material is used and parameters of porosity had been calculated. Fiberglass, wood and plastic are also need to be studied. This part of research should calculate parameter for individual material and same modelling can be implemented with different porosity.

[3] Another part of research in wet cooling media is uneven staging of media. Staging means providing various flow rate of water to different sections of the media according to the weather condition. This methodology can save amount of water and the pumping power. CFD modelling is useful to determine optimum staging and water flow rate for that by running parametric simulations.

[4] Water flow rate is also crucial factor in direct evaporation cooling. This CFD model is based on constant water flow rate. Another way for water inlet variable flow rate according to outlet relative humidity condition. Change at water inlet conditions in CFD model can be used to determine which way is better.

[5] Evaporative cooling is effective when utilizing most out of fresh air. Weather condition of environment can be outside of ASHRE psychometric chart operational envelope [9].

Evaporative cooling from direct and indirect system can bring condition inside envelope adiabatically. This cooling path can be optimized by smart use of both systems. This kind of thermal behavior can be optimized by CFD analysis.

CHAPTER 7 INTRODUCTION TO OIL IMMERSION COOLING

7.1 What is oil immersion cooling?

Servers and data center traditionally uses air cooling system from a long time. Air cooling can be economical and easy to install technique in cooling technologies. Nowadays processor power usage is increasing and high power computing is coming in more demand. Traditional air cooling will become insufficient to provide cooling to high-end performance servers. Liquid cooling can be an efficient option for this type of server cooling. Liquid cooling involves direct and indirect cooling with use of either water or dielectric fluid. Indirect liquid cooling uses water as coolant fluid with the use of cold plates or back door heat exchanger in the data center can bring benefits in server cooling power consumption [11]. Cold plates have been in use from the late 1970s as a method of water cooling to high power servers using thermal conductive module [12]. But still cold plate is a hybrid cooling technique amongst server cooling technologies. Another component of server is cooled by air cooling while cold plate is used for processor and GPU cooling. Another disadvantage of cold plate is use of water which is electrically conductive and leakage may lead to short circuit and server failure.

To get rid of the hybrid system, direct immersion cooling can be a prominent solution for server cooling. Direct immersion cooling involves server to directly immerse in a bath of dielectric fluid. Dielectric fluid can have high carrying capacity than air and water. It can be thermally efficient and less power consuming for high-end packages. Even fluids with low boiling points can be retrofitted to two-phase cooling for higher heat removal. This research work uses white mineral oil for oil immersion cooling.

7.2 Why use of mineral oil?

Mineral oil is dielectric coolant oil which can be promising fluid amongst other immersion coolants. Selection of these fluids can become challenging study. Mineral oil has a history of usage in high power transformers [14]. It has 1150 times’ higher heat capacity than air. Mineral oil is available very cheap in a market than other cooling oils. It is less flammable and dense than other petrochemical oils.

7.3 Importance of heat sink in server cooling

Heat sink is passive heat exchangers that are being fitted on the processor. It carries heat away from the processor to the ambient fluid through fins or other kind of configuration. Heat sink heat carrying is crucial for server performance. Air cooling server is solely reliable on heat sink because they play a major role in keeping processer temperature within limit. Without heat sink processor heat dissipation in the ambient fluid becomes inefficient because of less contact surface area. Even in oil immersion cooling heat sink plays a vital role because heat conductance can be higher than convection.

Heat sinks are optimized for air cooling systems and they are available with various type of fins and having different configurations. Simplest type of heat sink is parallel plate heat sink which is show in figure 21.

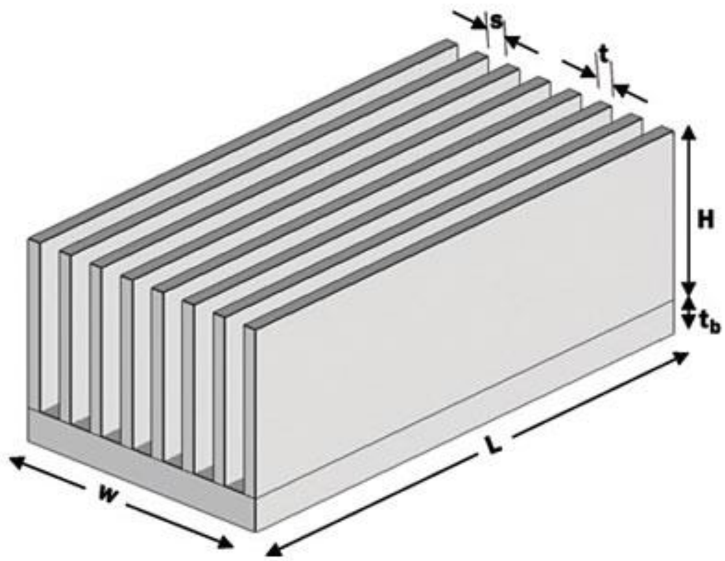

Figure 21 Parallel plate heat sink [15]

Heat sink design is a critical factor for air cooling and oil immersion cooling also. Let's take example of heat sink that if 44 mm tall heat sink can keep processor temperature within 65° C then it may be possible that 60 mm tall heat sink also keep temperature within 65° C. In this situation heat sink height after 44 mm case will become extra junk of material and it will not add value to thermal performance. This may also become true for a number of fins where extra fins may become resistance in the path of fluid after some optimum value. Determination of this optimum number of fin and height is the study of this research work. This study is the purpose to document the optimal design of parallel plate heat sink for oil immersion cooling.

The advantage of an optimized heat sink can be as follows.

- Reduced heat sink height can allow more servers per rack
- Material cost saving
- Installing optimal heat sink can increase server efficiency

8.1 Literature Survey

Heat sink characterization is an important factor in determining product development. Use of CFD to optimize the heat sink is becoming the useful technique. Heat sink aspect ratio which is fin height/ fin gap is approaching 10 and greater [16]. These large aspect ratio heat sinks need to overcome flow impedance. Channel velocity should sufficient to fulfill target thermal resistance of the design. Thus, heat sink design can become a crucial factor. Heat sinks are needed to be characterized in wind tunnel testing. CFD technique in older times was very expensive if all the details are needed to be modelled. Time consumption can increase with a number of grids and component modelling. One way to do heat sink CFD model is to put volumetric resistance instead direct model [16]. These type of volume resistance requires pressure drop and thermal resistance data at different velocity [16]. Volumetric resistance is sufficient to model heat sink. This technique is useful to optimize junction temperature for different heat sinks than full geometry modelling. Heat sink cost analysis is shown in figure 22 for various heat sink configurations.

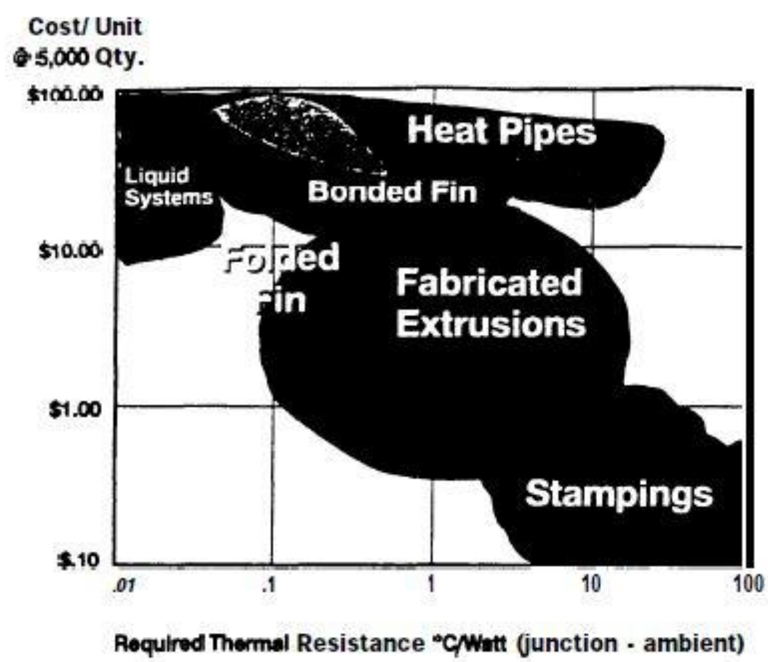

Figure 22 Heat sink cost analysis [17]

The cost of a heat sink is increased when low thermal resistance becomes in demand. Typically oil cooling system itself provides higher cooling and keeps thermal resistance around $0.1\frac{°C}{W}$. With this reason parallel plate heat sink which comes in bonded category will a serve better purpose. This factors should be considered in optimization of heat sink design [17].

- Fin height
- Fin length
- Fin thickness
- Fin shape
- Density of fins
- Cross cut patterns
- Material

8.2 Project Methodology

Considering literature survey this research seeks optimal heat sink design for application of oil immersion cooling using computational fluid dynamics approach. Facebook open compute generation 1 server is utilized for this study [10]. This research involves developing CFD model and validate it with previous experimental results [10]. Parallel plate heat sink is already in use for the server is one of the many reasons to take into consideration for optimization. The project has opted following parameters to consider for optimization.

- Fin height
- Number of fins
- Server orientation

CFD optimization involves two methods for optimization. The first method is fixed volume flow rate in the server at different parametric runs. Another method is fixed pumping power and comparison of both the methods has been presented.

CHAPTER 9 OPEN COMPUTE SERVER: CFD MODELLING AND VALIDATION

9.1 Geometrical modelling

Open compute server has three fans for air cooling system, but it has not been considered for this research work. Two CPU are taken into consideration for analysis and they have embedded power of 95 W. There are two parallel plate heat sinks being fitted on top of both CPUs. Additional components are taken into consideration are RAM units and capacitance units to guide the flow in the least resistance path. Geometry is being prepared in ANSYS Icepak which is shown in figure 23.

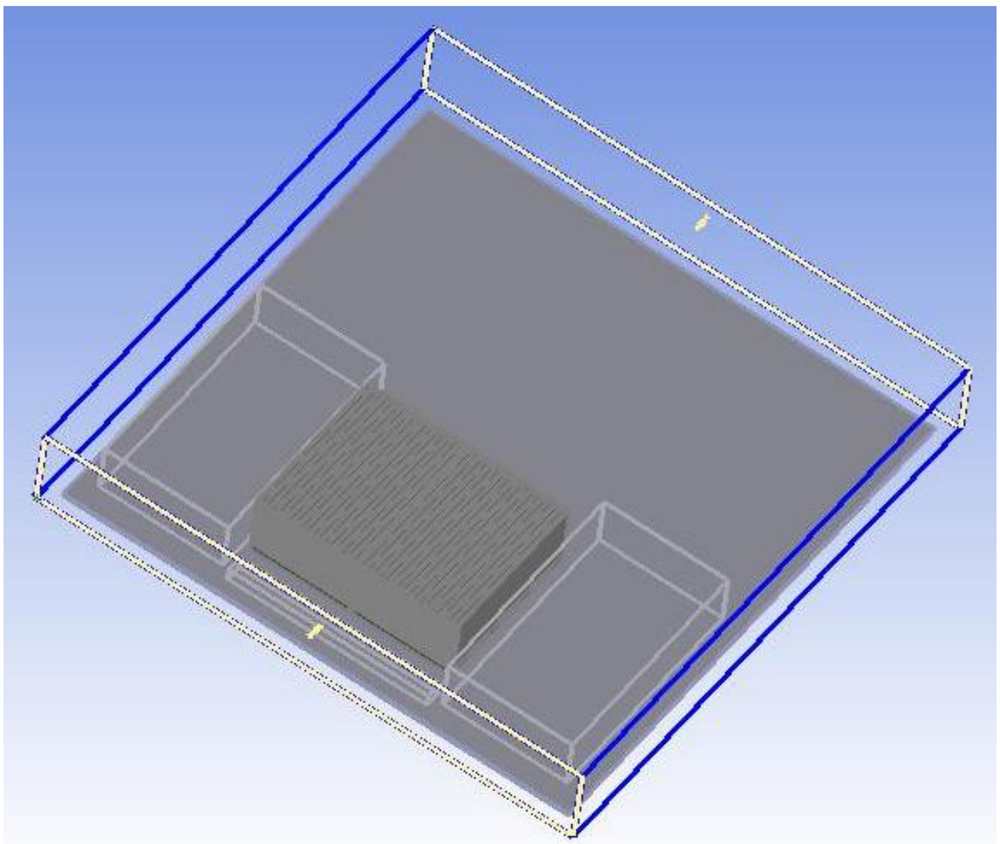

Figure 23 Open compute server geometry

All the dimensions of the model are shown in table 7. Heat sink material is taken as Aluminum extruded. For PCB, RAM and capacitors units solid material are FR-4 and surface material is copper.

Table 7 Dimensions of open compute server CAD model

Heat sink (All dimensions are in cm)	
Foot print	10×7
Base Height	0.3
Total height	4.4
Fin thickness	0.07
Number of fins	18
RAM Unit	
Foot print	20×5
Height	3.5
Capacitor Unit	
Foot print	3×11

9.2 Mineral oil properties

Mineral oil has a viscosity which is dependent upon temperature. In ANSYS Icepak this dynamic nature of viscosity has been taken into account. Other material properties are shown in table 8 [18].

Table 8 Mineral oil properties

Density	$849.3\,\frac{kg}{m^3}$
Specific heat	$1680\,\frac{J}{kg\,K}$
Thermal Conductivity	$0.13\,\frac{W}{m\,K}$
Dynamic viscosity	$C1 \times e^{\left(\frac{2797.3}{T+273.2}\right)}$ where T = fluid temperature

9.3 Flow conditions

Flow conditions are shown in table 9. Baseline case of flow rate is taken as 1 lpm which will be explained later in this chapter. This case is baseline case means heat sink size is 18 fins and 4.4 cm height.

Table 9 Flow conditions

Temperature	30° C
Volume flow rate	1 lpm
Velocity	0.00115 $\frac{m}{s}$
Reynolds number	3.6
Pressure	6 psi

After steady state analysis, temperature distribution inside the server is shown in figure 24. Maximum junction temperature is 339 K which is spreading over PCB in the wake region.

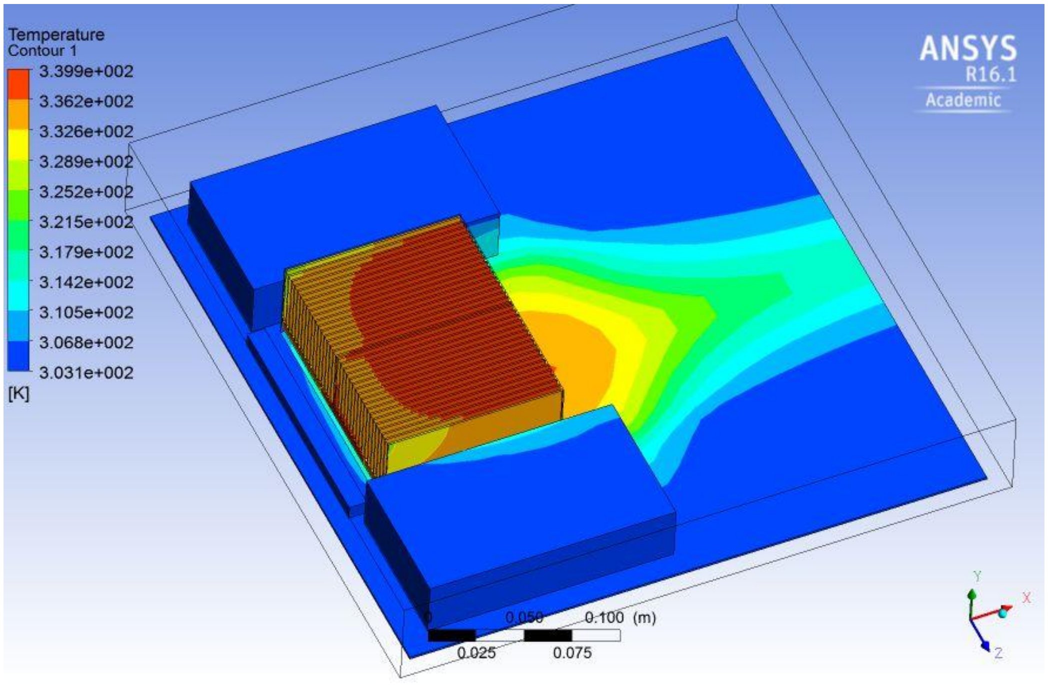

Figure 24 Temperature distribution in the cabinet

9.4 Validation of CFD model

CFD results are validated against previous experimental publishing [10]. Validation is a graph of junction temperature at different volume flow rate which is shown in figure 25. Experimental data is taken at inlet temperatures of 30° C to 45° C. For this research work, two inlet temperature of 30 °C and 35° C have been taken into consideration. For these two cases, CFD results are coming in good agreement with experimental data which validates the model methodology. This model has been further used for optimization work. Exp in legend represents experimental data [10]. CFD in legend represents CFD data.

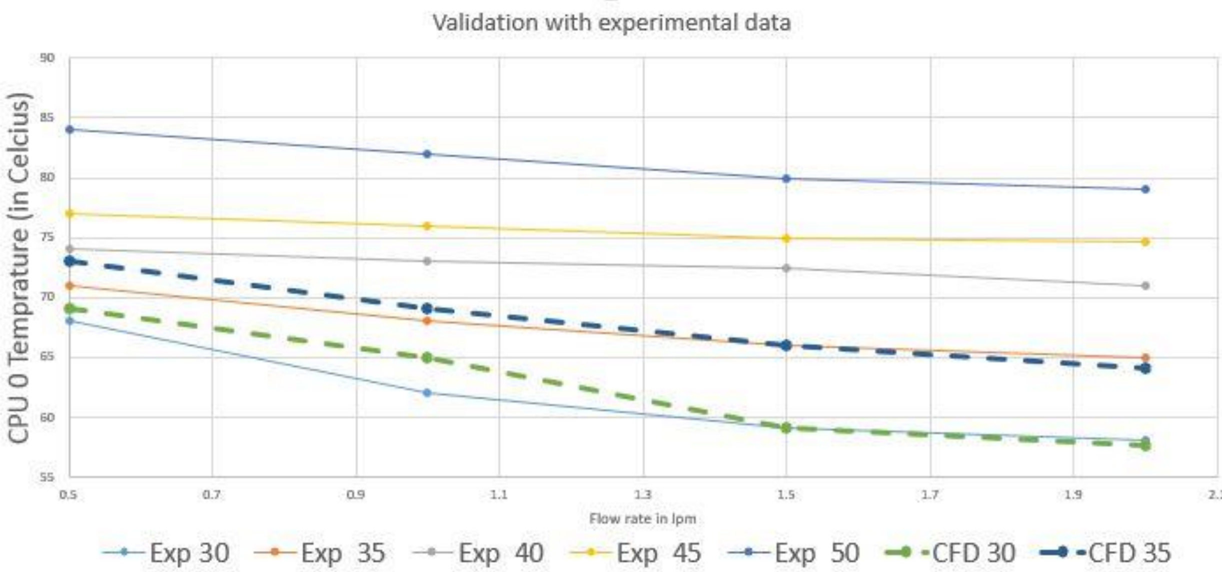

Figure 25 Validation plot of Temperature with experimental data [10]

Grid independence study has also been carried out for baseline case of heat sink design and temperature of 20° C and 1 lpm volume flow rate. 1 million cell is threshold value in this research work. Up to 1 million cells, junction temperature reduces significantly which is shown in figure 26. After this limit temperature change is within an agreement. For future study of this research work 1 million cells have been used for meshing.

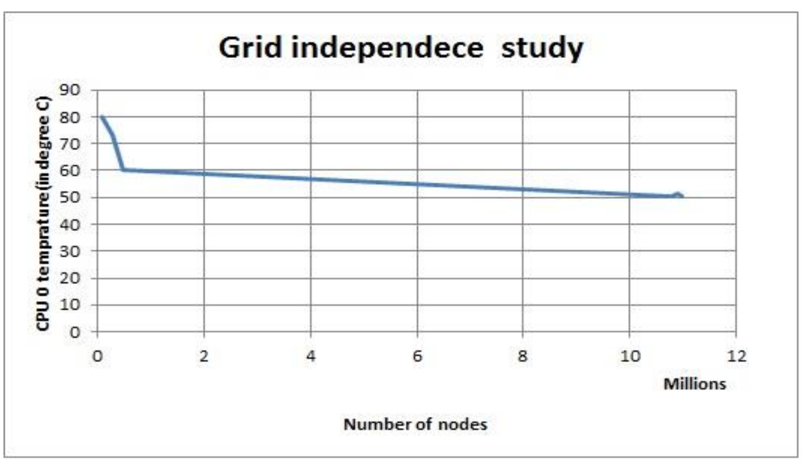

Figure 26 Grid independence study

Figure 27 shows thermal resistance at different volume flow rates. This is a case of 30° C for baseline heat sink design. Here thermal resistance does not change significantly after 1 lpm volume flow rate and it is considered baseline case for that reason.

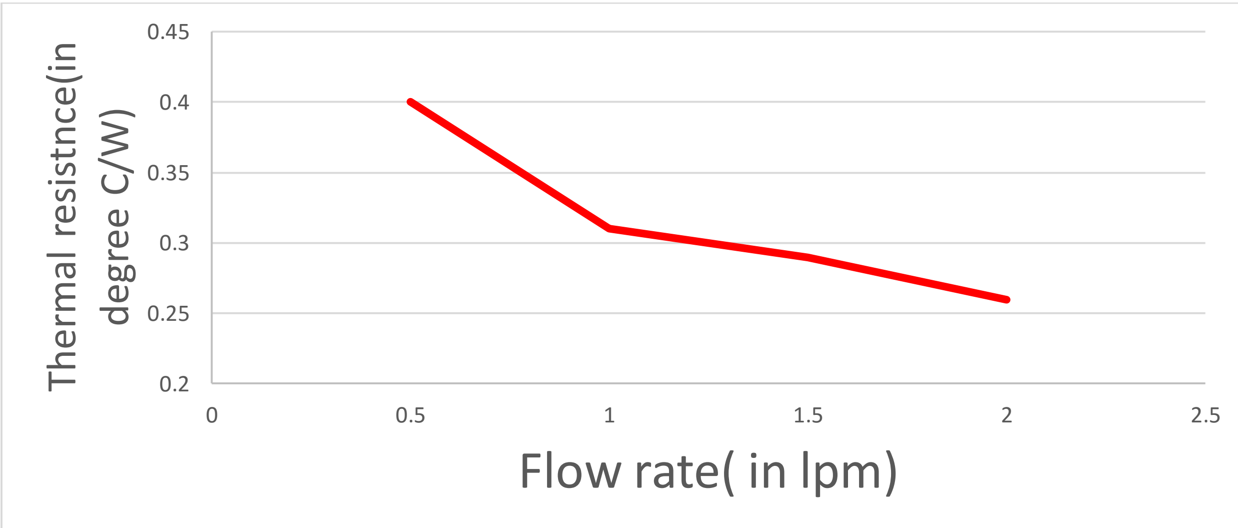

Figure 27 TR at various flow rate

CHAPTER 10 OPTIMIZATION RESULTS

10.1 Optimization by fixed volume flow rate

Volume Flow rate is kept fixed at 1 lpm in this parametric optimization. This case considers the impact of a change in inlet area and change in velocity accordingly. The result obtained is shown in figure 28 which is a surface plot of Fin height and number fin vs. thermal resistance. Fin height is taken from 0.5 cm to 4.4 cm. The number of fins are taken from 2 to 40.

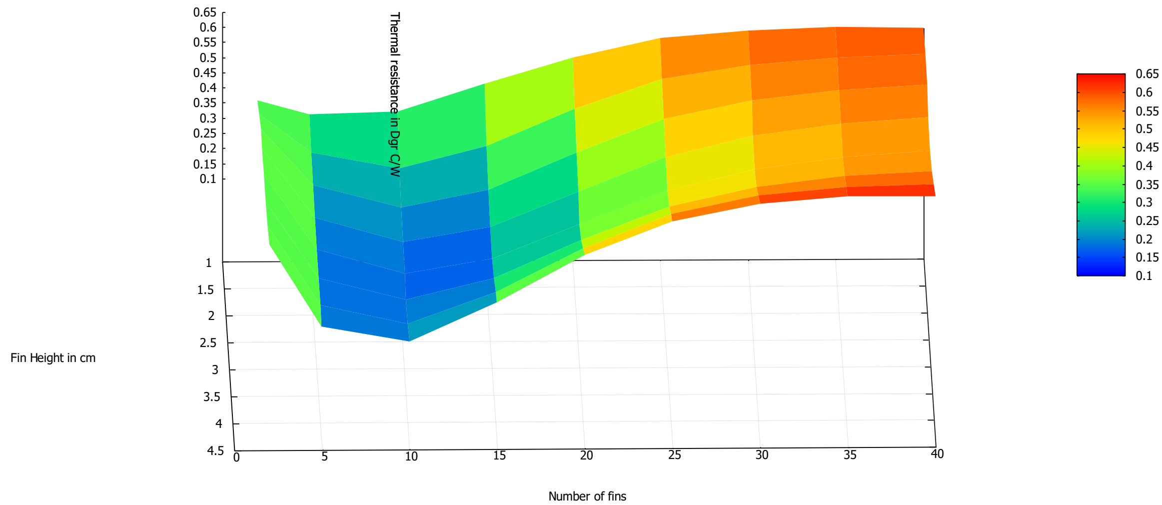

Figure 28 Parametric optimization at fixed volume flow rate

Blue color region in surface plot shows minimum value of thermal resistance and it is coming out around $0.1\frac{°C}{W}$. Up to 8 number of fins TR is decreasing to minimum and after 2 cm of fin height change of TR is not significant.

10.2 What is fixed pumping power optimization?

Pumping power is the multiplication of volume flow rate and pressure drop across the channel. For the base line case pumping power is calculated as 1.8 W. Now if heat sink height is increased than TR will increase but it may give significant pressure drop. Pressure recovery will consume more power than before and thermally optimal design at fixed volume flow rate will turn out to be more costly. Parametric optimization at fixed pumping power will ensure pressure drop and it will set volume flow rate accordingly for a new design. If pressure drop is significantly high than volume flow will reduce to keep pumping power same. TR will also decrease with volume flow rate and balanced design will come out in this study.

10.3 Optimization by Fixed pumping power

Figure 29 shows results from parametric optimization at fixed pumping power of 1.8 W. This is same surface plot as explained in section 10.1. Optimum number of fins are coming out to be 8 to 15. Optimal range of fin height is 3 cm to 4 cm.

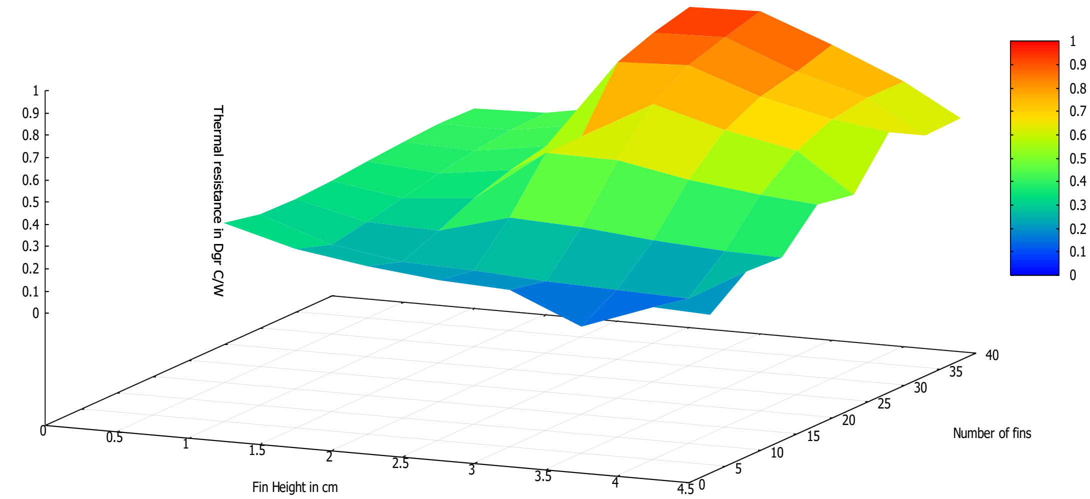

Figure 29 Parametric optimization at fixed pumping power

CHAPTER 11 HEAT SINK OPTIMIZATION: CONCLUSION AND FUTURE WORK

11.1 Conclusion

- ✓ CFD model has been validated with experimental data.
- ✓ Optimum number of fin are ranging from 8 to 15.
- ✓ Optimal fin height range is 3 cm to 4 cm.
- ✓ Cost of heat sink can be saved from 20% to 44.19% if optimized design is implemented.

11.2 Future work

- Plate fin and pin fin heat sinks can be studied and optimized
- Ducting of flow towards heat sink can lead to higher thermal performance
- Experimental validation of optimized results is also a good piece of work

REFERENCES

[1] R. A. Bucklin, J. D. Leary, D. B. McConnell, and E. G. Wilkerson," Fan and Pad Greenhouse Evaporative Cooling Systems" CIR1135.

[2] Sapounas, C. Nikita, T. Bartzanas, C. "Aspects of CFD modeling of fan and pad evaporative cooling system in greenhouses" 2nd PALENC Conference and 28th AIVC Conference on Building Low Energy Cooling and Advanced Ventilation Technologies in the 21st Century, September 2007, Crete island, Greece.

[3] S. Riffat, A. Oliviera, J. Facão, G Gan, P. Doherty "Thermal performance of a closed wet cooling tower for chilled ceilings: measurement and CFD simulation". INTERNATIONAL JOURNAL OF ENERGY RESEARCH Int. J. Energy Res. 2000; 24:1171}1179.

[4] Jairo Alexander Osorio Saraz, Ilda de Fátima Ferreira Tinoco, Keller Sullivan Oliveira Rocha, Marcio Arêdes Martins, Flávio Alves Damasceno. "Validation of a CFD model for prediction of the efficiency of evaporative cooling in porous panels". rev.udcaactual.divulg.cient vol.15 no.1 Bogotá Jan./June 2012.

[5] Shah, M. M. Rate of evaporation from undisturbed water pools to swimmbad Electrowaerme Internationaal 32 (A3), A115-A129, 2002.

[6] Li, Z., & Heiselberg, P. K. (2005). CFD Simulations for Water Evaporation and Airflow Movement in Swimming Baths. Instituttet for Bygningsteknik: Aalborg Universitet.

[7] Darcy, H. (1856). Les Fontaines Publiques de la Ville de Dijon, Dalmont, Paris.

[8] Sreeram, V, Gebrehiwot, B, Sathyanarayan, S, Sawant, D, Agonafer, D, Kannan, N, Hoverson, J & Kaler, M 2015, 'Factors that affect the performance characteristics of wet cooling pads for data center applications'. in Annual IEEE Semiconductor Thermal Measurement and Management Symposium. vol. 2015-April, 7100160, Institute of Electrical and Electronics Engineers Inc., pp. 195-202, 31st Annual Semiconductor Thermal Measurement and Management Symposium, SEMI-THERM 2015, San Jose, United States, 15-19 March., 10.1109/SEMI-THERM.2015.7100160.

[9] ASHRE TC9.9,” 2011 Thermal Guidelines for Data Processing Environments – Expanded Data Center Classes and Usage Guidance”, American Society for Heating, Refrigeration and Air-Conditioning Engineers, 2008.

[10] Richard Eiland ; John Fernandes ; Marianna Vallejo ; Dereje Agonafer ; Veerendra Mulay,” Flow Rate and inlet temperature considerations for direct immersion of a single server in mineral oil”. 14th InterSociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems, ITherm 2014.

[11] Almoli, A. Thompson, N. Kapur, 1. Summers, H. Thompson and G. Hannah, "Computational Fluid Dynamic Investigation of Liquid Cooling in Data Centers, " Applied Energy, vol. 89, pp. 150-155, 2012.

[12] R. C. Chu, U. P. Hwang and R. E. Simons, "Conduction Cooling for an LSI Package: A One-Dimensional Approach, " IBM Journal of Research and Development, vol. 26, no. 1, pp. 45-54, 1982.

[13] Fernandes, S. Ghalambor, A. Docca, C. Aldham, D. Agonafer, E. Chenelly, B. Chan and M. Ellsworth Jr., "Combining Computational Fluid Dynamics (CFD) and Flow Network Modeling (FNM) Design of a Multi-Chip Module (MCM) Cold Plate, " in ASME InterPACK Conference, San Diego, CA, 2013.

[14] L. W. Pierce, "An Investigation of the Thermal Performance of an Oil Filled Transformer Winding, " IEEE Transactions on Power Delivery, vol. 7, no. 3, pp. 1347-1358, 1992.

[15] Younes Shabany,” Simplified Correlations For Radiation Heat Transfer Rate In Plate-Fin Heat Sinks,” Journal of electronic cooling, August 2008.

[16] C.D.Pate1, "Backside Cooling Solution for High Power Flip Chip Multi-chip Modules," 44th IEEE Proceedings, May1 994.

[17] Seri Lee, “Optimum design and selection of heat sinks,” IEEE, 1998.

[18] “STE Oil Company data sheets and MSDS”. [Online]. Available at: http://www.steoil.com/msds-tech-data.

[19] “Water Cup evaporation’. [Online]. Available at: https://commons.wikimedia.org/w/index.php?curid=425897

[20] "Adiabatic process". [Online]. Available at: https://commons.wikimedia.org/w/index.php?curid=1721940.

[21] "Qanat". [Online]. Available at: https://commons.wikimedia.org/w/index.php?curid=8659258.

[22] "SanHe". [Online]. Available at: http://www.diytrade.com/china/pd/8323334/7090Evaporative_wet_cooling_pad.html.

[23] HVAC Systems and Equipment (SI ed.). Atlanta, GA: American Society of Heating Refrigeration and Air-conditioning Engineers (ASHRAE). 2012. p. 41.1.

[24] Allen, R.G.; Pereira, L.S.; Raes, D.; Smith, M. (1998). Crop Evapotranspiration—Guidelines for Computing Crop Water Requirements. FAO Irrigation and drainage paper 56. Rome, Italy: Food and Agriculture Organization of the United Nations. ISBN 92-5-104219-5. Retrieved 2007-10-08.

[25] ANSYS Fluent theory guide. Release 15.0, November 2013. ISO 9001:2008.

[26] Gibbs, J.W., Scientific Papers (Dover, New York, 1961).

BIOGRAPHICAL INFORMATION

Shivang Desai was born in Ahmedabad, India in October 1992. He has pursued his Bachelor's degree (BE) in Aeronautical Engineering from Gujarat Technological University, India in 2010. He completed his Master of Science in Mechanical Engineering from the University of Texas at Arlington.

He has a great interest in computational fluid dynamics area. He has conducted variety of projects in this area from undergraduate studies from aerodynamics drag reduction to boundary layer flows. His education in Aeronautics provided him the resources of fluid and structural simulations. He earned "Most innovative student of the year" award for this contribution.

He joined EMNSPC team in spring 2014. He has shown a great interest in oil immersion cooling as well as air cooling systems. He has been working on Evaporative cooling for data center application for Mestex Company. This work includes "CFD modelling of wet cooling media for direct evaporative heat exchanger" which is less explored work in the industry. This work promises to open many optimization studies to help improve the unit and installation work. On the other side, he worked on upcoming cooling technology for high performance computing which oil immersion server cooling. This project includes "Parallel plate heat sink optimization for oil immersion cooling" which gave the promise of 44% cost reduction with performance enhancement of heat sink if industry wants to retrofit the server technology for oil immersion cooling.

He has been honored with membership of Tau Beta Pi national engineering honor society in 2014. He is also active member of ASME and ASHRAE.