COMPARATIVE RACK LEVEL CFD ANALYSIS OF AIR TO HYBRID COOLING DATA CENTER

BY

JAY UMESHKUMAR PAREKH

THESIS

Submitted in Partial Fulfillment of Requirements

for Degree of Master of Science in Mechanical Engineering at

THE UNIVERSITY OF TEXAS AT ARLINGTON AUGUST,2019

Arlington, Texas

[Type here]

ACKNOWLEDGMENT

I would like to thank **Dr. Dereje Agonafer** for giving me an opportunity to work on this project. His continuous guidance throughout my thesis under him at The University of Texas at Arlington has helped me come around all problems that I faced.

I would like to thank **Dr. Abdolhossein Haji Sheikh** and **Dr. Andrey Bayle** for evaluating my work as committee members. I would like to thank **Uschas Chowdhury** for being patient and supporting me by clearing all doubts that I had throughout my stay at UT Arlington. I would like to thank all **EMNSPC team members** and **Friends** for supporting me.

Finally, I would like to thank god and my family members for standing by my side and for believing in me in every aspect. I would thank the Almighty God for giving me strength and inspiration.

ABSTRACT

COMPARATIVE RACK LEVEL CFD ANALYSIS OF AIR TO HYBRID COOLING OF A DATA CENTER

Jay Umeshkumar Parekh, MS

The University of Texas at Arlington, 2019

The demand of data storage in the world is increasing, with this increase, demand for large data centers is growing. In response, we need to look for energy-efficient solutions to reduce the energy consumptions of the data centers. We want to reduce the energy used for cooling while keeping the data center equipment temperature in allowable range in high power density data center. The liquid or hybrid cooled servers are the alternatives that may be used instead of traditional air-cooled servers at Rack Level for the cooling power - different inlet temperatures and different percentage of hybrid cooling. This will help us in considering options for a different type of application with efficient energy solution to achieve global need for saving energy. In this work, the server used is CISCO 220 M3 server 1U small form factor server with 2 CPU chips, 5 hot-swappable fans and 16 DIMMS slots. The air-cooled server has 2 CPU heat sinks to dissipate the total heat of the server. The hybrid cooled server consists of 2 cold plates with integrated pumps to cool the CPUs. In this study, a data center with 6 Racks and a capacity of 42 1U servers is considered along with 2 CRAH or CRAC unit, hot and cold aisle containment. Using the characterizations of air-cooled server component temperatures and flow rate at different inlet temperature in the room model, a comparison of power used by the CRAH or CRAC unit is shown. An efficient solution to reduce the cooling power for different inlet temperatures is proposed.

Table of Contents

Table of Figures

NOMENCLATURE

ρ	Density (kg/m3)
k	Thermal Conductivity (W/m-K)
ʋ	Velocity (m/s)
μ	Viscosity (N/m2S)
ε	Kinematic Rate of Dissipation (m2/s3)
ṁ	Mass Flow Rate (kg/sec)
Q	Heat Load (KW)
P	Power (W)
ϑ	Volumetric Flow Rate (cfm)
p	Pressure (Pa)
T	Temperature (K)
Cp	Specific Heat Capacity (J/kg-k)
Re	Reynolds Number
l	Characteristic Length

Chapter 1:- Introduction

The modern world has evolved from a period where physical copies where used to refer to information, wired phones and human brains were used for processing. Large storage areas were designated for storage of information in physical copies which was also used for backup. While in the modern world, things have digitized. The information now is available digitally for ease of access. This information needs to be stored, processed and shared, here comes the role of a modern-day data center. The data centers store information and help in processing it. Data centers accommodate many IT equipment such as servers, computer systems, data storage units and telecommunication devices. The U.S. Environment Protection Agency defines a data center as, "Primarily electronic equipment used for data processing (servers), data storage (storage equipment), and communications (network equipment). Collectively, these equipment processes, stores, and transmits digital information." "Specialized power conversion and backup equipment to maintain reliable, high-quality power, as well as environmental control equipment to maintain the proper temperature and humidity for the ICT equipment." [1].

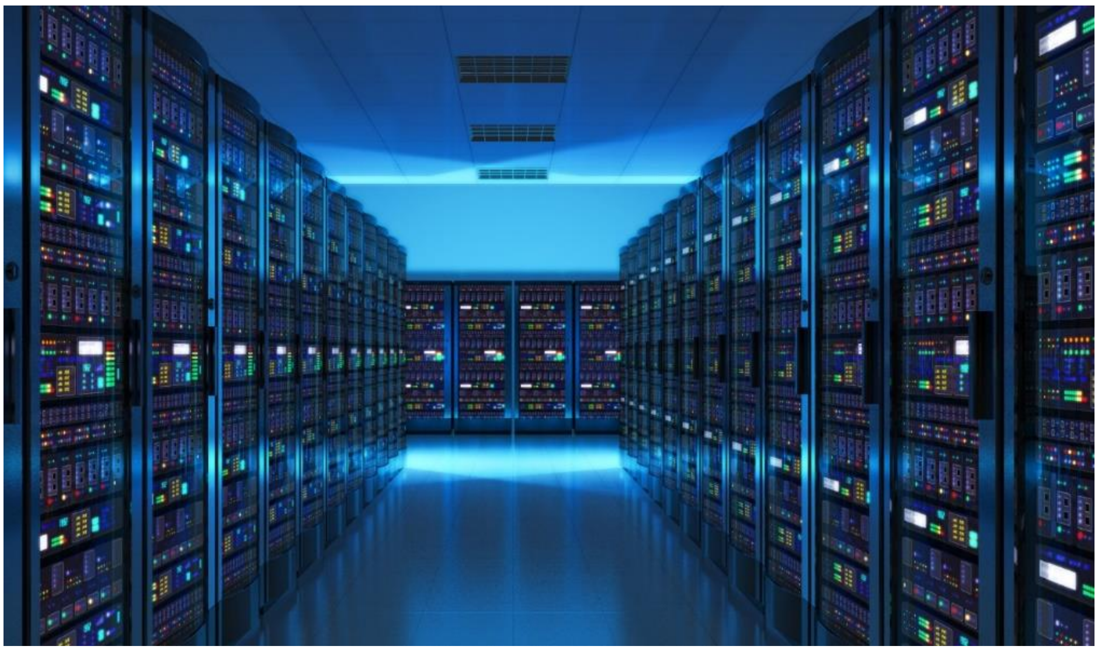

Figure 1:- Microsoft Future Data Center[2]

1.1 Power Utilization of Data Center in Industries

In the modern world, the demand for data storage, processing and sharing is burgeoning with a swift pace. With this growing demand, the demand for building bigger and new data center is increasing. Thus, the data center continues to advance as a significant consumer of electricity among different type industries. A data of total U.S. data center electricity use (servers, storage, network equipment, and infrastructure) from 2000-2020 is estimated using data available is shown in the figure below. In 2014, data centers in the U.S. consumed an estimated 70 billion kWh, representing about 1.8% of total U.S. electricity consumption. Based on current trend estimates, U.S. data centers are projected to consume approximately 73 billion kWh in 2020 [3].

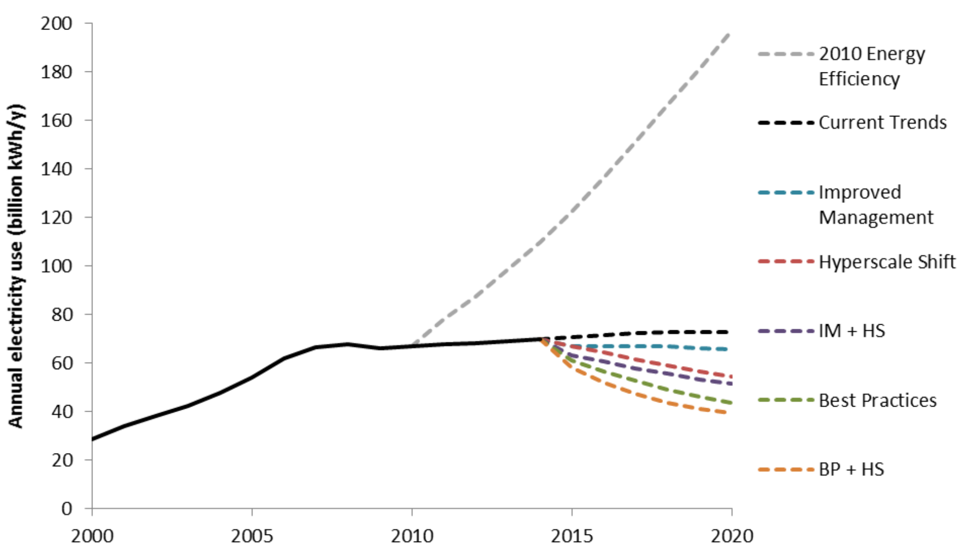

Figure 2:- Projected Data Center Total Electricity Use[3].

An estimate has been generated that comprises of servers, storage, network equipment, and infrastructure in all U.S. data centers. The solid line in the above graph depicts historical estimates from 2000-2014 and the dashed lines show five projection scenarios through 2020. Increasing usage of electricity leads to a concerned climatic change and aggravated carbon footprint. Environmental agencies around the globe and concerned about the increased energy usage by the data center. This electricity used at the data center can be substantially reduced by implementing additional energy-efficient strategies and techniques.

1.2 Energy and Thermal Management of Data Centers.

Data centers are a composition of information and communication technology (ICT) systems that run continuously and have high costs. Thermal management and energy efficiency are factors that help in maintaining a safe working environment, reliability and up-time of data center equipment. ICT system server the end users while the cooling systems like server fans and computer room air conditioners (CRACs) serve the ICT system. The temperature of the ICT systems is controlled by the cooling systems which in return maintains the quality of service. To reduce the energy consumed in data centers, energy efficient management of both ICT and cooling systems is done to satisfy the thermal constraints of ICT systems [4].

1.3 Energy Efficiency

The enormous use of Internet of Things, cloud-based services, record database and technological advancement calls for advanced IT equipment. The IT equipment consumes a lot of energy in the form of electricity and generates heat. In a data center reliability of components is crucial, to improve the reliability of component the heat generated must be reduced. IT equipment and cooling consume major energy of a data center. The share of cooling power in a typical data center is estimated at about 30-50%. So, efficient utilization of energy by cooling reduces the total power consumed by data centers. The figure below shows the worst-case HVAC Efficiency (a) and the best-case HVAC Efficiency (b) of different data centers. The figure depicts the segments in total energy consumption in the data center.

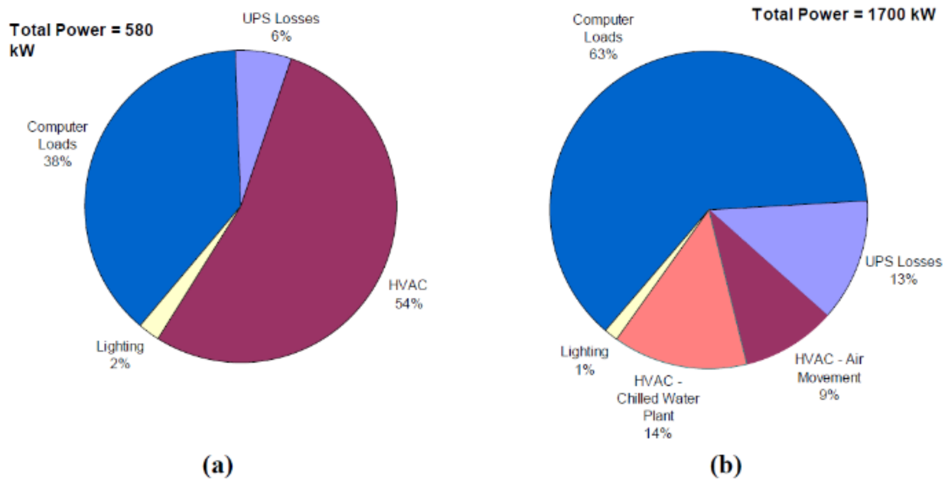

Figure 3:- Energy consumption break down [5].

Green Grid Association (TGG) works to improve the resource efficiency of information technology (IT) and data centers throughout the world. In 2007, The Green Grid published two "self-help" data center energy efficiency metrics for end users: power usage effectiveness (PUE™).[6]

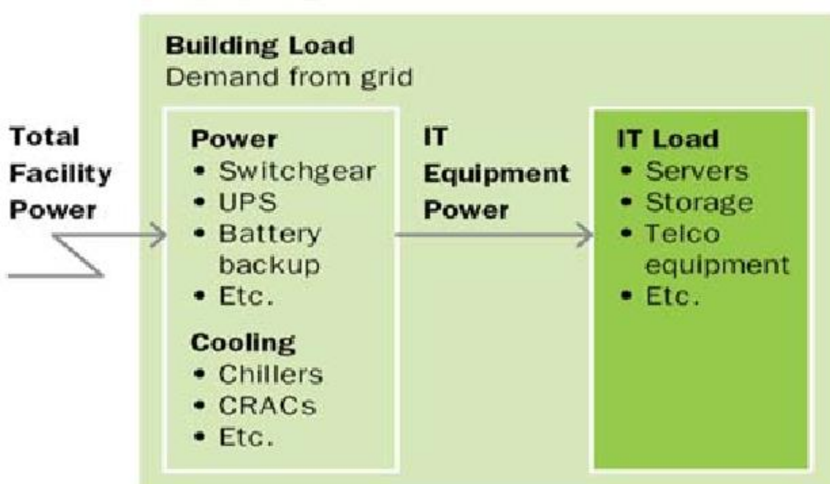

Figure 4:- PUE Calculation[6].

$Power\ Usage\ Effectiveness,$

$$PUE = \frac{Total\ facility\ energy}{IT\ Equipment\ Power}$$

$Data\ Center\ Energy\ Producttivity,$

$$DCeP = \frac{Useful\ work\ produced}{Data\ Center\ energy\ consumed\ to\ produce\ work}$$

Cooling in a data center is studied at different levels. Some of these are chip level, server level, rack level and room level. Cooling at the chip level includes the heat sink designing, thermal interface material use, location of chip, design of chip socket etc. At the server level, cooling is divided depending on the medium used to cool server namely air, water, oil or hybrid of this mediums. The cooling at server level typically used to be air cooling using fans. Modern advancement considers the use of warm water cooling, oil immersion cooling and hybrid cooling to achieve less cooling power consumption. Studying the flow of air through the rack and enhancing the flow of conditioned air towards the server using different methods are considered in the rack level cooling.

1.4 Thermal Management

In a data center, the ITE is enclosed in a cabinet to majorly save them from dust and other air particles known as racks. The size of the rack is measured in units of U where 1U is defined as 45mm (1.75 in). The racks usually vary in size, while the standard rack size defined by ASHRAE, is 42U in height (1.82m), in a data center it should have a footprint of 610mm by 1020mm[7].

Many steps have taken to improve thermal management of data center one example of is hot aisle-cold aisle arrangement. The arrangement specifies the architecture of data center to separate the cold supply air from hot exhaust air through alternating rows of perforated and non-perforated tiles. The cold air from the CRAC units is drawn through the perforated tiles to the computational equipment in the cabinets. The hot exhaust air is then pushed out and forced towards the ceiling of the room, where it is collected by the intake of the CRAC units and the cycle is repeated. The benefit of this configuration is the separation of the hot exhaust air from the cool inlet air, however, the complex recirculation in the upper portion of the room results in a degree of hot exhaust air being drawn into the racks in the cold aisle.

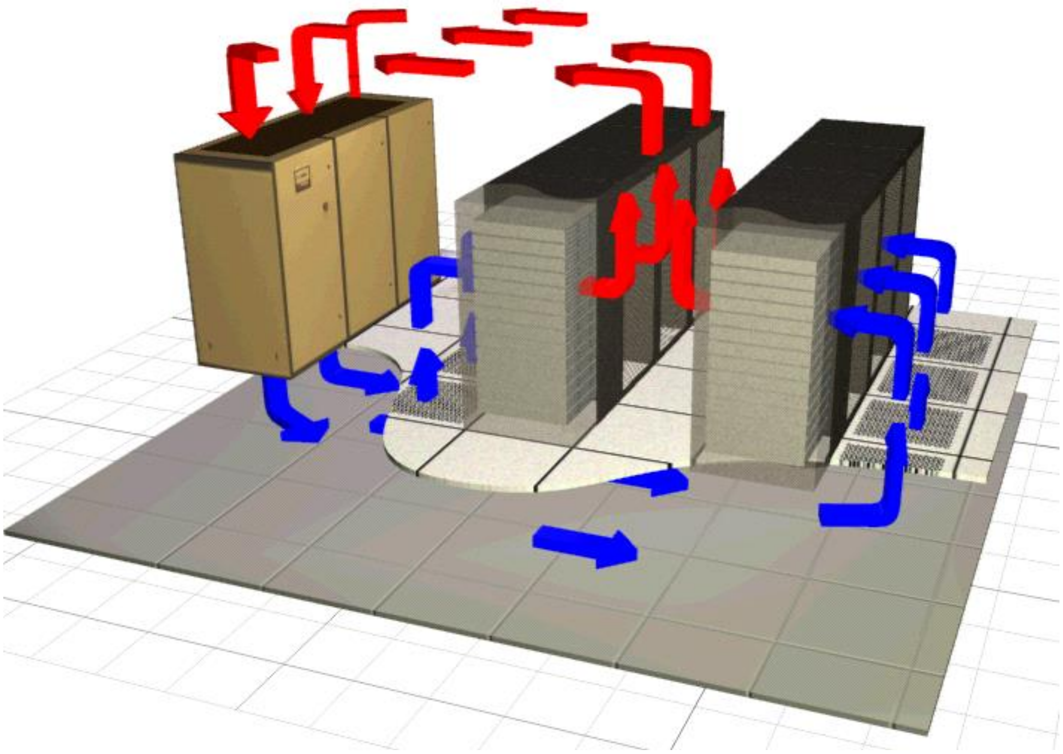

Figure 5:- Data Center hot aisle – cold aisle flow schematic: Center perspective[5].

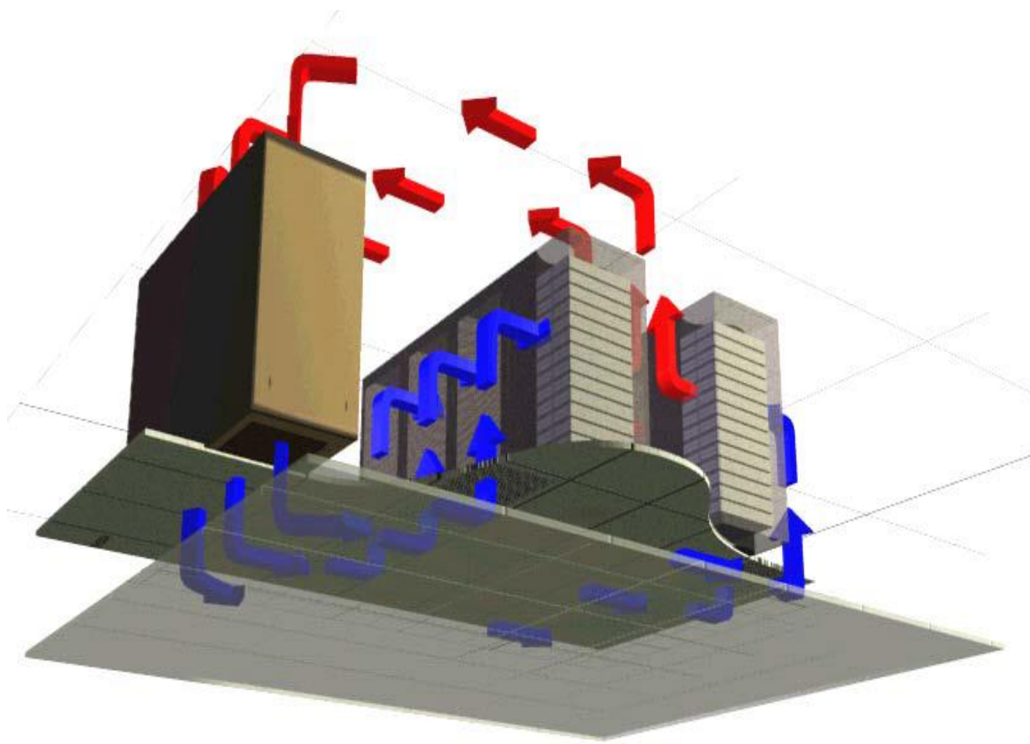

Figure 6:- Data Center hot aisle – cold aisle flow schematic: plenum perspective[5].

ASHRAE has standardized the working conditions in data center, by achieving the reliability of data center and allowing the operator to operate at most energy efficient mode through guidelines. In the psychrometric chart below, the figures includes the data for the two new classes added for the flexibility of operations in a data center.[7]

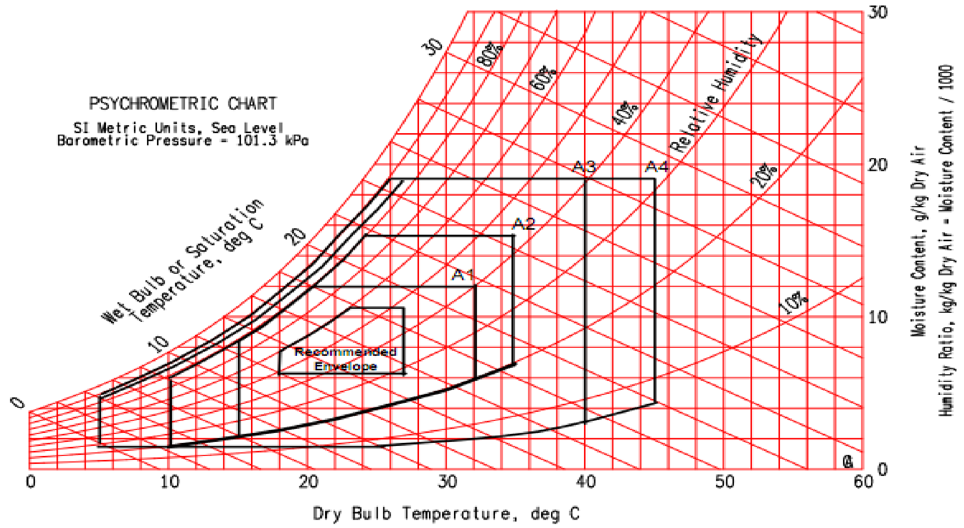

Figure 7:- ASHRAE Environmental Classes for Data Centers[8].

To achieve lower cost of ownership and high operating efficiency of data center, expanding operating ranges of humidity and temperature is desire of the hour. The table summarizes the updated ranges of operating temperature, dew point, altitude and humidity ranges for different classes published by ASHRAE in 2015[7].

Classes (a)	Equipment Environmental Specifications							
	Product Operations (b)(c)					Product Power Off (c) (d)		
	Dry-Bulb Temperature (˚C) (e) (g)	Humidity Range, non-Condensing (h) (i)	Maximum Dew Point (˚C)	Maximum Elevation (m)	Maximum Rate of Change(˚C/hr) (f)	Dry-Bulb Temperature (˚C)	Relative Humidity (%)	Maximum Dew Point (˚C)
Recommended (Applies to all A classes; individual data centers can choose to expand this range based upon the analysis described in this document)								
A1 to A4	18 to 27	5.5ºC DP to 60% RH and 15ºC DP						
Allowable								
A1	15 to 32	20% to 80% RH	17	3050	5/20	5 to 45	8 to 80	27
A2	10 to 35	20% to 80% RH	21	3050	5/20	5 to 45	8 to 80	27
A3	5 to 40	-12˚C DP & 8% RH to 85% RH	24	3050	5/20	5 to 45	8 to 85	27
A4	5 to 45	-12˚C DP & 8% RH to 90% RH	24	3050	5/20	5 to 45	8 to 90	27
B	5 to 35	8% RH to 80% RH	28	3050	NA	5 to 45	8 to 80	29
C	5 to 40	8% RH to 80% RH	28	3050	NA	5 to 45	8 to 80	29

Figure 8:- Equipment Environmental Specification[7].

1.5 CFD analysis

Server has various component included in its detailed design considering the flow pattern to remove the heat generated by the various component in server. To accommodate components of server and consider various thermal aspects in a server we require a software that answers all questions. The commercial software developed by Future Facilities, 6SigmaET & 6SigmaRoom is been chosen to house all the specification of server considered. The CFD analysis of server helps us in reducing cost and time used in achieving results through experimentation. The CFD analysis is more productive and giver numerous opportunities to improve the cooling of data center.

1.6 Motivation

The big giants of data center industries are building a large data center. The demand of energy saving can be achieved with efficient solution to reduce energy consumption by data center. Cooling contributes as one of the major energy consumption section of data center, efficient utilization of energy can prove to decide factor in total energy consumption of data center saving a large amount of energy in a large-scale data center which is the need of the hour. Liquid cooling is a promising alternative to the traditional air-cooling method. Water is extensively used in the liquid cooling technique. Cold plates using water as the coolant, to dissipate heat generated by the chip on which the cold plate is mounted. Water enjoys few advantages over the air as a medium of cooling. Water in contrast with the air has high heat carrying capacity, it has low cost of transportation and server operates efficiently at higher utilization.

The work seeks to understand the effect of hybrid cooling over air cooling when keeping to simulation on a data center model prepared in 6SigmaRoom software. The comparison will show the difference of power utilization by air and hybrid cooling and energy efficient solution to data center energy consumption problem.

Chapter 2:- Literature Review

This chapter will review the data from different research papers that was used in this work. The chapter will review the fan speeds at different inlet temperatures of air-cooled server, flow rate ratio curve of hybrid cooled server and review best practices of different percentage of liquid cooling used in hybrid cooled servers.

1.) The Cisco C220 M3 is placed under experimentation to do a comparative study of high ambient inlet temperature effects on the performance of Air vs Liquid cooled I.T. equipment. A closed case is presented to show that the liquid cooling provides wider inlet temperature which are viable to operate a data center with reduced risk of leakage current and sustain the performance and reliability of the IT equipment. The server is placed in an environmental chamber to operate it under different environmental conditions or boundary conditions and bash script is used to give the computational load to server for given amount of time and IPMItool is used to measure the parameters. The fan speeds are observed to be operated based on the coolant inlet temperature. The CPU core temperatures remains below high operating limit 86 ^{o}C at 100% computational load and 45 ^{o}C coolant inlet temperature[9]. We derived the relation of the inlet air temperature for air-cooled server and fan speed that can be used in model to extract other useful results. Defining the fan speed at environmental condition depending the following data gives us the flow rate through the server or test chamber. The flow rate is further used to find the flow rate ratio curve which is primary result we are desiring in this work.

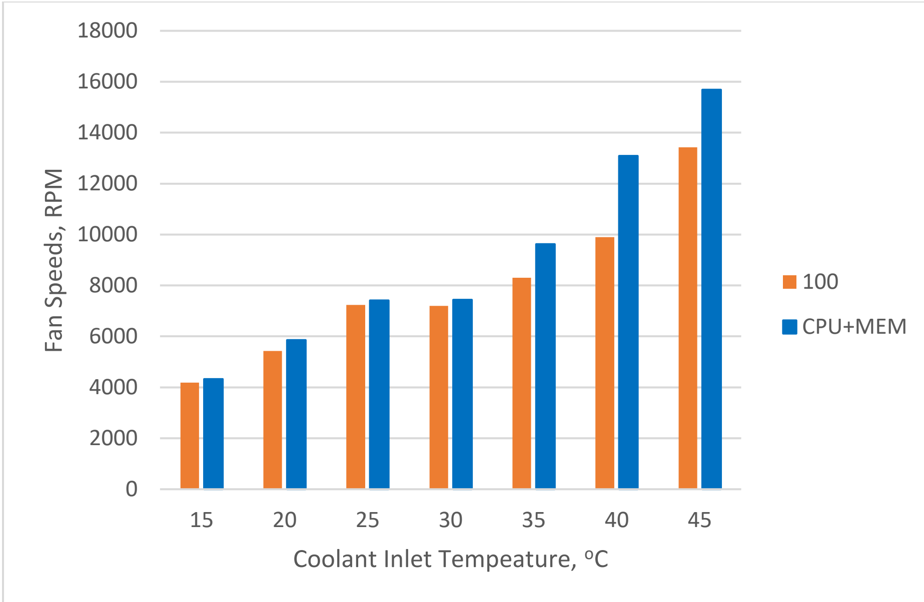

Figure 9:- Average Fan speed varying with Inlet air temperature[9].

2.) In this literature, to find power consumption minimization in hybrid cooled server by fan reduction a Cisco C220 M3 hybrid server is used for experimentation. The server is experimented on the airflow bench to find result for fan power reduction. The server is put to data acquisition setup to get the data for the results. PWM control and tachometer are connected to fans and fan speeds are monitored with IPMItool and is controlled by external PWM signal generator. The analysis gives us the flow rate, component temperatures and system resistance curve for hybrid cooled server. The paper shows us rise in component temperatures with rise in inlet air temperature[10].

3.) In this literature review, the thermal performance modeling of hybrid liquid-air cooled server is done. The results are interpreted with three parameter model which allows characterization of the major factors affecting the heat recovery efficiency of the system. The objective of the analysis was to leverage data center level measurements to characterize system level performance[11]. The analysis helps use to find the percentage of liquid cooling in a server. The graph below shows that 42.2%, 55.56% and 64% of liquid cooling is best practice.

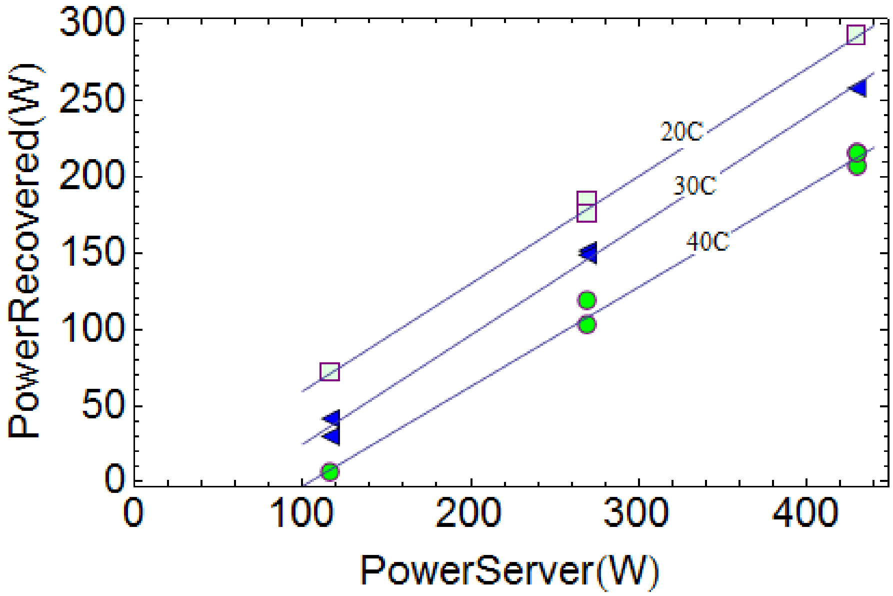

Figure 10:- Recovered Power vs Sever power at different workloads and different average temperatures[11].

4.) The research in this literature shows, the correlation between high ambient inlet and reliability of IT hardware, which says that the COP of the data center cooling infrastructure has negative impact on leakage current which is direct function of chip operating temperature, the conclusion was derived from analysis on data center inlet temperature and chip leakage power through a prepared model. The study shows the importance of determining the optimum operating strategy for data center while trading off the infrastructure energy saving, challenging the perception of hot air cooling[12]. A study shows that, the data center power consumption, which is a function of coolant temperature, the computational state of chip, has not shown any improved efficiency during maximized free cooling, since an additional leakage is accompanied with higher coolant temperatures. The study was done to overcome the limitation of air cooling for operating temperature. The energy efficient liquid cooling was examined for its temperature dependent leakage at higher temperatures[13]

Chapter 3:- Server Description

In an air-cooled data centers, the air is supplied to the server from the computer room air conditioning (CRAC) units. The air supplied from CRAC unit is at 18C in a normal data center. The data center equipment is placed on a raised plenum and air from CRAC units is blown under-plenum. The cooled air enters the computer room from perforated or permeable tile in cold-aisle containment. The cold air then passed through racks and is warmed by the servers. The hot air from server is sent back to CRAC unit through hot aisle. The hot air in the CRAC unit is cooled with the help of chilled coils placed inside the CRAC unit. During this process air is diminished due to blending of hot and cold air, thus expanding proficiency.

An air-cooled server uses only air to cool the heat generated by power consuming components in server. A typical air-cooled server uses fans and heat sinks to cool the major components of server. In a server, CPU or GPU chip consumes highest power generating thus are primary heat generating components. Such components are cooled with help of designated cooling components like heatsinks and cold plates. An air-cooled server has push or pull configuration for air flow in server. The servers are normally designed to channel the air flow majorly through heat generating components though baffles or ducts.

3.1 Server Specification

The Cisco® UCS C220 M3 rack server is designed for performance and density over a wide range of business workloads from web serving to distributed database. The enterprise-class UCS C220 M3 server extends the capabilities of Cisco’s Unified Computing System portfolio in a 1U form factor with the addition of the Intel Xeon E5-2600 v2 and E5-2600 series processor family CPUs that deliver significant performance and efficiency gains. In addition, the UCS C220 M3 server provides 16 DIMM slots, up to 8 drives and 2 x 1 GbE LAN-on-motherboard (LOM) ports delivering outstanding levels of density and performance in a compact 1U package.[14]

Figure 11:- Front View of Cisco C220 M3 Server[15].

3.2 Server Motherboard

The motherboard of Cisco C220 M3 server has following specifications: -

Server Chassis	**1U Small form factor Server**
CPU	Intel Xeon E5- M2600 v2 & ME5 – 2600 processor
CPU TDP	135W
DIMM	16 DIMM slots (Samsung 8gb 2rx8 pc3l-12800s-11-13-f3)
Chipset	Intel® C600 series chipset[16]
HDD	Savvio® 10K.5[17]
Ethernet	Intel® Ethernet Converged Network Adapter X520-SR2
PCIe Riser Card	USCRaid9271CV-8I LSIMegaraid 9271cv-I Pci Express SAS/SATA Ports

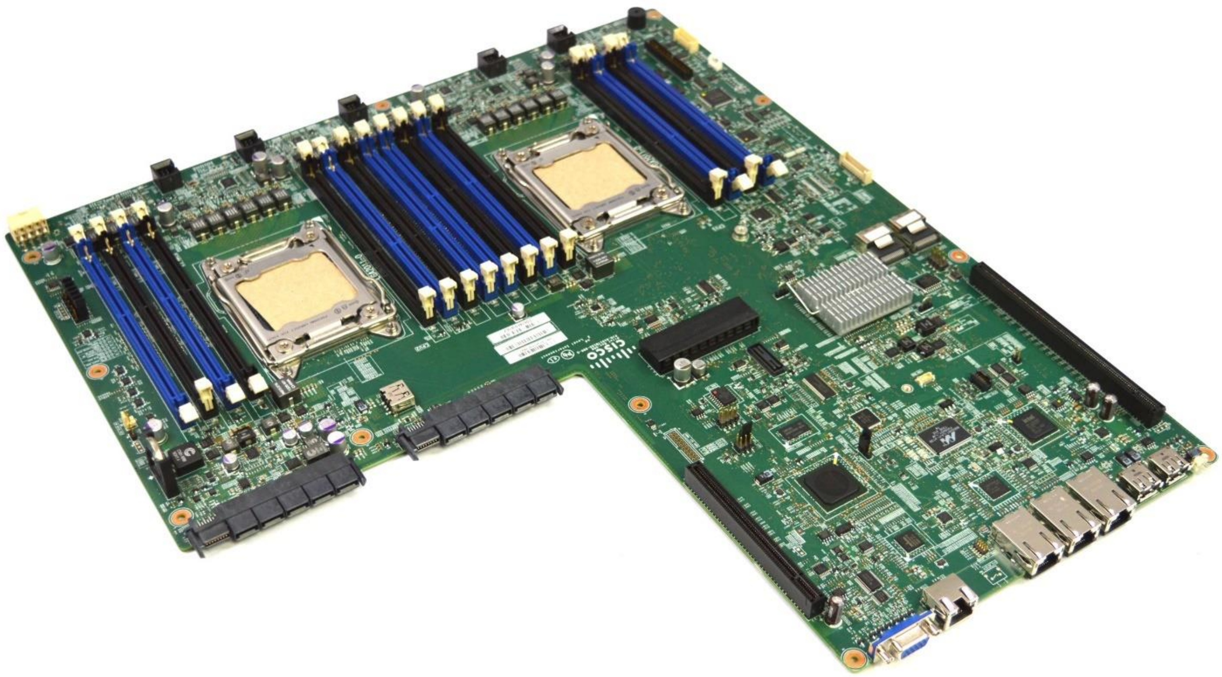

Figure 12:- *Motherboard of Cisco C220 M3 Server*[18].

3.3 Server Chassis

The server chassis is 44mm tall as it is server with 1U form factor. It is a server which is a small form factor disk drive model. The server chassis is 1.2mm thick with zinc plated corrosion free body. The chassis has 5 fans and an array that can house 8 drives. The server being a small form factor server cannot house tall heatsink for heat generating components.

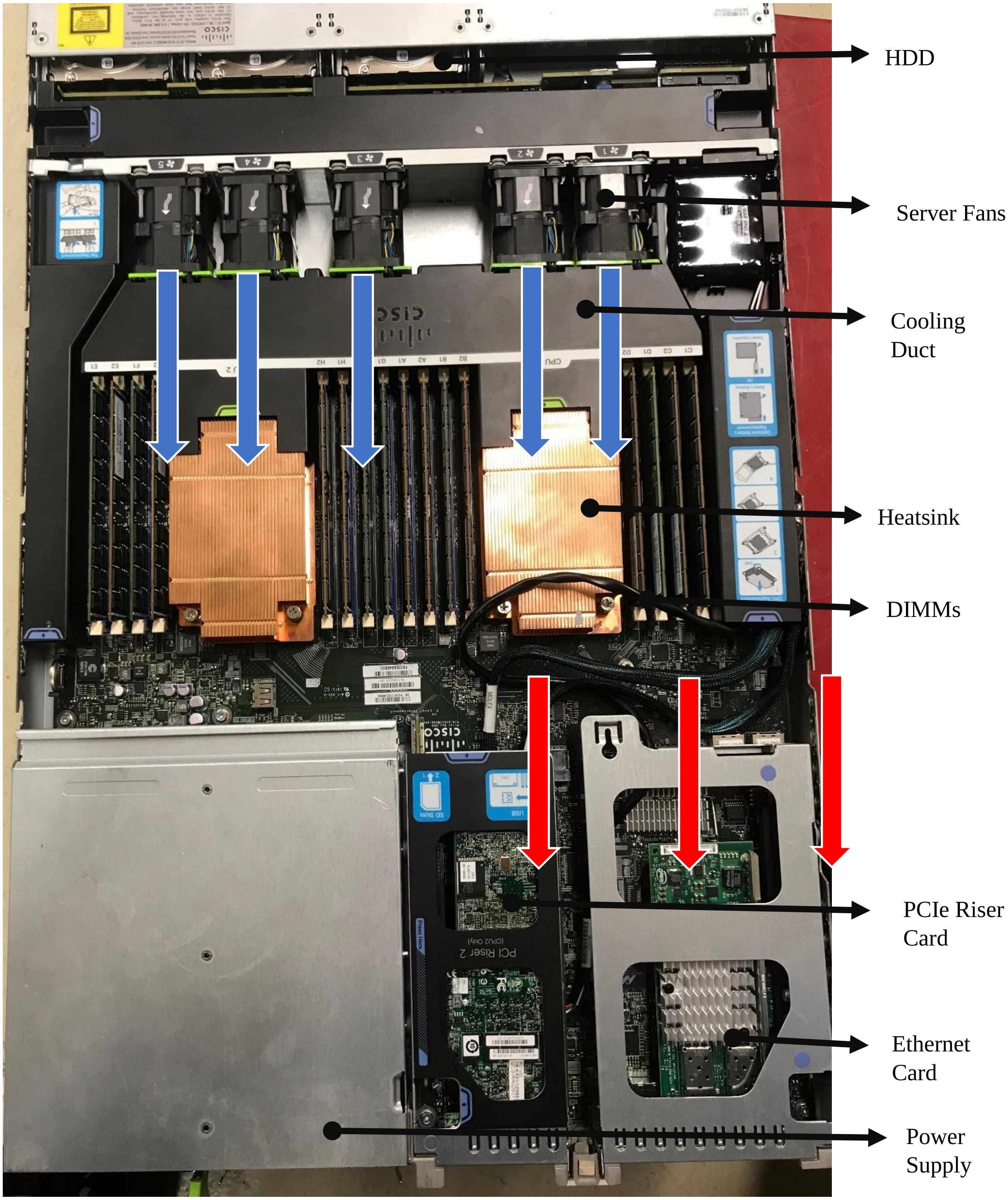

Figure 13:- Top view of Cisco C220 M3 Server with airflow rate.

3.4 Server Fans

The server has 5 hot-swappable fan modules that provide front to end cooling. The dimensions of the fans are 40cmx40cmx56cm. The speed of fans is controlled by motherboard through an algorithm depending on core temperature of CPU. The fans are placed just before server chipset is accompanied with a cooling duct to efficiently use air coming from fans.

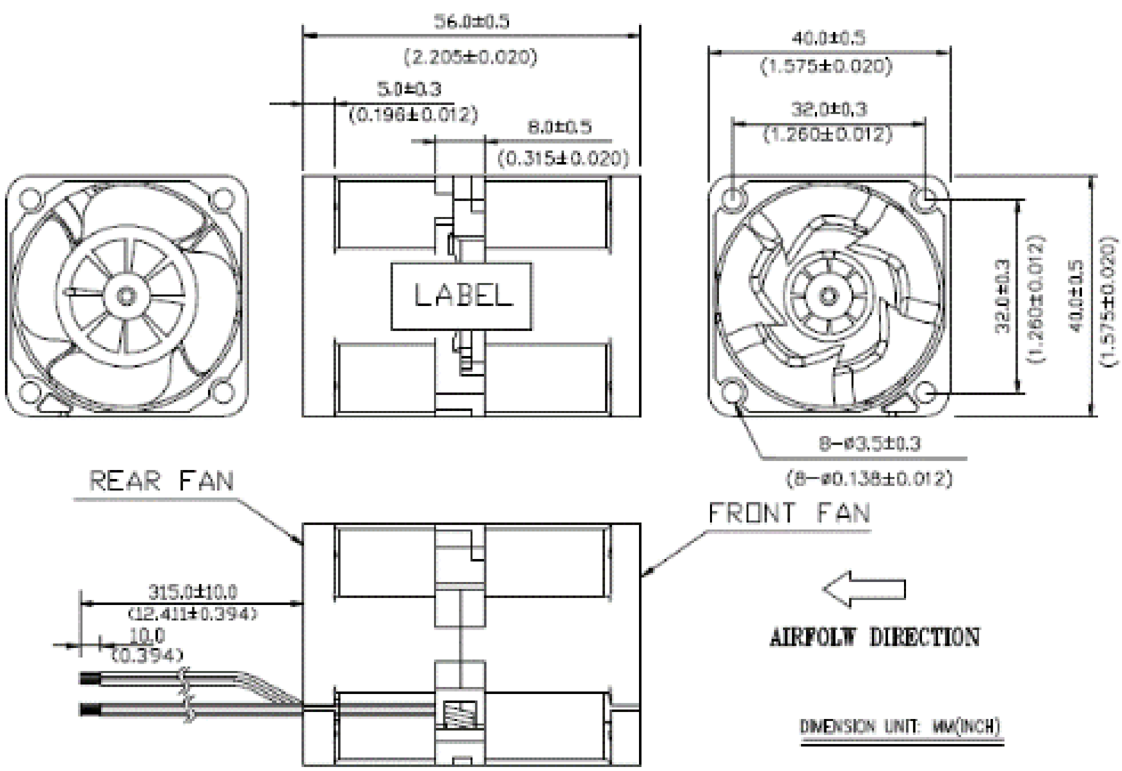

Figure 14:- *Dimensioning Model of Fans*[19].

The fans have single phase motor and four poles which are defined as DC brushless axial flow fans. The fans have 32.179 CFM as the highest flow rate under static condition when the static pressure is zero and 2.631 inch of water is maximum air pressure produced at zero airflow. The highest rated speed of the fan is 16000 rpm.[19]

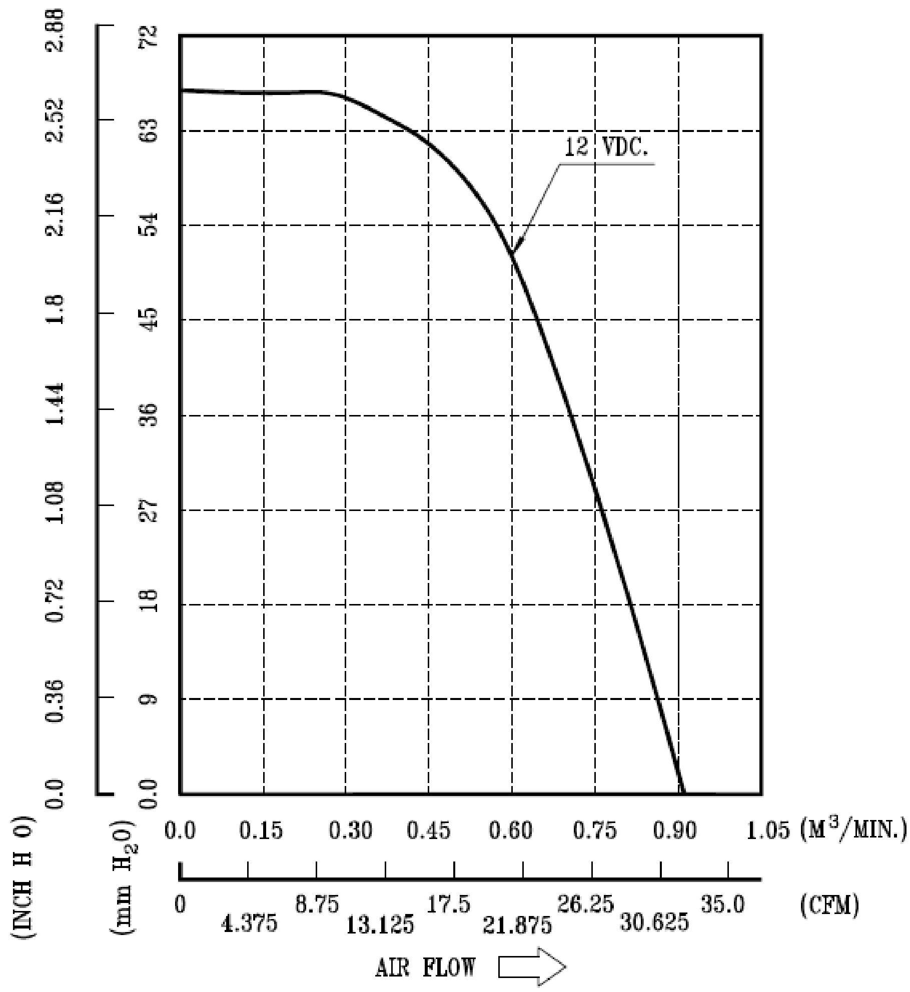

Figure 15:- P & Q Curve of Hot Swappable Fan[19].

To calculate amount of air required to cool a system if power dissipated and temperature rise is known it can be done accurately. To select the best air mover for the application it is important to know the pressure required to force the air through the system. If an improper selection is made. The power input to air mover, size, and acoustic noise will be high if a improper selection is made. Or it will result in overheating if the estimation is low.

As mentioned, the importance of pressure required to force the air through system, it can be measured through Airflow Bench. The Airflow Bench helps in finding the system resistance curve which can used to select the air mover. The intersection of the system resistance curve and fan curve gives us theoretical operating condition.

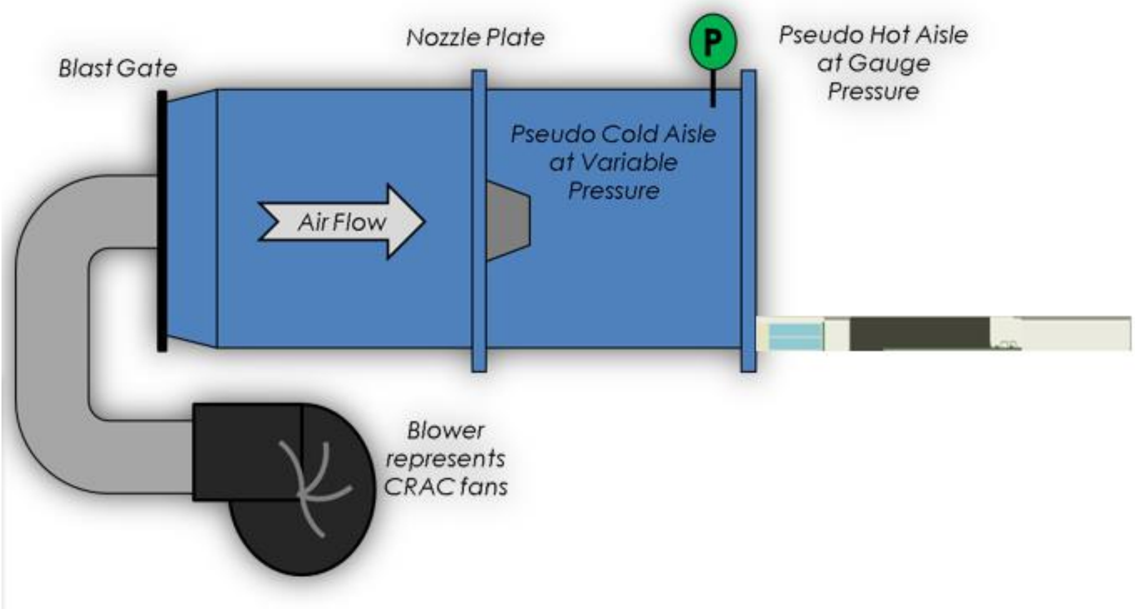

Figure 16:- *Schematic Diagram of Air Flow Bench*[20].

The airflow bench consists of a motor attached with blower for airflow. It has a blast gate that can be changed to regulate the flow of air. The air flow bench has nozzle plates which has different sized nozzles. The nozzles of different size provide different flow rate through equipment. The nozzle selection chart is provided to select the size of nozzle to be used.

A tensioned diaphragm and electrode are used to form a variable capacitor. The change in pressure forces diaphragm to move thus increasing or decreasing the capacitance. This change in voltage is converted in values for pressure through transducer with a differential pressure range of 0 to 5 inch of water.

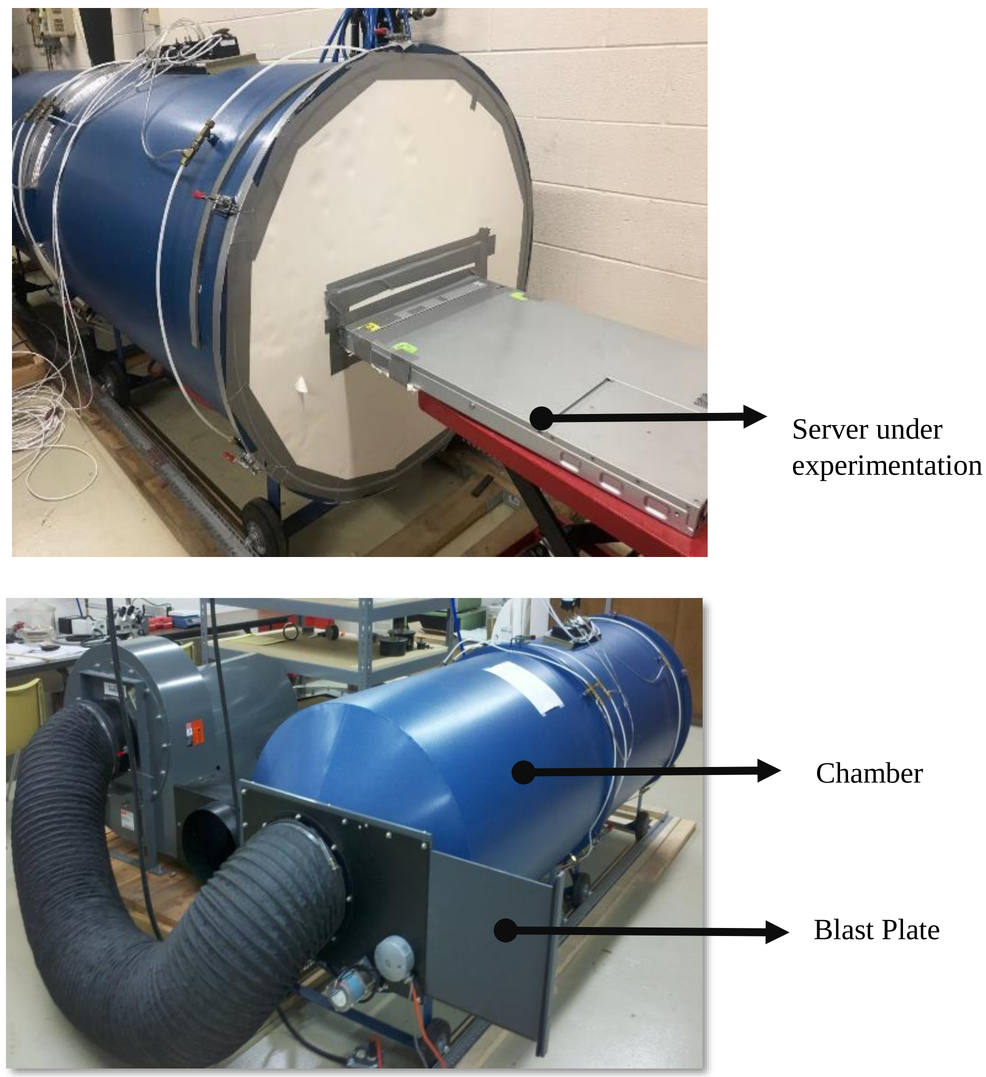

Figure 17:- Server Under experimentation.

The airflow bench consists of a motor attached with blower for airflow. It has a blast gate that can be changed to regulate the flow of air. The air flow bench has nozzle plates which has different sized nozzles. The nozzles of different size provide different flow rate through equipment. The nozzle selection chart is provided to select the size of nozzle to be used.A tensioned diaphragm and electrode are used to form a variable capacitor. The change in pressure forces diaphragm to move thus increasing or decreasing the capacitance. This change in voltage is converted in values for pressure through transducer with a differential pressure range of 0 to 5 inch of water.

As suggested earlier the airflow bench is used to find the system resistance curve of server. A range of working flow rate, CFM is considered depending on the fan curve and number of fans. In the case of Cisco C220 M3 server a range of 7-110 CFM is selected to plot the differential pressure and flow rate graph is considered. In these cases, depending on the range of flow rate we selected 2 nozzle sizes 1” (7-20 CFM) and 1.6” diameter (14-110 CFM). The air flows from blower through flexible hose into chamber through a fully open blast gate. Transducer gives us the values of differential pressure across system. Combining data received through transducer we can plot system resistance curve[20].

Figure 18:- Nozzle Disc of Airflow Bench.

Figure 19:- Pressure Gauge of Airflow Bench.

Figure 20:- Transducers on Airflow Bench.

Chapter 5:- CFD Analysis

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by using numerical process that is done by solving the mathematical equations which govern these processes. CFD helps in analyzing relevant engineering data through the results[21]Few of such data are:

- Conceptual studies of new designs.
- Detailed product development.
- Troubleshooting.
- Redesign.
- Foresee, recreate and examine different mechanical properties of design.
- Optimize and analyze design process or products

For Computational fluid dynamics, foundation of experimental fluid dynamics was initially developed around France and England in 17th Century. The 18th and 19th century saw development of theoretical fluid dynamics primarily in Europe. In 1970s, due to computer algorithms that existed at that time, solutions where essentially limited to two dimensional flows. The real world of computational fluid dynamics is machines – compressions, turbines, flow duct, airplanes etc. which is a three-dimensional world but in 1970s the computer speed and storage capacity was not sufficiently developed to accommodate such changes. By 1990, the scenario for computational fluid dynamics was completed changed due to substantial systems to support CFD analysis. CFD initially was used was used for aircraft and automobile industries by now it has found its application in various fields like data center industries, telecommunication industries, civil engineering, naval architecture, turbomachinery, electrical and electronic engineering, biomedical engineering, chemical process engineering, hydrology and oceanography etc. CFD results reduces total effort required in laboratory for testing and experimentation. Computational fluid dynamics results are directly analogous to wind tunnel results obtained in a laboratory – they both represent sets of data for given flow configuration at different Mach number, Reynolds number, Weber number etc[22]

CFD uses numerical methods, called discretization's for governing equations of fluid mechanics to develop approximations in fluid region of interest. Where discretization is a process by which a closed form of mathematical expression, such as a function or a differential or an integral equation involving functions, all of which are viewed as having an infinite continuum of values throughout some domain, is approximated by analogous expressions which prescribe values at only a finite number of discrete points or volumes in the domain.

– Governing differential equations: algebraic.

– The collection of cells is called the grid.

– The set of algebraic equations are solved numerically (on a computer) for the flow field variables at each node or cell.

– System of equations are solved simultaneously to provide solution[21].

The solution is post-processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.). Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the "grid" or the "mesh."

General conservation (transport) equations for mass, momentum, energy, etc., are discretized into algebraic equations. All equations are solved to render flow field[21].

5.1 CFD Governing Equations.

The keystone of computational fluid dynamics is the fundamental governing equations of fluid dynamics—the continuity, momentum and energy equations and codes of computational fluid dynamics are based on Navier-Stokes Equation. The numerical solution of heat transfer and fluid flow is obtained using these three fundamental equations.[22] They are the mathematical statements of three fundamental physical principles upon which all of fluid dynamics is based:

(1) mass is conserved;

(2) F = ma (Newton's second law);

(3) energy is conserved.

Simplifying and solving conservation of mass, momentum and energy equation we derive the following basic computational fluid dynamics governing equations.

- Law of conservation of mass for generalized case;

$$\frac{\partial \rho}{\partial t} + \text{ div } (\rho u) = 0$$

- Law of conservation of momentum for generalized case;

$$\frac{\partial (\rho u)}{\partial t} + \text{ div } (\rho u \vartheta) = \text{ div } (\mu \text{ grad u }) - \frac{\partial P}{\partial x} + S_M$$

- Law of conservation of Energy;

$$\frac{\partial (\rho i)}{\partial t} + div\ (\rho i \vartheta) = -p\ div\ \vartheta + div\ (k\ grad\ T) + \emptyset + S_i$$

5.2 Computational Domain

A CFD simulation begins with the creation of the problem geometry. The region to be modeled is often called the computational domain, and the geometry consists of the boundaries of that domain and all the objects contained within. While computer-aided design (CAD) software focuses on the solid objects, CFD focuses on the space inside or in between objects, where the fluid is likely to flow. [9] The computational domain is a closed volume in a finite region where the fluid flow will be considered. This closed volume is called control volume of fluid flow. The control volume is a limited locale or space where the governing equation are applied through discretization.

The solutions to governing equations are obtained by fixing boundary conditions for computational problem that is considered. The boundary conditions are normally defined by external conditions like ambient temperature, velocity, pressure, fluid viscosity, mass flow in and out and other environmental conditions. The boundary conditions also specify type of heat transfer in the computational domain or control volume it may either be conduction or convection along with radiation factors included. The boundary conditions depend on type of domain wall it may be open, closed or symmetrical in nature.

The major steps in process of CFD numerical solution are: -

1. Defining geometry of problem.
2. Dividing geometry or control volume in discrete cells called meshing.
3. Defining boundary conditions for governing differential equations.
4. Solving governing equations through applied boundary conditions.

The differential equations are applied to discrete cells created called the mesh. The solution of CFD depends on type of mesh considered and generated for a particular CFD problem. Hence, the discretization process is important as it helps in creating algebraic equation of CFD problem. The discretization of problem can be done in following ways: -

- Finite Element Method
- Finite Volume Method
- Finite Difference Method

Finite Element Method is normally used for the analysis of the solid structures, but it can also be used for analysis of fluid flow. In FEM, discrete cells can be structured or unstructured and link between solution representation and geometrical representation of domain is strong. In finite difference method, differential terms of governing equations are discretized in series of grid points. Finite difference method is not preferable when the coefficients involved in equation are discontinuous. In finite volume method, differential equations of discrete cells in defined control volume are integrated for solution of CFD problem.[23]

The computational fluid dynamics code considered for numerical analysis of Cisco C220 M3 server and rack level analysis in 6SigmaEt and 6SigmaRoom is finite volume method where domain is discretized in control volumes or locales. In Finite volume method discretization process is divided in two steps. In the first step, the integration and transformation of partial differential equation is done over element balanced equations. An integration quadrature of specified order of accuracy is used to change the surface and volume integrals into discrete algebraic relation over elements and their surfaces. In second step, the algebraic relation is transformed to algebraic equations by choosing interpolation profiles to approximate the variation of the variable within the element and relate the surface values of the variables to their cell values.[24] It also allows the use of arbitrary geometry, i.e. allowing unstructured and structured cells for numerical analysis.

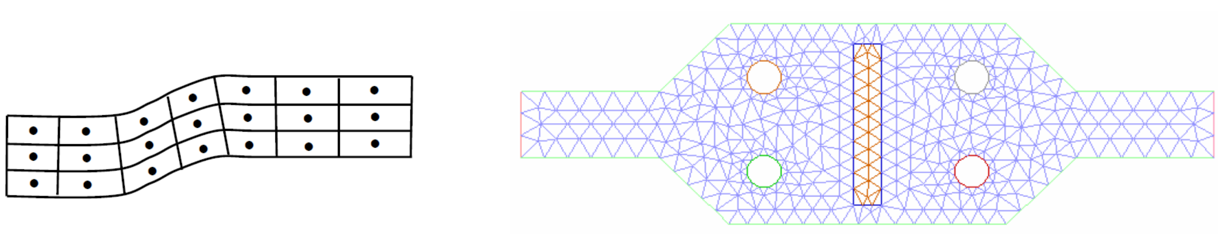

Figure 21:- Graphical Representation of Finite Volume method Discretization[25].

5.3 Turbulence Modelling

Reynolds Number defines flow of the fluid, it can be stable, unstable or random in every direction. A higher Reynolds number gives an unstable flow. Turbulent Flow is defined as variation of velocity and all other properties in random and chaotic way[26]. When Reynolds Number of a fluid goes above critical Reynolds number Re_{crit}, the glow becomes turbulent. The Reynolds Number when is below Re_{crit} the flow is turbulent flow. 6Sigma software used K-Epsilon turbulent model for CFD Solution.

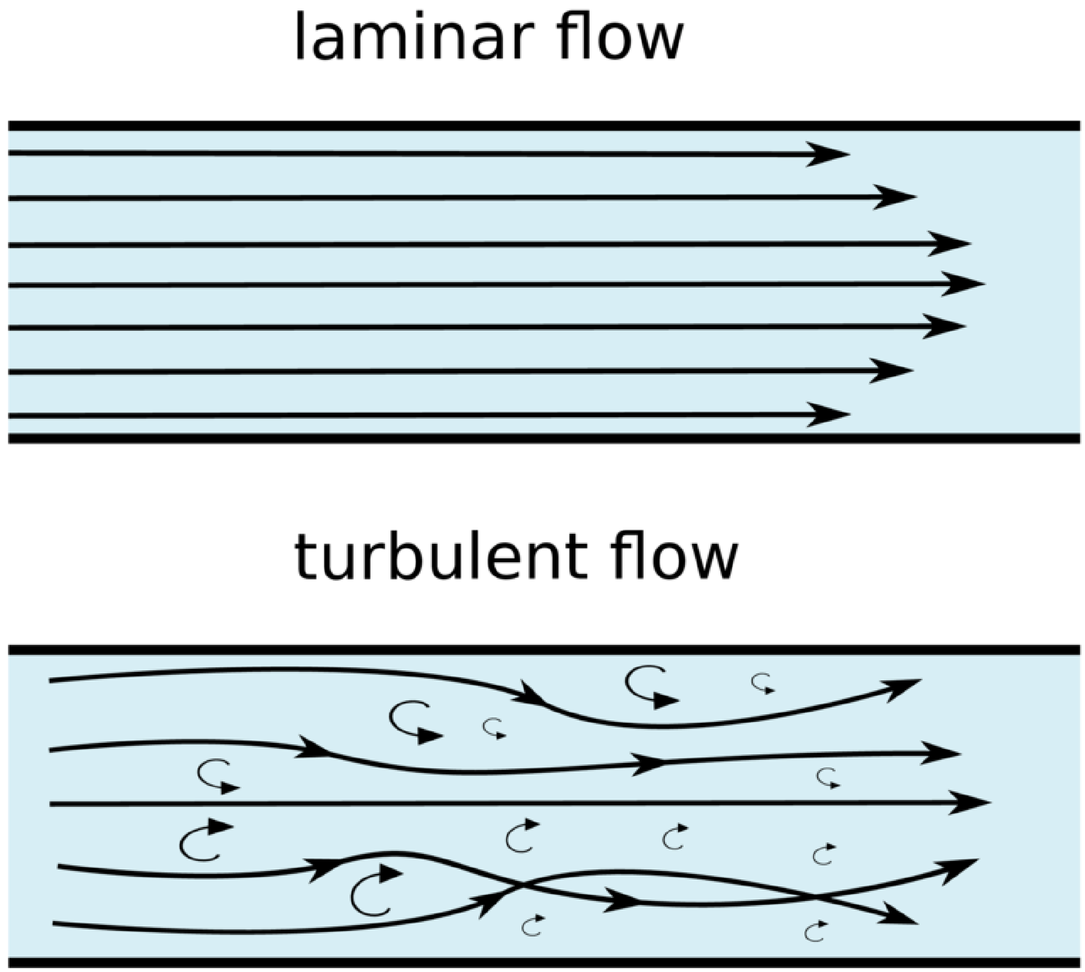

Figure 22:- Sketch of difference between Laminar and Turbulent Fluid Flow in a Pipe[27].

5.4 K-Epsilon Turbulence Model

K-Epsilon Model analysis is one of model that is used in Computational fluid dynamics modelling to analyze turbulent flow in model. The model basically uses 2 equations defined below to represent turbulent properties of flow. The transport equations of model are for turbulent kinetic energy and dissipation[28].

- For Turbulent Kinetic Energy,

$$\frac{\partial(\rho k)}{\partial t} + div\ (\rho k U) = div\ \left[\frac{\mu_t}{\sigma_k}\ grad\ k\right] + \ 2\mu_t S_{ij} S_{ij} - \ \rho\varepsilon$$

- For Turbulent Dissipation,

$$\frac{\partial(\rho\varepsilon)}{\partial t} + div\ (\rho\varepsilon U) = div\ \left[\frac{\mu_t}{\sigma_\varepsilon}\ grad\ \varepsilon\right] + \ C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t S_{ij} S_{ij} - \ C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

5.5 Grid Configuration and Meshing

6SigmaET uses cartesian grid in grid configuration, which helps in specifying maximum and minimum cells across the geometry of model. The grid configuration helps in defining boundary conditions for cell dimensions. Grid configuration allows change multiple grid formation parameters like aspect ratio, target cell count, expansion ratio, maximum sizes in X, Y, Z. In 6Sigma each component is readily available as object with default gridding rules to automatically mesh component. The default set of rules helps in selecting best grid for simulation of model.

5.6 Objects

6Sigma uses components that are directly identical to standards used in electrical industry. Hence, it provides components that can directly be mounted on PCB while modelling. The nomenclature for objects is like ones used in industry. 6Sigma parametrically know the functions of objects which makes meshing, gridding and analysis easy. The objects are directly available to use with or without further modification depending on the CFD problem or requirement. Objects commonly used which are available in software makes it easy to design a complete PCB.

5.7 Test Chamber

The model can be tested under different flow conditions in the test chamber which is a kind of faux wind tunnel. It acts as an enclosure for the server chassis, which can be used to attach different environment and prescribe the flow. In this case environment attached to test chamber is ambient temperature for air cooling. Also, front and rear end of test chamber are specified as an open hole for the air to let in and out and top, bottom, left and right side of test chamber are specified as wall.

5.8 PAC

PAC Study in 6SigmaET® stands for parameterize, analyze and compare, which is used for determining results of various alternative cases through parametric analysis. In PAC study, initially input and output parameter are defined. The input parameter can be changed depending on the requirement of the model and results to be obtained. The input parameter variations give output parameters through parametric analysis. The changes to input parameter can be done in PAC matrix. The model can be simulated for different input condition with the help of PAC matrix and updated output parameter in matrix after simulation or parametric analysis helps in easily comparing parameter depending on different condition applied to test chamber. By using PAC study, various analysis can be done like mesh sensitivity analysis can also be performed.

Chapter 6:- Server CFD Modeling

The modeling of Cisco C220 M3 Server is done using 6SigmaET a modelling software developed by future facilities. The server shelters number of components. Some of the components affect the flow of fluid in the server and some may not. All this component on motherboard are hard to model as they might not resist the flow of fluid to obtain analysis results. Similarly, there are few components that contribute negligibly to total power generated by server and is difficult to model all such components. While modeling the server all the major components restricting the flow of fluid and major power generating components are considered in the model. The heat generating components in server are defined on Thermal Design Power (TDP).

Device	Thermal Design Power (TDP)
CPU	135 W
DIMMS	3.5W
Chipset	8W
Fans	0.6-7W
Hard Drive	3.5 W
Ethernet Port	6.8W
PCIe Riser Card	6.8W

The server is designed in test chamber as explained in previous chapter to provide external environmental condition for air-cooling.

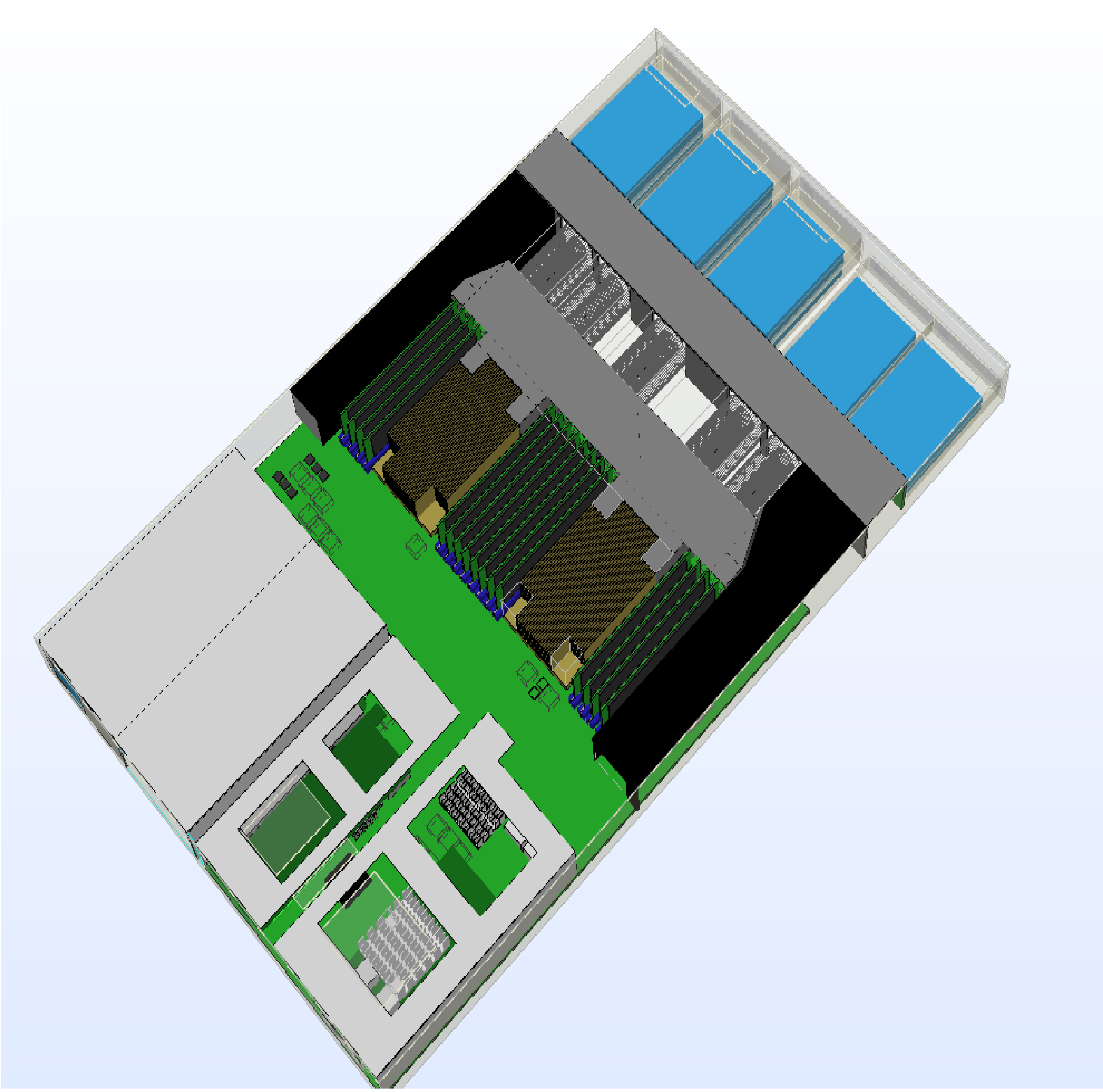

Figure 23:- Cisco C220 M3 6SigmaET Model.

6.1 Heatsink and Duct Modelling

The heatsink is modelled and placed above the CPU to provide cooling to it since it is primary heat generating component in server. The heatsink dimension have been considered from actual server. The model of heatsink of CPU is shown in figure below.

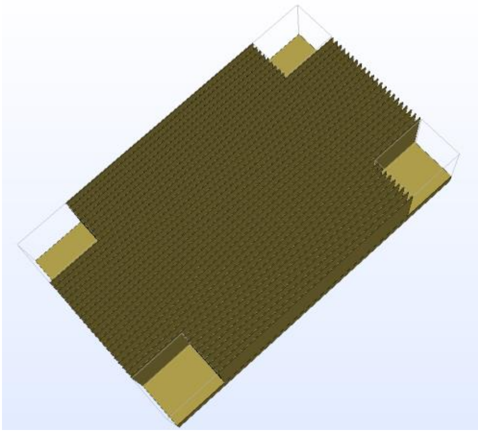

Figure 24:- Heatsink Model of Server.

The duct in 6SigmaET is designed as a solid obstruction. The duct is placed between fans and chip. The duct directs cold air from fans to chips via heatsinks. Cooling duct concentrates cold air towards heat generating component mainly chip (Heatsink), DIMMs and chipsets. The model of cooling duct is shown below.

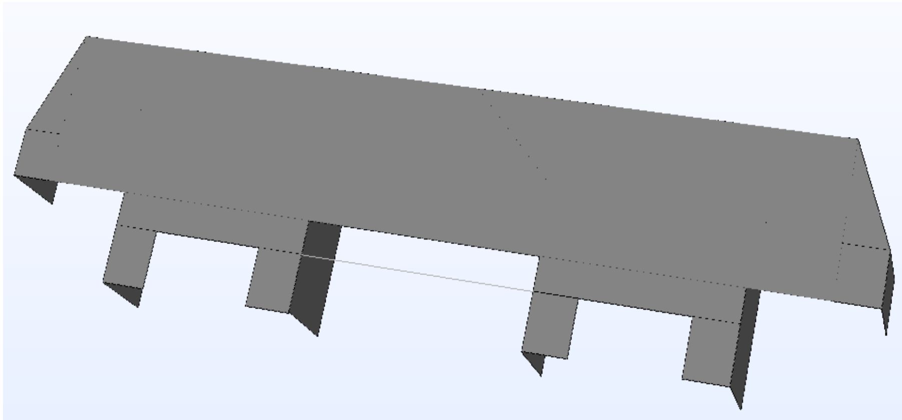

Figure 25:- Front view of Cooling Duct.

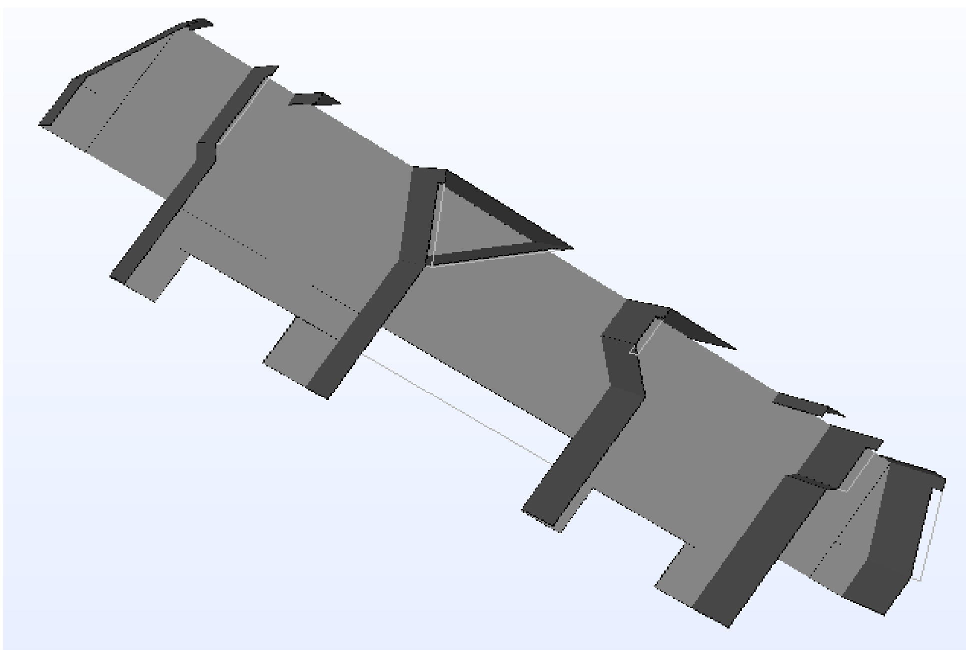

Figure 26:- Rear View of Cooling Duct.

6.2 Server Components

1. Ethernet: - Intel® Ethernet Converged Network Adapter X520-SR2

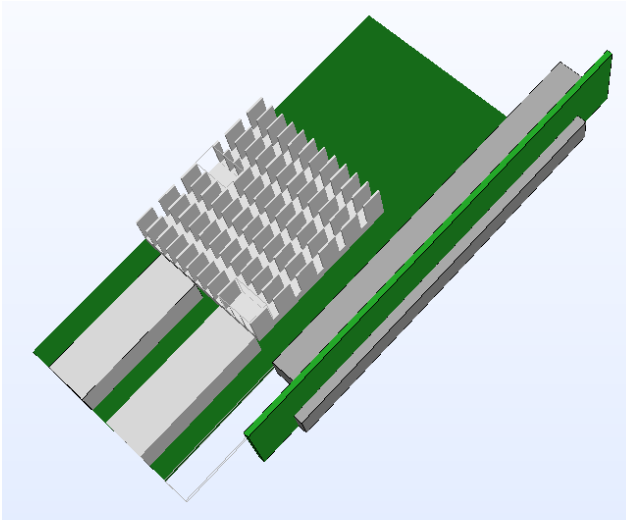

Figure 27:- Ethernet Card Model of Sever.

2. PCIe Riser Card: - USCRaid9271CV-8I LSIMegaraid 9271cv-I Pci Express SAS/SATA Ports

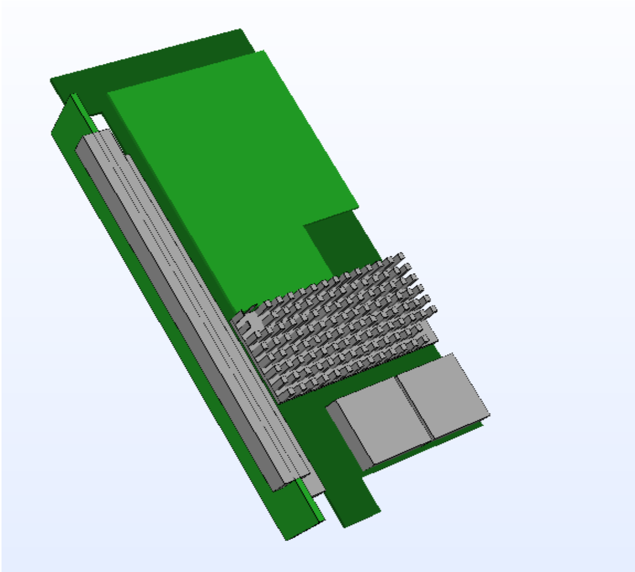

Figure 28:- PCIe Riser Card Model of Server.

Chapter 7:- Results

7.1 Model Calibration: - System Impedance Curve

Impedance is defined as obstruction to movement, but in case of server it is obstruction to flow of air. A pressure gradient is formed upstream of the obstacles as flow of air slows down due to resistance or impedance. This static pressure drop represents the impedance of the system and works in opposition to the flow. [29]

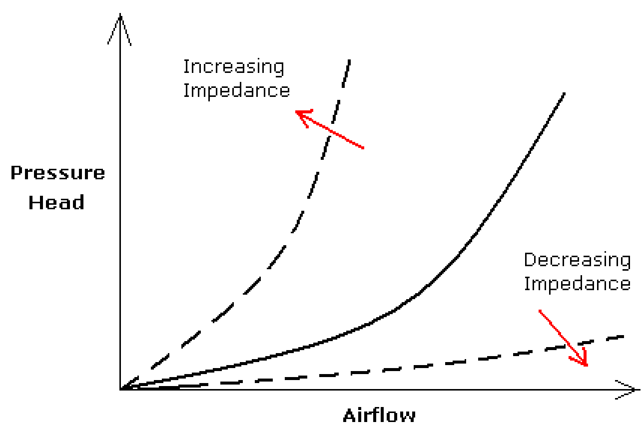

Figure 29:- General representation of system resistance curve[29]

As discussed earlier, the system resistance curve of server was experimentally characterized to validate the model prepared in 6SigmaET. In the experimentation, front end of server was inserted in testing chamber. The airflow rate was kept constant during experimentation and static pressure across server was measured. A graph of static pressure vs airflow rate was plottedThe graph obtained from experimentation is as follows: -

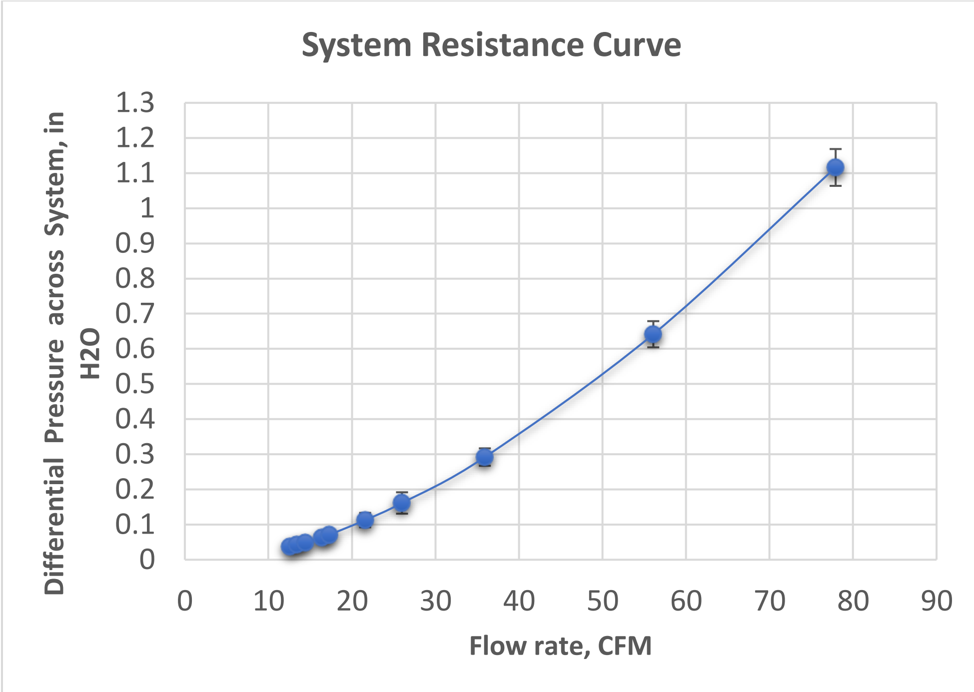

Figure 30:- Experimental System Resistance Curve of Server model.

The graph for server model was obtained by simulating the boundary conditions set like experimental condition. The flow rate and environmental condition where given as input parameters to obtain the static pressure. The prescribed flow of air was like experimental setup as shown in following figure.

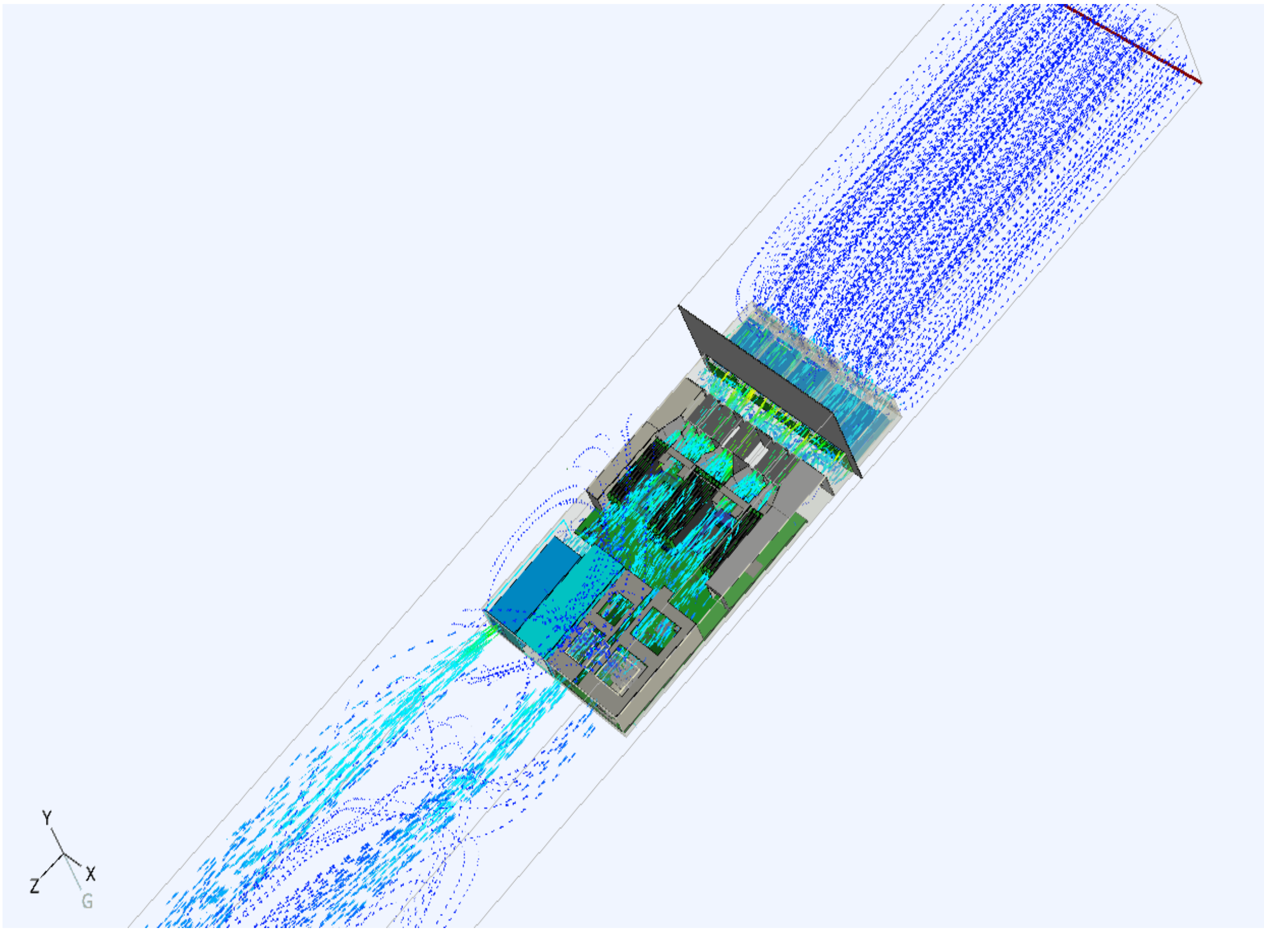

Figure 31:- Airflow Pattern in Server Model.

In 6SigmaET, pressure was measured through pressure sensor used in test chamber to obtain the static pressure, similar, to experimental setup where we used transducers to obtain static pressure. The graph plotted from simulations compared with graph of experimentation looks as follows. The deviation of both graphs is slight to be accounted for. This shows accuracy of model with respect to system resistance to airflow. The overall percentage variation is less than 10% numerically.

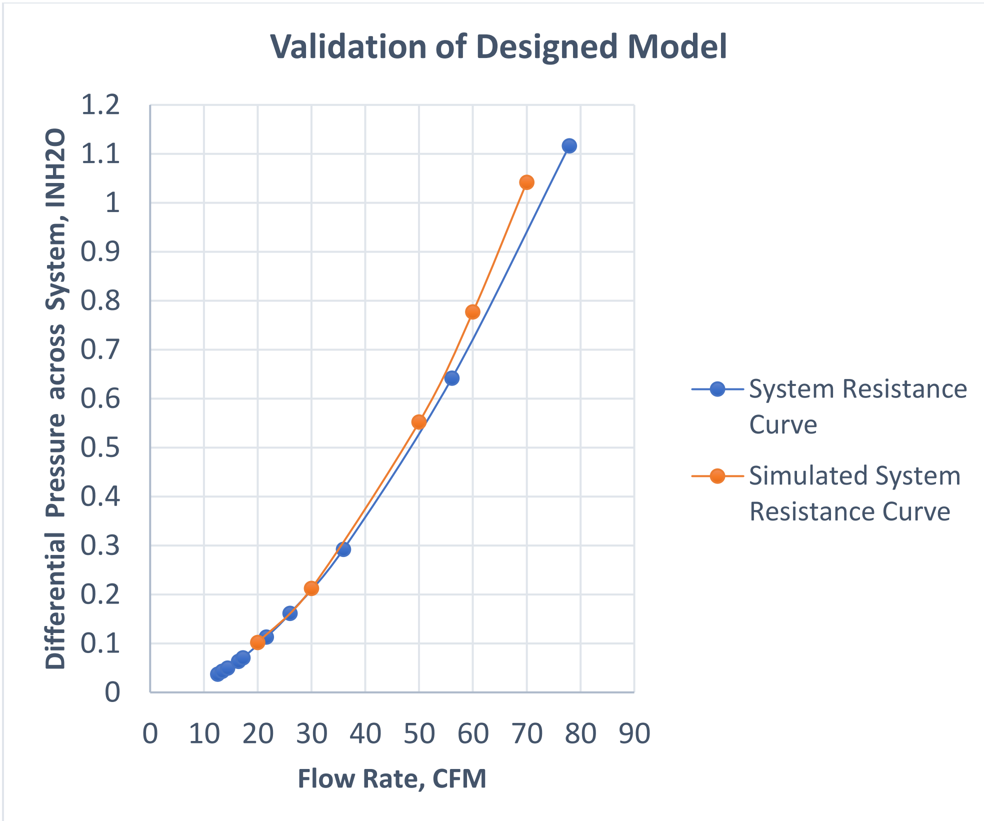

Figure 32:- Experimental and Simulated System Resistance curve comparison.

7.2 Mesh Sensitivity analysis

In order to achieve accurate results, the grid or mesh applied to the model needs to be refined. The measure of refining the mesh is known as mesh sensitivity analysis. In CFD analysis, accurate can be obtained with help mesh sensitivity analysis. It is trial and error method which can be easily done with help of PAC study. In order to achieve the best mesh for modelled design we change the grid parameter to obtain constant parameters in complete simulation process. The simulated analysis gives us the following graph which shows that the solution control of 30 Million target best suits our model.

Solution Control			Pressure Drop
Max Size of X, Y, Z	**Target Grid Count**	**Actual Grid Count**	
20	10,000,000	10,842,068	0.5383
20	20,000,000	17,417,094	0.5449
15	30,000,000	19,959,747	0.5128
12	40,000,000	24,446,004	0.5189

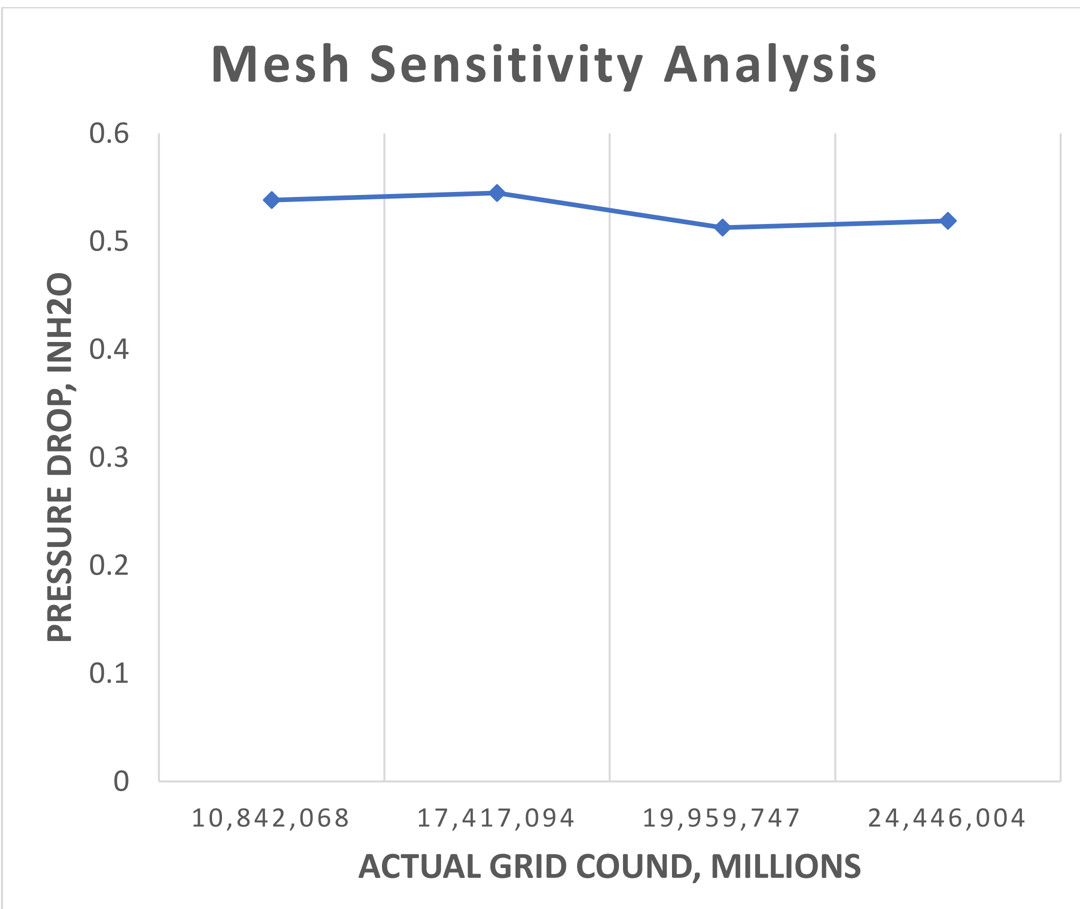

Figure 33:- Mesh Sensitivity of Server Model.

7.3 Results of Air-Cooled Server

The air-cooled server simulation is done on 100% utilization and different inlet temperatures. The different inlet temperature where selected based on safe working environment standards for server provided by ASHRAE T.C. 9.9. The fan rated speed followed the graph taken from previous studies. The component temperatures of server at different boundary conditions can be shown in the graph below. The fan rated speed as mentioned varies. In this case, the fan RPMs are kept high to keep the temperature of component constant. As fan RPMs ramps up in the server to reduce the temperature of component.

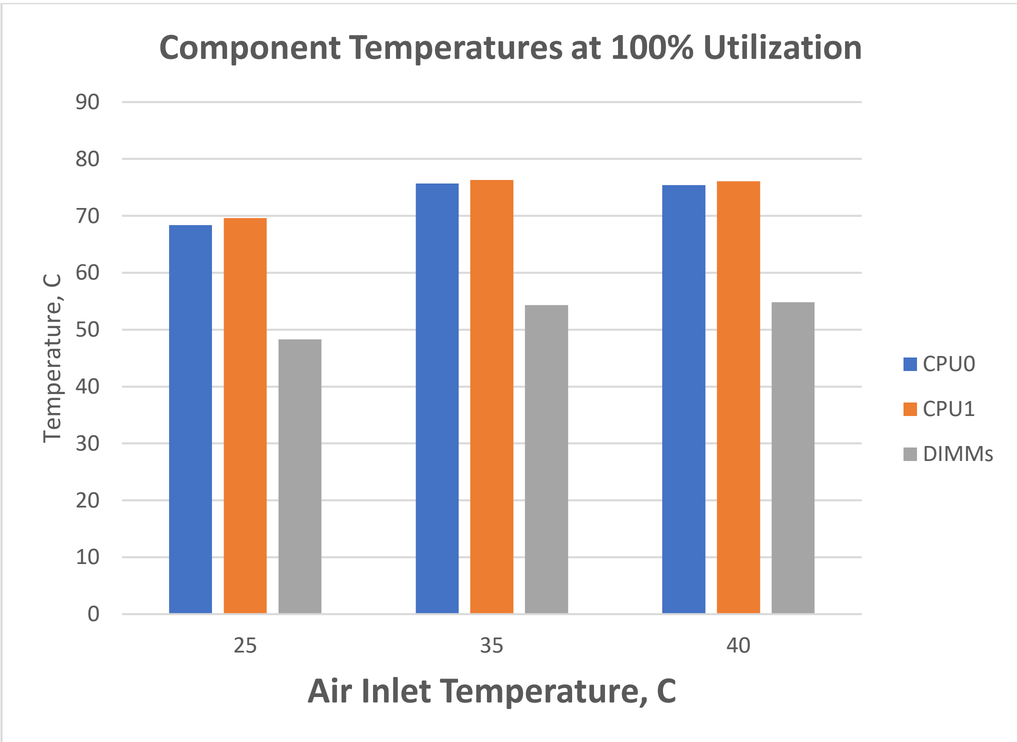

Figure 34:- Component Temperatures of Server.

Along with component temperature of server at different inlet temperature we obtained flow rate ratio curve for air cooled cisco server that will be used in room model to compare air-cooled and hybrid cooled data center. The flow rate ratio curve can be shown as follows.

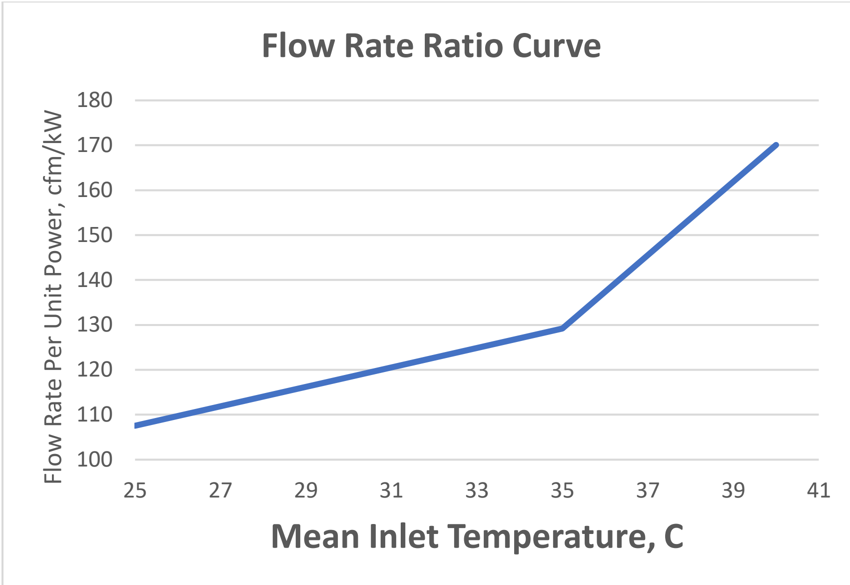

Figure 35:- Air-Cooled Server Flow Rate Ratio Curve.

Chapter 8:- Room CFD Modelling

A data center model is prepared for comparing the air-cooled and hybrid cooled data center. The room model is prepared in 6SigmaRoom software developed by future facilities. The air-cooled model will import data from the simulation obtained from 6SigmaET model and for hybrid cooled data center, data is considered from previous studies. The room is simulated for different containment – cold and hot aisle containment. The room has 6 cabinets (21 kW) and 2 ACU unit (80kW). The server specification are as follows. The slotted grille selected is 0.6m, the dimension selected is according to best practices.[30] The width of the hot is 1.2m. The cooling power capacity of ACU units is 80kW. The server are set at 100% utilization.

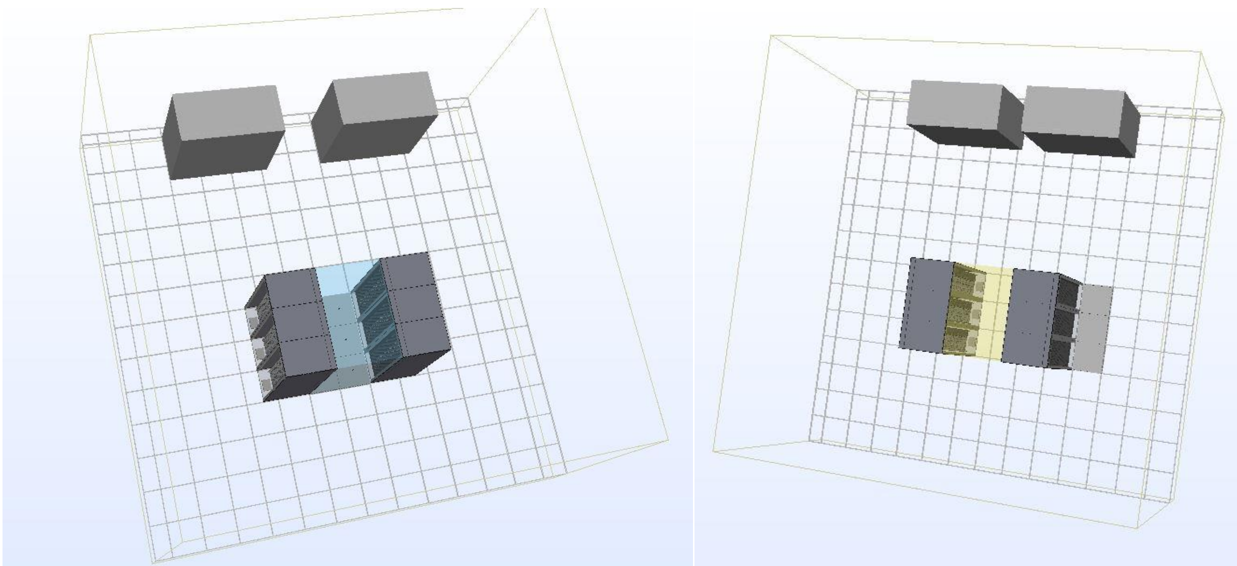

Figure 36:- Room CFD Model.

Cabinet	Servers
Power Limit: - 21 kW	Name Plate Power: - 450W
Capacity: - 42 1U Servers	Type: - Hybrid and Air Cooled
Status: - Filled	Mounting: - Simple Mounting Rails.
Doors: - Perforated Front & Back Doors	

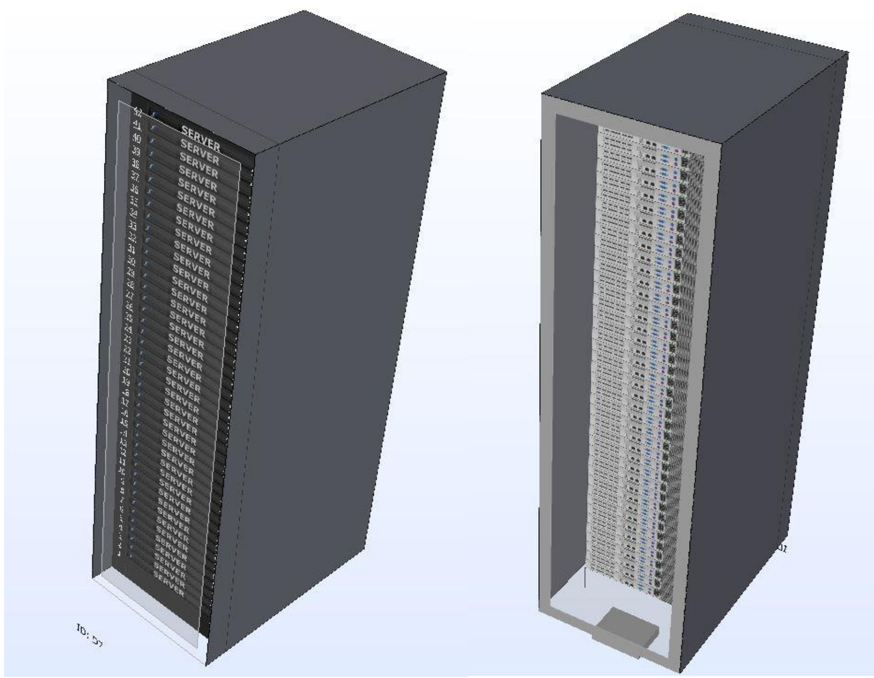

Figure 37:- Cabinet or Rack Model in Room Model.

Chapter 9:- Results

9.1 Room Model Results

The data center model for air and hybrid cooled systems is created with the specification discussed earlier. The flow rate ratio curve of air-cooled server and hybrid cooled server is used for Racks filled with the servers. The flow rate ratio curve for air-cooled server is procured as discussed, while the flow rate ratio curve for hybrid cooled server is obtained from previous studies.

The results we got from air-cooled server analysis and previous study the flow rate ratio curve can be used in data center room model. In data center room model, racks are number according and the simulation data is received in consonance of refence given by software. The reference data center model is shown below.

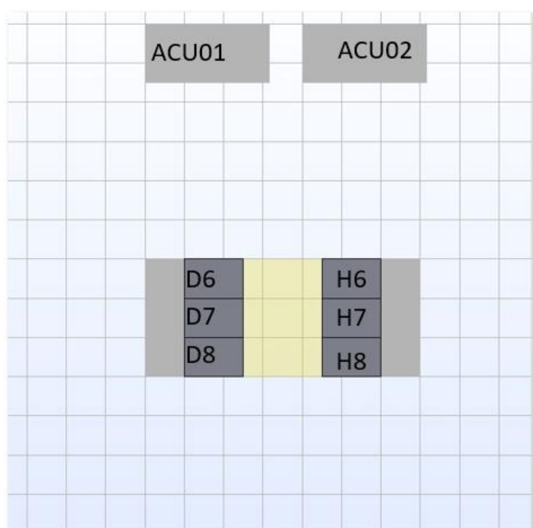

Figure 38:- Reference Naming of Room Model.

In air-cooled data center, servers are 100% cooled by consolidation of air from ACU units and server fans. In hybrid cooled data center, servers are cooled by water flowing from cold plates and air from the server fans and ACU units. The liquid cooling percentage in hybrid-cooled server can be specified in 6SigmaRoom. As mentioned, the percentage of cooling considered in this work is 42.2%, 55.56% and 64%. The percentage are selected based on best practices.[11]

The comparison of air-cooled and hybrid-cooled data center was done by providing different set of mean inlet temperatures from ACU units for 2 different configuration of air containment – Hot-Aisle Containment and Cold-aisle Containment. The hybrid cooled data center is analyzed with different cooling percentages as discussed earlier. The graphs show the clear picture of comparison. The room model is considered as black box model as different power and flow rate are given at different temperature depending on findings from 6SigmaET model.

1.) Cold-Aisle Containment Results

The graph shows the inlet temperature vs cabinet outlet temperature for different percentage of liquid cooling. The graph shows the temperature of cabinet remain constant for each type of liquid cooling. The temperature of the cabinets is less for higher liquid cooling percentage.

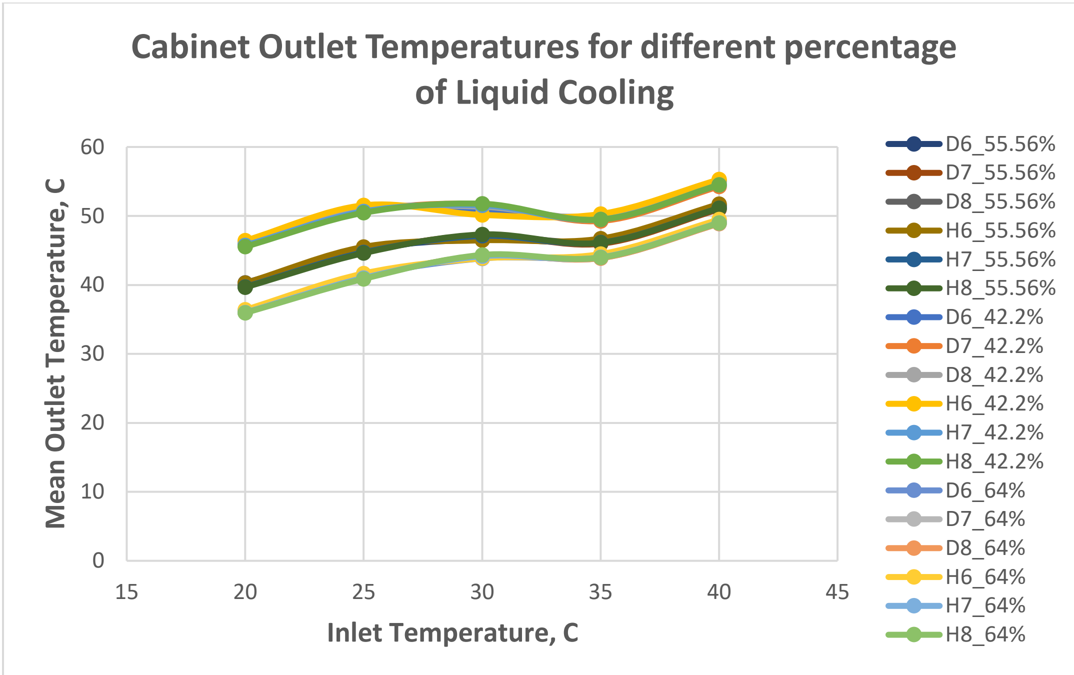

Figure 39:- Cabinet Temperature of Hybrid Cooled Room Model at different liquid cooling percentage.

The graph below shows the graph for air cooled data center for inlet air temperature vs cabinet outlet temperature. The temperature of the cabinet near to ACU is the least.

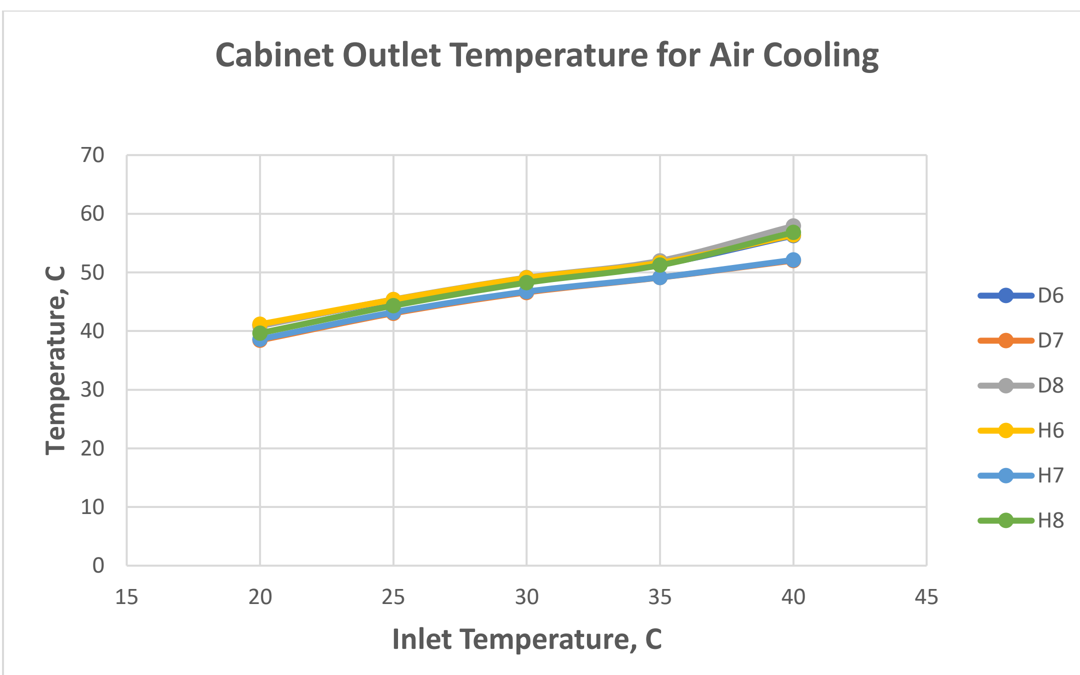

Figure 40 :- Cabinet Temperature of Air-Cooled Room Model.

The following graphs shoes the power utilization of room model or data center under air and hybrid cooling with different liquid cooling percentages. The simulation is done for cold-aisle containment.

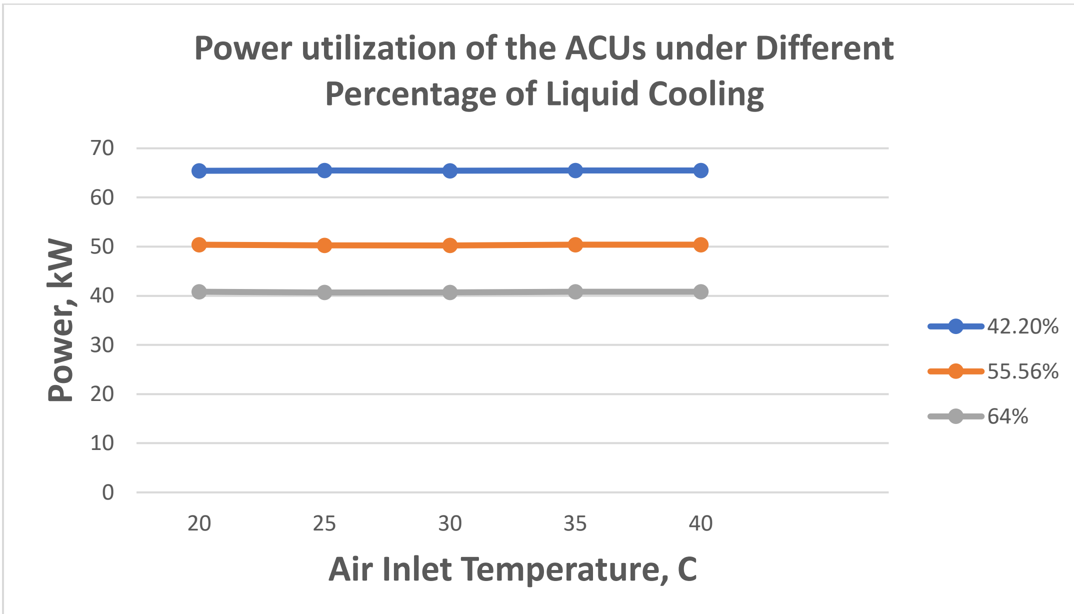

Figure 41 :- Power utilization of the ACUs under Different Percentage of Liquid Cooling

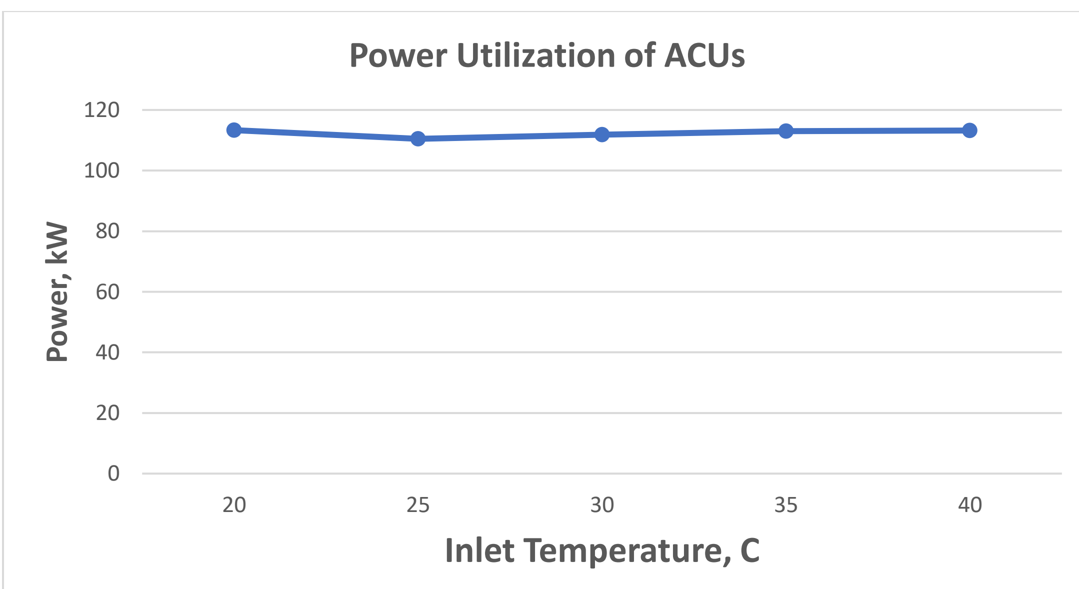

Figure 42 :- Power Utilization of ACUs for Air Cooling

2.) Hot-Aisle Containment Results

The graphs below show that cabinet temperature is least for the liquid of cooling percentage the maximum. We can see that the liquid cooling gives better cooling of the IT equipment in the data center. Hence, the IT equipment reliability is increases when liquid cooling is used since it keeps it cool.

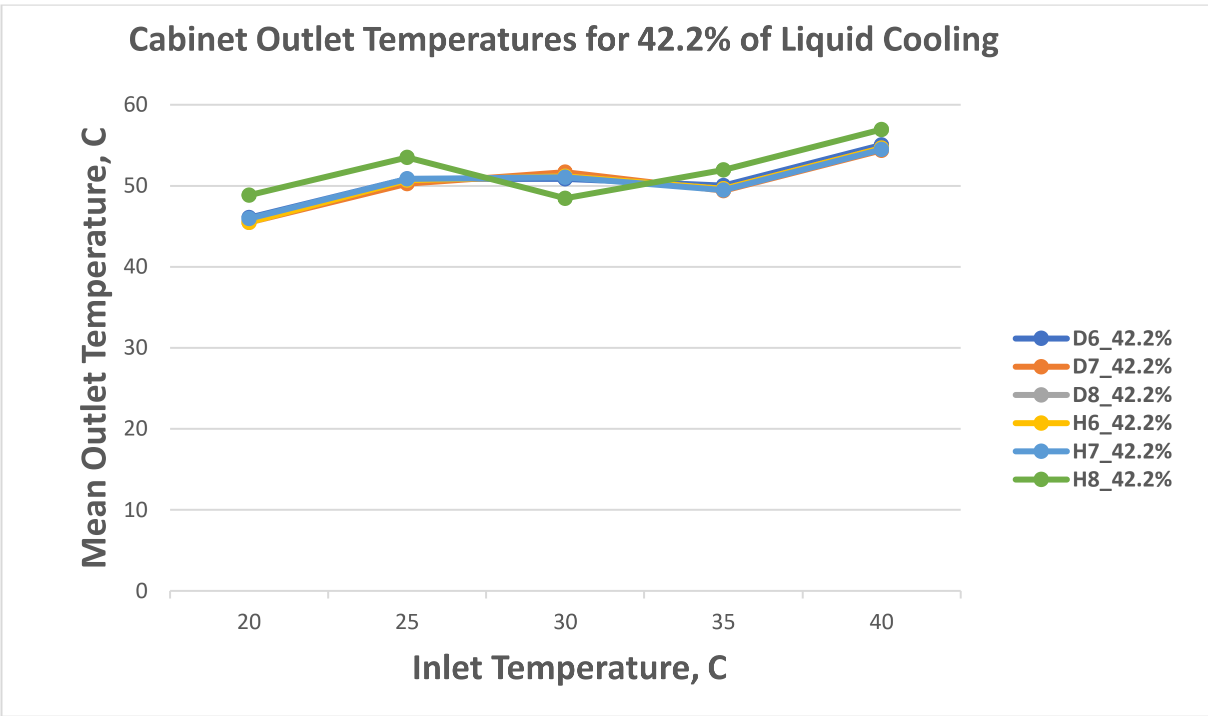

Figure 43:- Cabinet Temperature of Hybrid Cooled Room Model at 42.2% liquid cooling percentage.

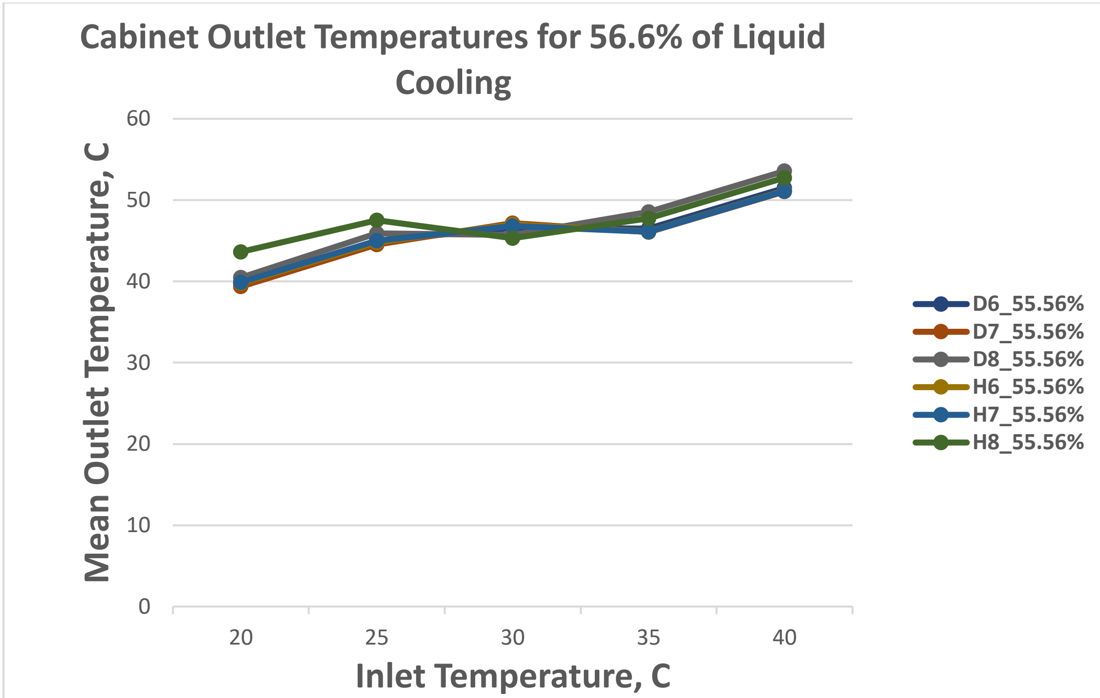

Figure 44:- Cabinet Temperature of Hybrid Cooled Room model at 56.6% liquid cooling percentage.

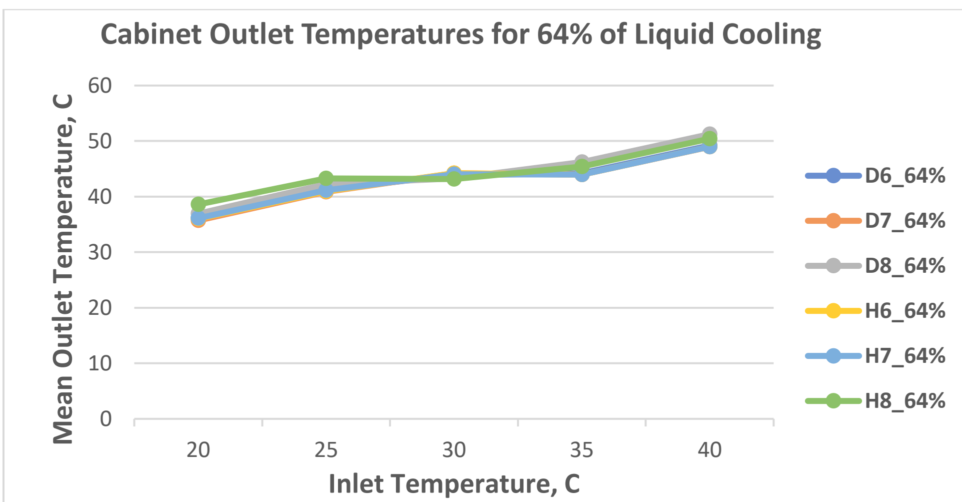

Figure 45:- Cabinet Temperature of Hybrid Cooled Room Model at 64% liquid cooling percentage.

The cabinet outlet temperatures are shown for hot-aisle containment for air-cooling in the graph below. The temperature of cabinet is more for racks that are away from ACU and vice-versa. The temperature of the racks for air-cooling data center is higher compared to that in case of hybrid-cooled data center.

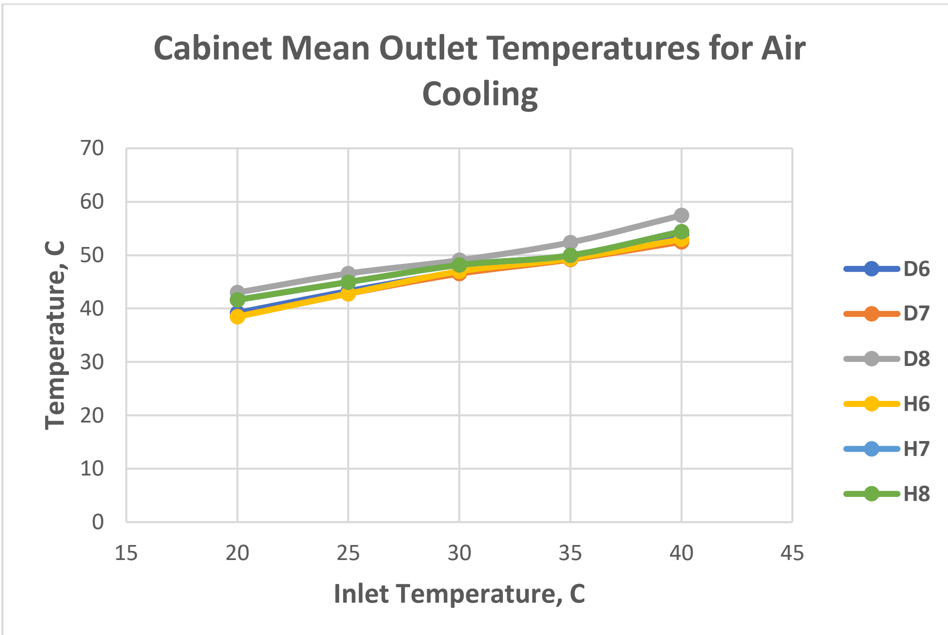

Figure 46:- Cabinet Temperature of Air-Cooled Room model.

The following graphs shoes the power utilization of room model or data center under air and hybrid cooling with different liquid cooling percentages. The simulation is done for hot-aisle containment.

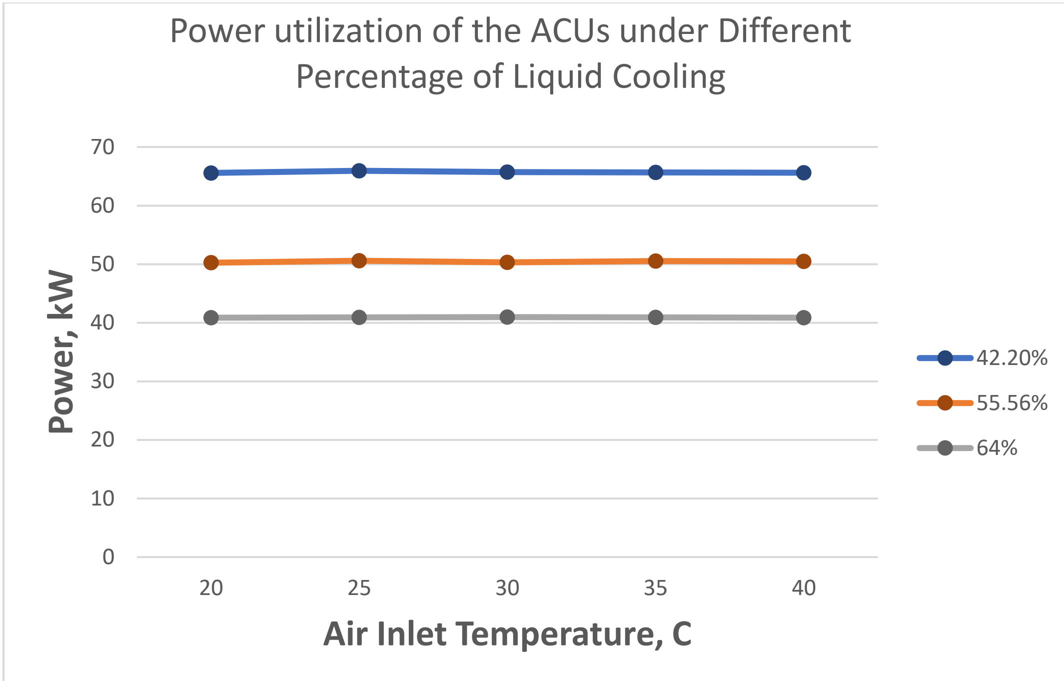

Figure 47:- Power Utilization of ACUs in Hybrid Cooled Room Model for Different liquid cooling percentage.

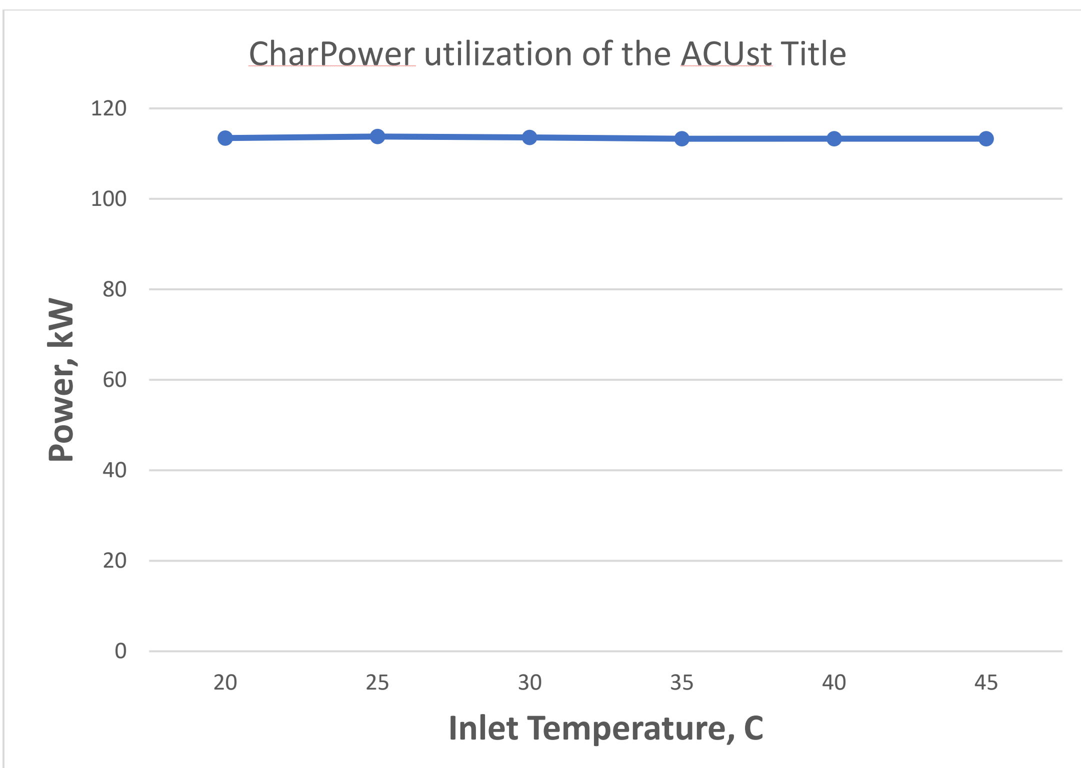

Figure 48:- Power Utilization of ACUs for Air Cooled Room Model

Chapter 10:- Conclusion

In this study, Air-Cooled Server was analyzed to obtain the results for comparison of Air to Hybrid cooled data center. The air-cooled server was analyzed at higher fan to maintain temperature of server components and achieve data to be used in data center model for Air-Cooled Server. The data for hybrid cooled server was used from previous study. The results show that with raised inlet temperature in air cooling, the power drawn by the server increases, the component temperature increases, and the reliability of IT equipment decreases. One of the reasons for these changes is leakage current which is in direct relation with chip operating temperature.

In case of hybrid cooled server with increase inlet temperature delta T achieved is within the ASHRAE guidelines for classes A1 and A2 classes for IT equipment zone. In case of hybrid cooling we can also use free cooling to increase hours of economizer. Comparing power drawn by the air to hybrid cooled data center it shows the hybrid cooled data centers with higher percentage of liquid cooling draws less power compared to Air Cooled data Center. The results also show that different percent of cooling can be used to improve results for data center cooling.

Reference

[1] H. Geng, *Data Center Handbook*. 2015.

[2] P. Sawers, "Microsoft beats Google and Amazon to announce first African data centers."

[3] A. Shehabi *et al.*, "United States Data Center Energy Usage Report," *Lawrence Berkeley Natl. Lab. Berkeley, CA, Tech. Rep.*, 2016.

[4] M. Nakamura and R. Nakamura, "Energy efficient cooling in data center using optimization approach," *SICE Annu. Conf. 2012*, pp. 2103–2108, 2012.

[5] T. A. Faculty, N. Rolander, and I. P. Fulfillment, "AN APPROACH FOR THE ROBUST DESIGN OF AIR COOLED DATA CENTER SERVER CABINETS by AN APPROACH FOR THE ROBUST DESIGN OF AIR COOLED," no. December, 2005.

[6] V. Avelar, D. Azevedo, and A. French, "PUE^{TM}: A Comprehensive Examination of the Metric," *GreenGrid*, pp. 1–83, 2012.

[7] ASHRAE, "Data Center Power Equipment Thermal Guidelines and Best Practices," pp. 1–60, 2016.

[8] ASHRAE Technical Committee 9.9, "Thermal Guidelines for Data Processing Environments," *Am. Soc. Heating, Refrig. Air-Conditioning Eng. Inc*, pp. 1–45, 2011.

[9] M. Sahini *et al.*, "Comparative study of high ambient inlet temperature effects on the performance of air vs. liquid cooled IT equipment," *Proc. 16th Intersoc. Conf. Therm. Thermomechanical Phenom. Electron. Syst. ITherm 2017*, vol. 7, pp. 544–550, 2017.

[10] U. Chowdhury, M. Sahini, A. Siddarth, and D. Agonafer, "RAISING INLET AIR TEMPERATURE FOR HYBRID-COOLED SERVER RETROFITTED WITH LIQUID COOLED COOLED COLD PLATES," pp. 1–6, 2018.

[11] R. Zeighami, W. A. Saunders, H. Coles, and S. Branton, "Thermal performance modeling of hybrid liquid-air cooled servers," *Thermomechanical Phenom. Electron. Syst. -Proceedings Intersoc. Conf.*, pp. 583–587, 2014.

[12] T. J. Breen *et al.*, "From Chip to Cooling Tower Data Center Modeling: Chip Leakage Power and

Its Impact on Cooling Infrastructure Energy Efficiency," *J. Electron. Packag.*, 2012.

[13] S. A. Hall and G. V. Kopcsay, "Energy-Efficient Cooling of Liquid-Cooled Electronics Having Temperature-Dependent Leakage," *J. Therm. Sci. Eng. Appl.*, 2013.

[14] S. Sheet, "Cisco UCS C240 M3 Rack Server (Small Form Factor Disk Drive Model)," 2014.

[15] "Cisco."

[16] E. Design and S. Eds, "Intel ® C600 Series Chipset and Intel ® X79 Express Chipset," no. May, pp. 1–1018, 2012.

[17] Seagate, "Savvio 10K.5 ®," pp. 2011–2012.

[18] "Cisco C220 M3 Server Motherboard."

[19] G. P. Battery, "Specification for Approval," no. Dc, pp. 1–14, 2012.

[20] Airflow Measurement Systems, "Air Flow Bench Specification."

[21] A. Bakker, "Lecture 1 - Introduction to CFD Applied Computational Fluid Dynamics," *Fluid Dyn.*, no. 2002, pp. 1–36, 2006.

[22] J. Anderson, "Computational Fluid Dynamics: The Basics with Applications. 1995," *McGrawhill Inc*, 1995.

[23] V. Pandian, "Development of Detailed Computational Flow Model of High End Server," no. December, 2012.

[24] F. Moukalled, L. Mangani, and M. Darwish, "Erratum to: The Finite Volume Method in Computational Fluid Dynamics," 2016.

[25] A. Bakker, "Lecture 5 - Solution methods Applied Computattional Fluid Dynamics," *Text. Res. J.*, vol. 105, no. 2, pp. 52–59, 2014.

[26] H. Versteeg and W. Malalasekera, "An Introduction to Computatational Fluid Dynamics(2nd Edition)."

[27] "Laminar vs Turbulent Flow."

[28] K. Dhinsa, C. Bailey, and K. Pericleous, "Low Reynolds number turbulence models for accurate thermal simulations of electronic components," no. February, pp. 483–490, 2004.

[29] San Jose State University, “Fan Configuration and Airflow Impedance Lab Manual Part 1 ME146.” .

[30] K. C. Karki and S. V. Patankar, “Airflow distribution through perforated tiles in raised-floor data centers,” *Build. Environ.*, vol. 41, no. 6, pp. 734–744, 2006.