OPTIMIZATION OF AIR COOLED PORTION IN A SERVER WITH WATER COOLED MICRO-PROCESSORS

By

NEHA INAMDAR

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 guidance, patience and support during the course of this research work. His invaluable advice and encouragement enabled me to complete my thesis in timely manner. The opportunity provided to me was a great learning experience. I would also like to thank Dr. Haji-Sheikh and Dr. Miguel Amaya for serving on my thesis committee.

I would also like to thank Ms. Sally Thompson, Ms. Debi Barton and Ms. Flora Pinegar for helping me in educational matters. They have always supported for the last two years at the university. I am obliged to Dr. Seiichi Nomura for his help.

I extend my thanks to all my friends for the moral support. Lastly, I am grateful to my family for supporting me emotionally and financially. I would not have completed my degree without their support and good wishes.

April 18, 2016

Abstract

OPTIMIZATION OF AIR COOLED PORTION IN A SERVER WITH WATER COOLED MICRO-PROCESSORS

Neha Inamdar, MS

The University of Texas at Arlington, 2016

Supervising Professor: Dereje Agonafer

A Data center is the heart of our associated world. It is a facility that houses computer systems and related components. According to IDC, because of enormous increment in pattern of IoT (Internet of Things) and distributed computing the introduced limit of server farm workloads will increment by around 750% around 2014 and 2019. Data center spaces can consume up to 100 to 200 times as much power as standard office spaces. With such huge influence utilization, they are prime focuses for vitality proficient outline measures that can spare cash and lessen power use. In a regular Data center with an exceptionally effective cooling framework, 50% of the data energy is consumed by the IT equipment. Utilization of effective IT hardware will altogether lessen these heaps inside the Data center, which subsequently will scale back the gear expected to cool them. Introduction to air-water cooled system will help us achieve more effective cooling of IT equipment while reducing the power consumption. Air-water hybrid gives us the

flexible design choices. This hybrid system cools the high heat generating components using water and the rest of the components are cooled using the internal fans. Air cooling is the conventional way of cooling for IT equipment and it suffice for cooling of components with low thermal demand. In addition, air cooling is cheap, easily available and has high serviceability than rest of the cooling methods.

The objective of this work is to optimize the airflow through one such air-water cooled server in order to achieve effective cooling power. Since, the high heat generating components are cooled by water, the other components will need less volume of air for cooling. Also the inlet air can be at much higher temperature to achieve cooling following ASHRAE standards. In the following work, we develop a detailed server model using software 6SigmaET® equivalent to the reference server to study the air cooling part and optimize the airflow. The supply airflow rate is simulated by varying the speed of fans to see the volume of airflow in the system. It is then seen for less number of fans or changing number of fans to study the airflow in the system. Thus, the efficiency of the cooling power due to air cooling is studied.

Table of Contents

List of Illustrations

List of Tables

CHAPTER 1

INTRODUCTION

A Data center is the heart of our associated world. It is a facility that houses computer systems and related components. According to IDC, because of enormous increment in pattern of IoT (Internet of Things) and distributed computing the introduced limit of server farm workloads will increment by around 750% around 2014 and 2019. [1] Data center spaces can consume up to 100 to 200 times as much power as standard office spaces. With such huge influence utilization, they are prime focuses for vitality proficient outline measures that can spare cash and lessen power use. [2] In a regular Data center with an exceptionally effective cooling framework, IT gear burdens can represent over portion of the whole office's vitality use. Utilization of effective IT hardware will altogether lessen these heaps inside the Data center, which subsequently will scale back the gear expected to cool them. Acquiring servers outfitted with vitality proficient processors, fans, and power supplies, high-productive system hardware, merging stockpiling gadgets, uniting power supplies, and actualizing virtualization are the most favourable approaches to lessen IT gear loads inside a server farm.

Rack servers have a tendency to be the fundamental culprits of wasting energy and these also form to be the biggest segment of the IT energy load in a typical data center. Servers take up the majority of the space and drive the whole operation. The major share of servers keep running at or underneath 20% use more often than not, yet still draw full power amid the procedure. As of late endless upgrades in the interior cooling frameworks and processor gadgets of servers have been made to minimize this energy wastage.

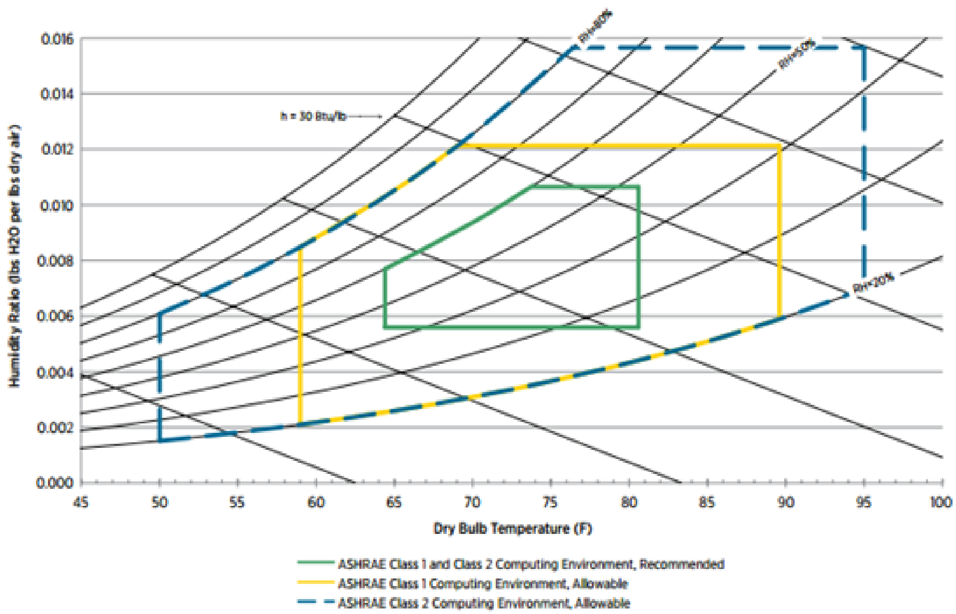

Figure 1 2009 ASHRAE environmental envelope for IT equipment air intake conditions [2]

1.1Types of server chassis

The servers these days are intended to be proficient, quick and minimal too. Reducing the server foot print lessens data center space. Servers are grouped in light of their application. Stage servers, Application servers, mail servers, proxy servers, web servers and communication servers are few of the sorts. The servers are likewise grouped regarding the chassis as Blade servers and Rack Mount servers. The blade servers will be servers with secluded configuration upgraded to minimize the utilization of physical space and power. [3] These servers are set in an enclosure which can hold different servers and give other non-processing related operations like cooling, power and systems administration. [3] Blade servers are cost-proficient and thin, housed inside a chassis, which is additionally called a cabinet. The blade servers inside the same chassis are associated utilizing a transport wiring framework. The rack mount servers are contained in cases measuring standard sizes termed as 1U servers (1U=1.75”), which are mounted in a rack inside a cabinet. [3] The rack servers a have individual cooling and can operate exclusively. Other advantage of using rack servers is the ability of using servers from various manufactures.

1.2 Why cooling is necessary

Server downtime is a critical term utilized as a part of data center industry. Server downtime is the time amid which the server is unusable. Server downtime might be brought on because of different reasons like memory overload, overloaded processors and so forth. All the specified reasons would affect the cooling required by the server. The organization's data center is regularly extremely basic and must be very much secured. The separate chips are the fundamental wellspring of heat era and it is essential to keep the chips inside the operating temperature range. Servers generate a lot of heat in generally little foot print. The force utilized by server is dissipated as heat into the surrounding. The force of a processor can to a great extent rely on upon the measure of heat evacuated. The heat dissipated is to a great extent non-uniform and vacillates much of the time. Non uniformity in force might be brought about because of different reasons. For instance, heat load variety may happen with time because of expansion of workload or because of the blend of equipment. Data center suits segments touchy to heat and mugginess. Expansive number of disappointments has found to happen at chip level. As we probably am aware, the server execution is relative to the temperature at which the chip works and it likewise influences the timeframe of realistic usability of the chip extensively.

The air flow through server chassis is key perspective influencing the cooling productivity of a server as the fundamental method of heat move happening in a server is through convection and conduction. Conduction happens at chip level and convection happens at a cabinet level which is of an alternate scale when contrasted with chip level. Parts like heat sinks, fans are utilized to upgrade the procedure of convection. Interchange cooling arrangements like heat channels, vapour chamber heat sinks might be used be that as it may, its proficiency relies on upon the accessibility of wind current through the server. While outlining consideration ought to be taken to guarantee appropriate course through the server is accessible to fulfil the changing heat load. Consideration ought to be taken while giving vents to help the server cooling process. [3]

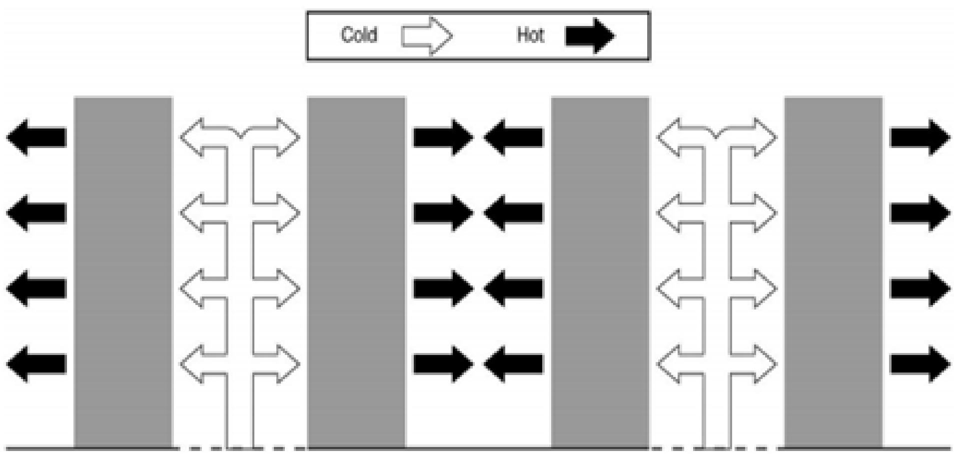

Figure 2 Hot Aisle and Cold Aisle layout in a data center [4]

1.3 *Why CFD modelling*

As examined in the past area, one can comprehend the significance of air flow through the server. The variable influencing the flow through the server is pressure drop. To push air from a point to another, a potential difference is required. This potential difference is the pressure drop. It is proper to decide the pressure drop required to drive the required volume of air through the server. A CFD model helps us deciding these parameters in this way giving us an expectation about the ability of the configuration. These expectations might be resolved tentatively. Be that as it may, trial testing devours time and is costly. This hole is crossed over by broad utilization of CFD software which gives us the adaptability and exactness in basically simulating a test setup. Enhancement of servers requires tries different things with different mixes, introductions of segments and enhances the execution envelope. It is simpler to think about the impacts of the progressions for all intents and purposes than by trial testing. The center of this study is to build up a definite flow model of the server, which can be utilized for thermal analysis furthermore might be utilized to build up a smaller/rearranged model for leading learns at rack and room level.

CHAPTER 2

SERVER DESCRIPTION

To determine the pressure losses and the volumetric air flow inside the system, we have generated the CFD model. CFD model may not give the exact same results as the experimental but it saves time and helps estimate the nature of the system. Hence we try to develop a detailed model as close to the actual model as possible representing its key physical features for our study. The component specifications are approximately matched in the CFD model to the actual specifications.

The server under consideration is enterprise-class UCS C220 M3 server with 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, 3 drives and 2 x 1 GbE LAN-on-motherboard (LOM) ports. [5] The micro-processors are mounted with the cold plates from Asetek. In this setup, microprocessors conduct heat from the die through thermal interface materials (TIMs) to liquid cooled cold-plates, to enhance the heat transfer by replacing standard air cooled heat sinks

2.1 Motherboard

The server under consideration consists of a one rack unit chassis which is populated by dual Intel Xeon® E5-2600 v2 or E5-2600 series processor family CPUs with 16 Dual-In-Line-Memory Module (DIMM) slots. These CPUs have the Thermal Design Power (TDP) of about 135 W. It constitutes of 4 memory channels per CPU and up to 2 DIMMs per channel. The motherboard also consists of an Intel C600 series chipset. The motherboard houses Small Form Factor (SFF) drives. Up to eight 2.5-inch SAS or SATA hot-swappable hard disk drives (HDDs) or solid state drives (SSDs) [5]

Figure 3 Server under consideration – Cisco UCS C220 M3

2.2 Chassis

The server under consideration has a chassis of 1U rack (1.6" height). The chassis body is made of aluminium and has 5 fans mounted on the rear side.

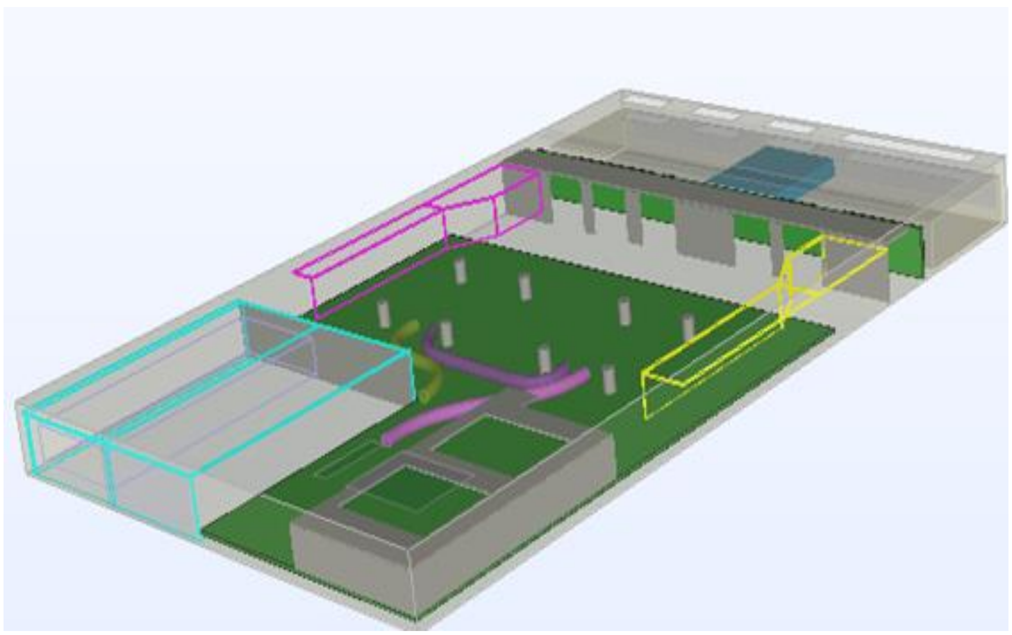

Figure 4 Graphical representation of the server chassis

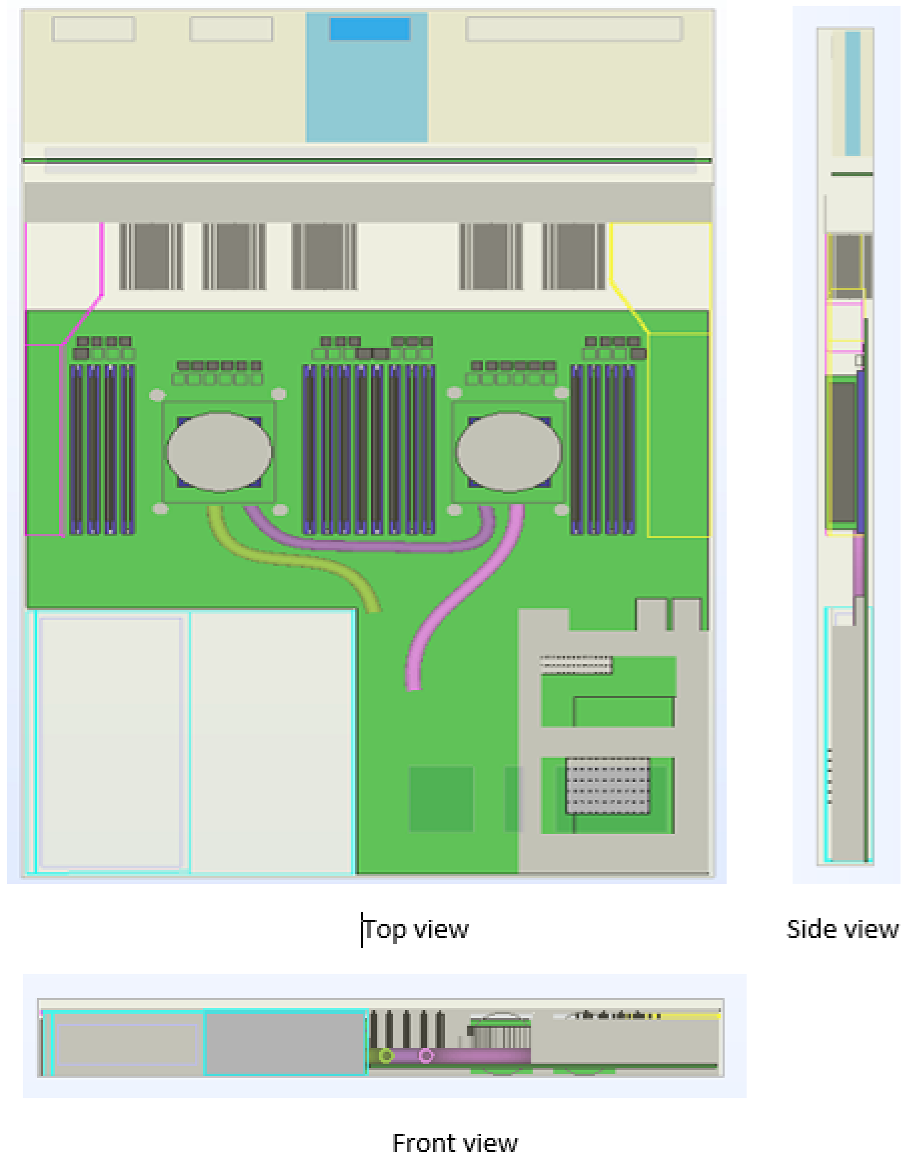

Figure 5 Graphical representation of the server top, side and front view

2.3 *Server Fans*

The server under consideration is a Hybrid-cooling server, i.e. the main components CPUs are water cooled using cold-plates, but the other heat generating components like PCH, DIMMs need cooling as well. These other components are hence cooled with the help of Delta fans (40X40X56 mm by dimension) mounted on the rear side of the server.

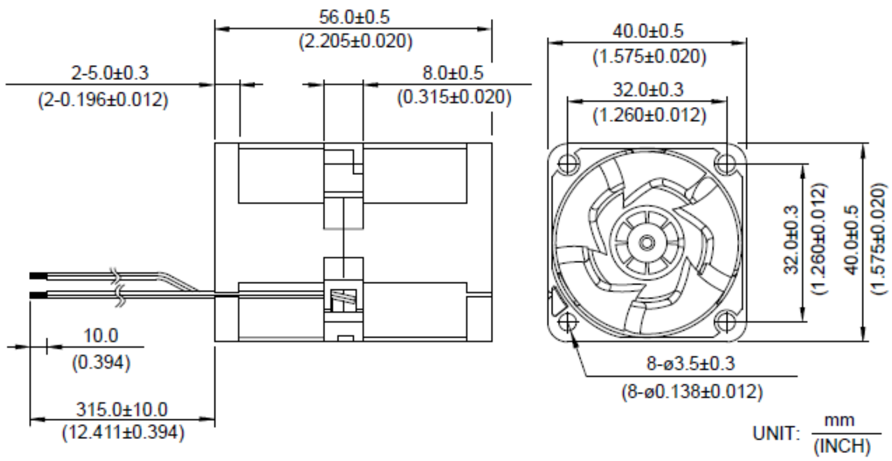

Figure 6 Fan dimensions [6]

The fans used are DC brushless axial flow fans with four poles and single phase motor. These have maximum airflow of 32.179 CFM at zero static pressure and the maximum air pressure produced is 2.631 inch of H2O at zero airflow. The rated speed for these fans is 16000 rpm. [7]

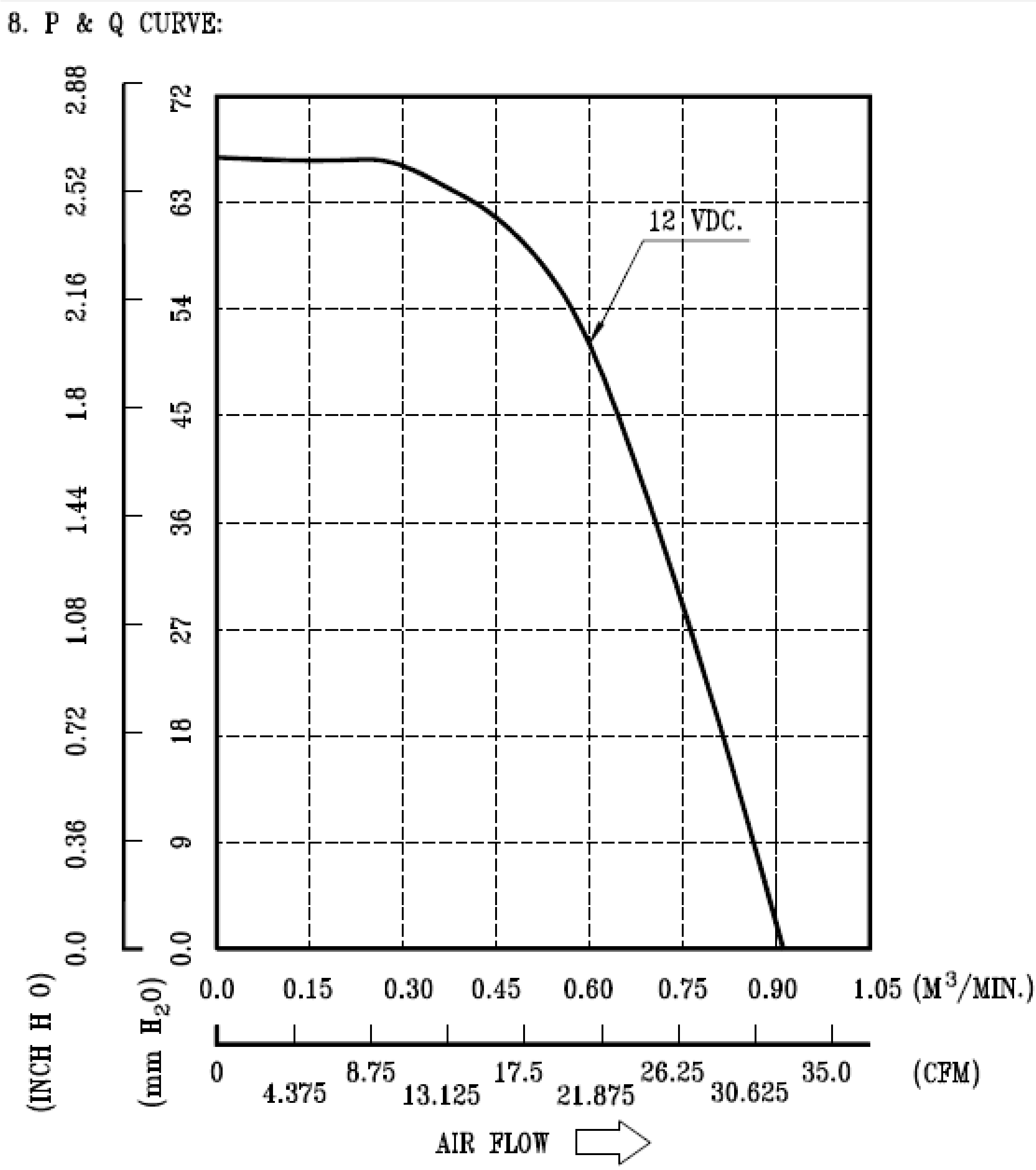

Figure 7 Fan performance graph from manufactures data [7]

CHAPTER3

CFD (COMPUTATIONAL FLUID DYNAMICS) ANALYSIS

3.1 *Introduction to CFD*

CFD (Computational fluid dynamics) is a branch of fluid mechanics which manages the numerical simulation and analysis of fluid flow, heat exchange qualities and pressure attributes. Computational fluid dynamics utilizes numerical techniques to foresee, recreate and examine velocity, pressure, temperature and different variables all through the domain. The results can be used to analyze the design and helps to optimize the design process and products [9]. Computers are utilized to perform the calculations required to simulate the fluid interactions with surfaces taking into account the boundary conditions. CFD analysis was basically utilized as a part of aircraft and automobile industry. Be that as it may, it is currently broadly utilized for different applications, for example, data center commercial ventures, systems with high heat loads, telecom industry, and a few more.

CFD results are analogous to wind tunnel results acquired in laboratory –they both represent sets of data for given flow arrangements at various Reynolds numbers [3]. CFD goes about as a connection between unadulterated hypothesis and immaculate test by helping us plainly comprehend the aftereffects of hypothesis and investigation. CFD is likewise a design apparatus. The upside of utilizing these numerical strategies is that the issue can be discretized in view of an arrangement of numerical parameters and comprehended. It is less costly when contrasted with leading an analysis for each change we make to optimize the design. At the point when contrasted with directing an examination, CFD is quick as we can reenact numerous cases in particular time period. The simulation devices offer a storehouse of components that can be utilized, for example, grid generation, mesh sensitivity analysis and a few different elements. A numerical forecast is utilized for the generation of a mathematical model which speaks to the physical space important to be comprehended and examined. In this specific case, the study includes the framework level gear, for example, the server chassis, the hardware and other gear like the force supply unit, hard plate drive and so forth that are housed in them.

With the expanding use of CFD in different fields, requirement for fast processing has likewise expanded quickly. Prior serial machines were utilized for rapid figuring, yet now distinctive PC designs are being utilized to speed up the calculation process. The two new designs fundamentally utilized are vector preparing and parallel handling [3].

3.2 *Governing Equations*

Computational fluid element codes depend on Navier-Stokes equations. The numerical answer for heat exchange and fluid flow based issues is acquired by comprehending a progression of three differential equations. These three differential equations are the protection of mass, preservation of force and protection of vitality. They are usually known as the governing differential equations [3].

For a generalized case, the conservation of mass is given by:

$$\frac{\partial \rho}{\partial x} + \nabla.(\rho u) = 0$$

The conservation of momentum for a generalized case is given by:

$$\frac{\partial}{\partial t}(\rho u) + \nabla.(\rho u u) = \nabla.(\mu gradu) - \frac{\partial p}{\partial x} + B_x + V_X$$

The conservation of energy for a steady low velocity flow is given by:

$$\nabla.(\rho u h) = \nabla.(kgradT) + S_h$$

3.3 *Global Computational Domain*

In a general flow field, we consider a closed volume inside a limited locale of flow. This closed volume is characterized as a controlled volume. The computational area or the arrangement space is the locale or space inside the closed volume in which the governing differential equations are understood. The control volume might be settled in space or might be moving with the fluid [19]. The answers for these equations are acquired by settling the boundary conditions for the arrangement area. The boundary conditions for most computational issues incorporate the outer ambient temperature, velocity, pressure, mass flow at inlet and outlet, fluid thickness, thermal conductivity, particular heat and other environmental conditions.

It relies on upon the sort of heat exchange, for example, conduction or convection furthermore any radiation variables. Moreover, the conditions at the space divider additionally should be indicated whether they are open, close (adiabatic) or symmetrical in nature. The significant strides in CFD is characterizing the geometry of the issue, isolating the volume into discrete cells likewise called as meshing, applying boundary conditions lastly tackling the governing equations. The discretization is vital as it changes over the differential equations into logarithmic equations. There are numerous approaches to discretize an issue and the essential strategies are Finite Difference Method, Finite Element Method and Finite Volume strategy. Limited Element Method is broadly utilized as a part of analysis of strong structures, yet can likewise be utilized to break down fluids. In this technique, the geometry is isolated into little components and tackled in connection with each other.

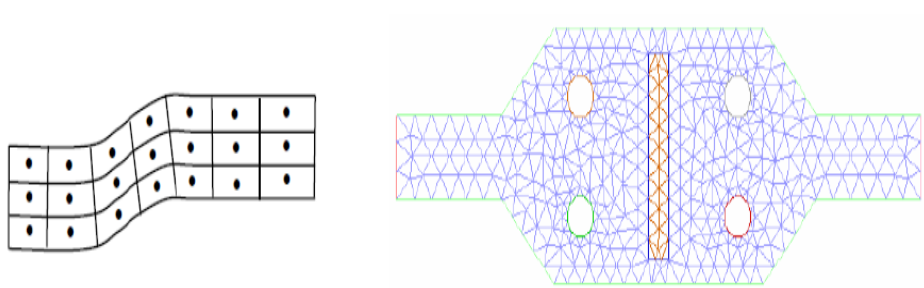

Figure 8 Discretization of cells by FVM [3]

Limited Volume Method where the governing equations are coordinated around the mesh components whose volumes are considered for the arrangement and Finite Difference Method where the differential terms are discretized into arrangement of grid focuses. The limited difference strategy gets to be hard to utilize when the coefficients included in the equation are discontinuous [3].

The computational fluid dynamics code considered for the numerical analysis in 6SigmaET® [10] is the limited volume technique where the arrangement area is discretized into control volume locales. In this manner, the governing equations are understood by incorporating over the control volume and applying dissimilarity hypothesis. Considering the volume of mesh components and the variables to be ascertained are situated at the centroid of the limited volume. FVM is not constrained by cell shape. It might be utilized on subjective geometries which is an essential motivation to utilize limited volume technique for fathoming the governing equations to be specific preservation of mass, protection of force and protection of vitality than the other computational strategies. The FVM is locally conservative as it is based on a "balance" approach [3]. In FVM, the solution domain is divided into finite control volumes by a grid. The grid defines the boundary conditions [3].

A series of algebraic equations are used for discretizing the results such that each of them relates a variable's value in a cell to its value in the nearest neighboring cell. As an example, the variable for velocity “V” can be calculated using the following algebraic equation:

$$T = \frac{C_0V_0 + C_1V_1 + C_2V_2 + C_3V_3 + \cdots \ldots\ C_nT_n + S}{C_0 + C_1 + C_2 + C_3 + C_n}$$

Where V_0 represents the velocity value within the initial cell; V_1, V_2 & V_3 are the velocity values in the neighbouring cells; C_0, C_1, C_2, C_n represent the coefficients that connect each cell value to each of its neighboring cell values; and S denotes the source term. These algebraic equations are solved for the field variables T, u, v, w and density ρ. This implies that if n cells are present in the domain, then a total of 5n equations are solved.

3.4 *Turbulence Modeling*

Turbulent flow is characterized as a flow regime described by irregular fluctuations in all headings and infinite number of degrees of freedom unlike laminar flow which is smooth and streamlined. The flow is depicted as three dimensional with fast changes in velocity and pressure. Flows at higher Reynolds number (more than a couple of thousand) are by and large viewed as turbulent while those with a lower Reynolds number are viewed as laminar.6SigmaET® uses K-Epsilon turbulence model.

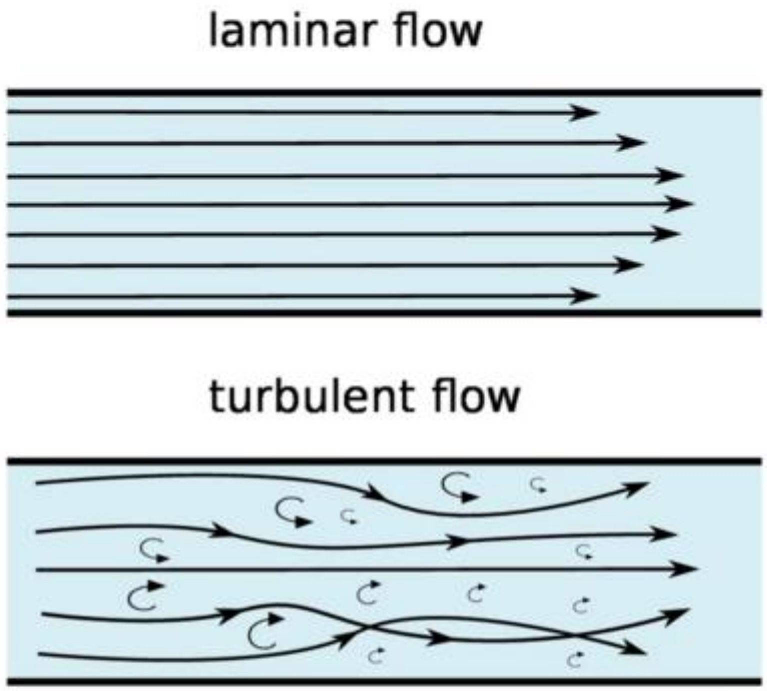

Figure 9 Laminar flow vs. Turbulent flow [8]

3.4.1 *K-Epsilon Turbulence Model*

K-Epsilon turbulence model is widely used model for turbulent flow modelling and is also commonly known as two equation model. This model uses two variables; the kinetic energy of turbulence (k) and the dissipation rate of kinetic energy of turbulence (ε) [11]. The two equation model computes viscosity depending on the grid cells instead of calculating the viscosity due to the walls. The K-Epsilon model is applicable for problems with thin shear layers and recirculating flows [11].

The governing transport equations are as below: [12]

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial \rho k u_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] + G_k + G_b - \rho\varepsilon$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial \rho \varepsilon u_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1S}\frac{\varepsilon}{k}(G_k + C_{3S}G_b) - C_{2S}\rho\frac{\varepsilon^2}{k}$$

3.5 *Objects*

The objects available in the software 6SigmaET® are defined with respect to the nomenclature used in the electronics industry. These predefined components/objects are equipped with the functions and behavior similar to actual components which makes the task of building a model and analysing it much easier.6SigmaET®'s these logical components eradicate the need of using simple blocks in the model and instead use objects like PCB, chassis, power supply, chip socket, etc. which come with defined properties like thermal, material, surface, etc. which can also be modified as per requirements of the developing model.

3.5.1 *Test Chamber*

Test chamber is a pseudo wind tunnel that can be generated to achieve the required external environment conditions for the model developed. It is an enclosure in which one can set up a model and structure the air flow in various ways to conduct the simulation. The airflow in test chamber can be either prescribed or external environment features can be attached to the walls of this chamber. In our case, top, bottom, left and right sides of the test chamber are specified as walls. And the airflow is through rear to front side of test chamber.

3.5.2 *Chassis*

Chassis is an enclosure that forms a casing of a typical IT equipment. By default, chassis is a rectangular box, measuring 440mm (X) by 500mm (Z) and 1U (Y) in height. The size can be adjusted as per one's requirement. In our case the chassis dimensions are 434mm (X) by 724mm (Z) and 1U(Y). It represents the outer shell of the server which holds all the electronic and other components. 6SigmaET® provides flexibility in changing the material and thickness properties as well as environment as per specifications for each side of the chassis individually. The chassis object is commonly used for modeling data centers, racks and servers. Everything directly attached to the Chassis is organized under the Chassis node and consists of:

Cooling - consists of Fans, Blowers and Heat sinks

Electronics - consists of all heat generating objects like Disk Drives and Bays, PCBs and the Components mounted on them, (including all of their associated hardware).

Fluid Cooling - includes any internal Ducting for a Fluid Cooling System and associated Pumps if they are internally mounted.

Obstructions – consists generally of no heat producing objects which cause obstruction to the air flow in the system such as internal ironmongery, DIP switches, crystals, and various battery types etc. and heat producing items such as transformers, large chokes and solid state relays etc.

Power - consists of Power supply units and their attached Fans (this is a logical object predefined in the software).

Sensors - consists of only the Sensors attached to the Chassis. As Sensors can be attached to many different objects, those Sensors will be shown in the Model Tree under the object to which it is attached.

3.5.3 *PCB*

Printed circuit board, (PCB) is sheets of copper laminated onto a non-conductive substrate. PCB has functional property of being the conductive path to the components mounted on it and interconnecting all the components, while leaving other areas non-conductive. In 6SigmaET®, PCB's are used to represent the motherboard housing the electronics of the server.

Sensors

Sensors are the devices used for measurement of the required data like temperature at a particular point or object within the solution domain. These can be attached to any object to record the desirable data. This includes chassis, PCB's, components, and vents. They are used for monitoring critical regions to record the temperature, pressure and velocity.

3.5.4 *Fans*

Fans are devices used for cooling electronic equipment by creating air to flow through the device. Fans create air flow by converting the torque supplied to the propeller shaft to impart kinetic energy to the air flowing across the fan rotor. In doing so, these devices also increase the static pressure across the fan rotor. The most commonly used fans in cooling applications are axial flow. Axial flow fans deliver air flow in the direction parallel to the fan blade axis. These fans can deliver very high flow rates. They produce air flows with high volume and low pressure. They are used for cooling IT equipment and several other electronic devices. Axial fans can be classified into propeller fans, tube-axial fans and vane-axial fans [3].

3.6 *PAC*

The PAC in 6SigmaET® stands for parameterize, analyse and compare is utilized for performing parametric examination for option cases. It permits us to change quantities of input conditions to consider comparing outputs by selecting the required input parameter. It decreases the time taken to compare models by incorporating the yield in a solitary window. A mesh sensitivity examination can likewise be performed by differing the quantity of grid elements and performing a parametric study to acquire solutions to all trials at the same time. Parametric investigation can be performed for fans with various fan speeds, flow rate, heat sink fin thickness, and fin count. Correspondingly, outputs that are required can be chosen in the arrange yield parameters apparatus. Particular screen focuses and locales can be chosen to decide minimum, maximum and mean temperature, pressure and velocity instantly. [3]

CHAPTER 4

CFD MODELING AND FLOW ANALYSIS

4.1 *Detailed Model Development*

We have used 6SigmaET®® to develop a detailed model of the server under consideration. It is not always feasible to include all the minute components on the motherboard as they might not contribute to the system resistance. Some components heat dissipation would be negligible when compared to the main heat generating components of the system. Hence the components are selected by their contribution to system resistance (determined by geometry) and heat generating capacity (determined by their Thermal Design Power TDP), for the detail model developed.

Table 1 List of TDP of major components

Device	Reference	TDP (W)
Micro-processors	CPU 1	135
	CPU 2	135
DIMMs	Total 16 No	0.81
C600 Chipset	PCH	8
Fans	Total 5 Fans	0.7-5.64
Hard drives	HDD	7.8

In the model created for the simulation, the air flows from the fans side and exists from the other end. The dimensions of the model are same as the server under consideration. The airflow is supplied in the test chamber to achieve the external environment similar to actual environment. The server fans, VRDs, capacitors, DIMM slots, and chipsets are modelled as per actual model.

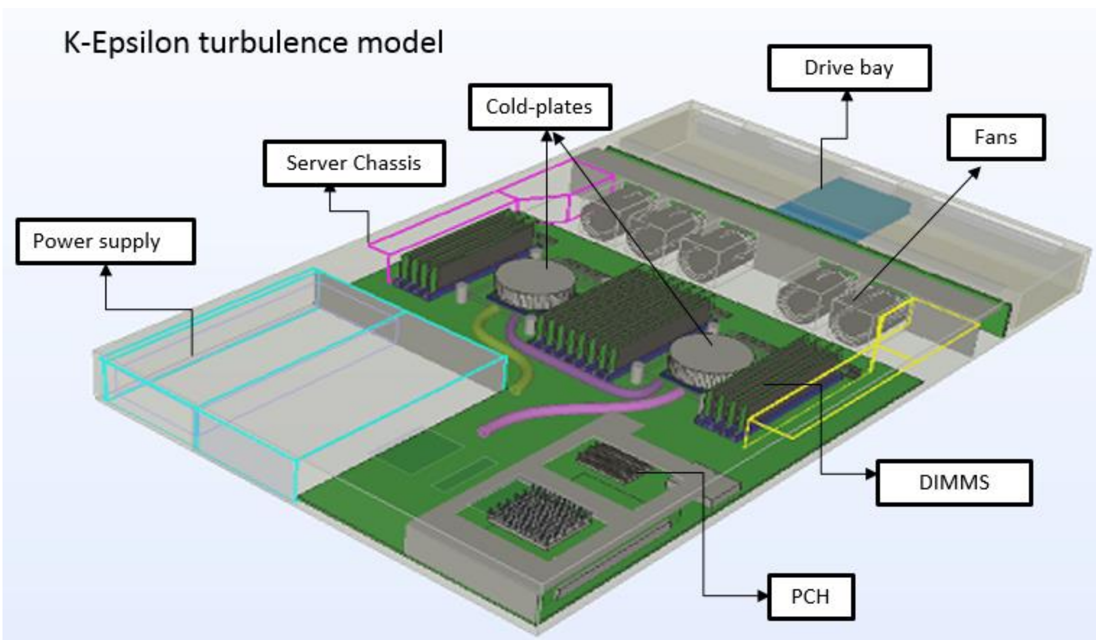

Figure 10 Detailed server model

CHAPTER 5

RESULTS & CONCLUSIONS

5.1 Mesh Sensitivity Analysis

We use the numerical analysis to avoid the make and break approach of the experimental setups. For this we carry out the analysis using the commercially available software like 6SigmaET® in this case.

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. It ensures the results are mesh size and count independent. For our model we varied the mesh count from 5 million to 65 million and found the pressure at the outlet to be constant after the mesh count of 50 million. To save the computational time, we carried out analysis at 50 million mesh count.

Table 2 Cell count analysis for the server

Cell count in Million	Pressure (Pa)
5	1.367
10	1.6
15	1.838
20	2.139
25	2.361
50	2.642

55	2.642
60	2.642
65	2.642

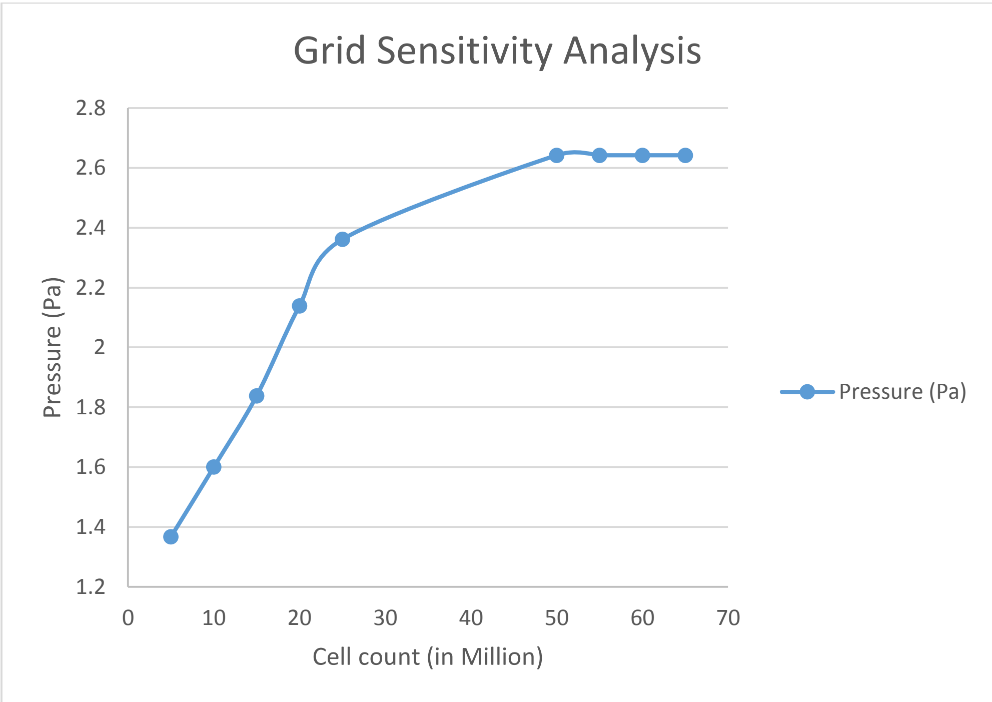

Figure 11 Mesh sensitivity analysis for the server

5.2 system impedance

The resistance offered by the system to the airflow is the system impedance. To determine the nature of the system under consideration, we can represent the system impedance in a graphical form by simulating the airflow through the system server. The air flow is given in the external chamber just like it would be in the air chamber in an experimental setup. The air flow rate is varied and the pressure drop across the inlet and outlet of the system is calculated. The fans of the servers are turned off during determining the system impedance.

Table 3 Static pressure with change in CFM

CFM	Press (in of H2O)
0	0
20	0.034
40	0.071
60	0.228
80	0.557
100	1.039
120	1.460

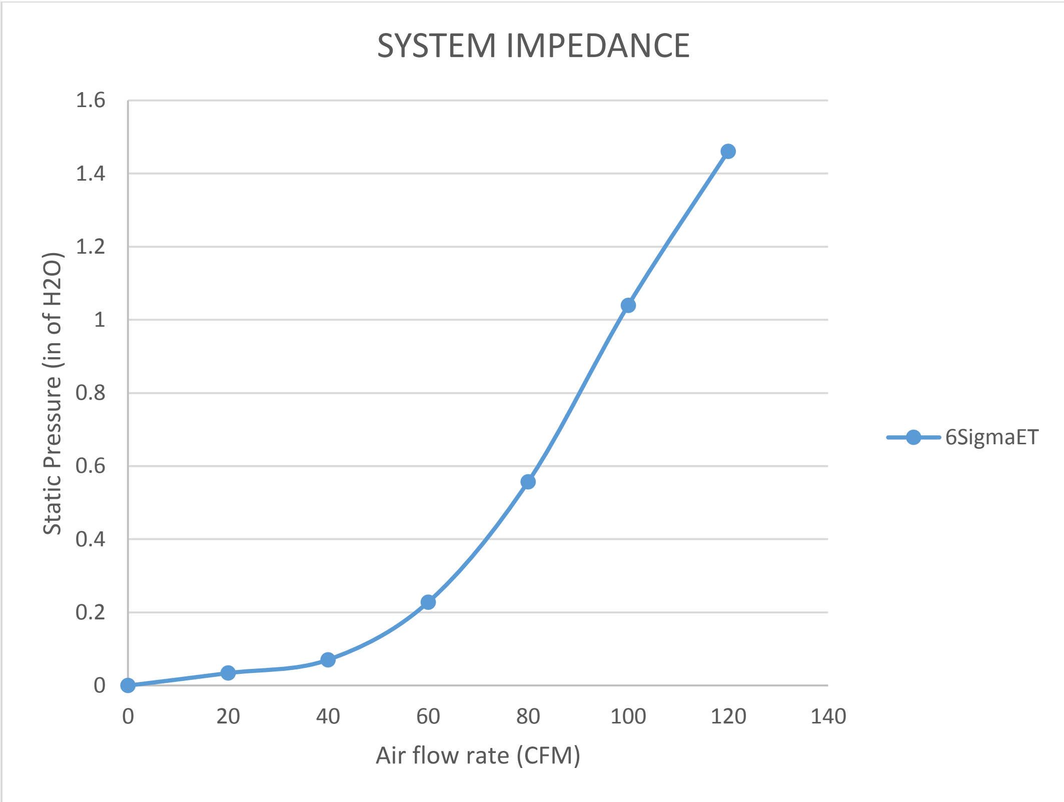

Figure 12 System impedance curve

5.3 Fan Flow Rate

To calculate the airflow through the server, the fans installed at the rear end were run at different speed and airflow at the inlet and outlet was measured in the simulation.

We have measured the airflow for 3 cases in this particular study. First all 5 fans are running t speed of 7000 rpm. In case 2, fan in front of one CPU is shut off and in case 3, fan in front of another CPU is also shut off.

Table 4 Comparison of airflow rate with change in fan speed

RPM	5 Fans	4 Fans	3 Fans
2000	12.3	11.3	10.03
4000	24.9	23.3	21.93
8000	36.2	32.1	31.44
12000	48.5	40.7	37.83
16000	61.05	52.52	46.53

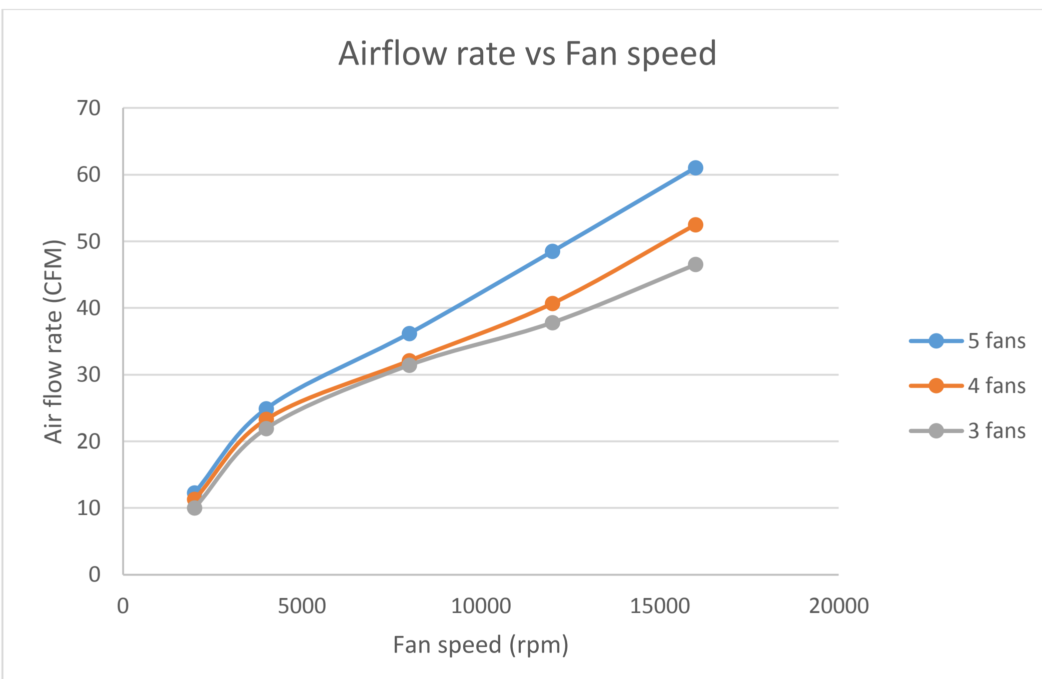

Figure 13 Air flow rate vs. fan speed

5.4 PCH temperature

In the server under consideration, the main heat generating components, the CPUs are water cooled by the cold-plates. The heat generating component that falls next in line to the CPUs in this server is PCH with TDP of 8 W and critical temperature of 92.7°C. We run the simulation by running first 5 fans, 4 fans and last 3 fans at varying speed to study the effect in temperature changes of the PCH.

Table 5 Comparison of PCH temperatures with change in fan speed

RPM	PCH temp 5 fans	PCH temp 4 fans	PCH temp 3 fans
4000	64.4	68.2	80.1
8000	60.6	63.5	75.2
12000	55.3	57	72.8
16000	50.2	53.1	65.1

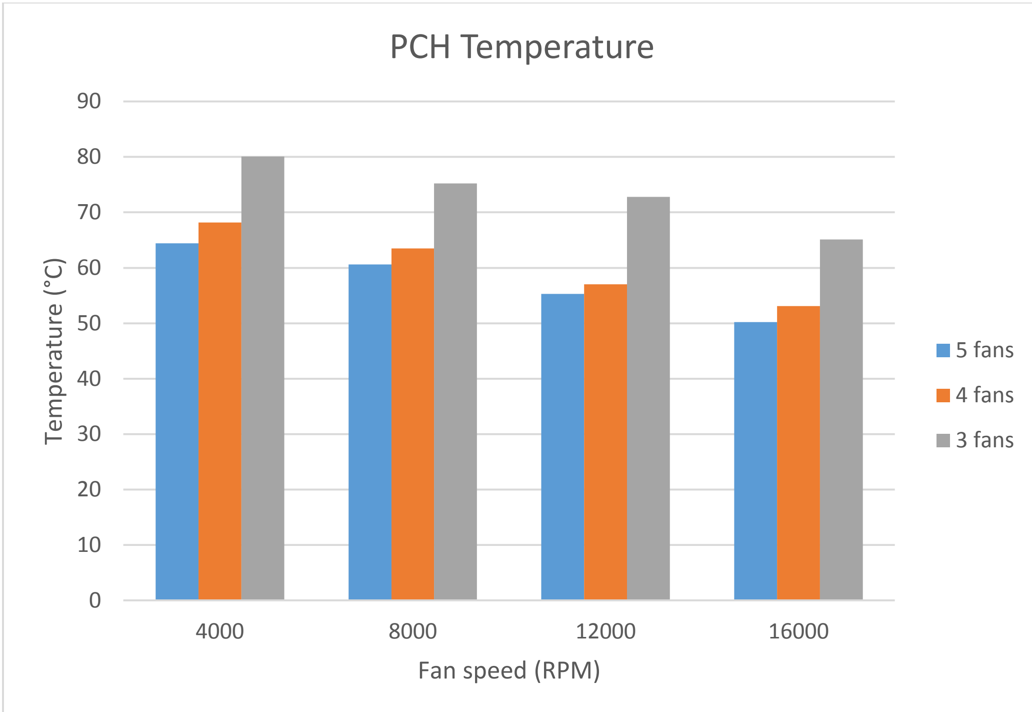

Figure 14 Comparison of PCH temperatures

CHAPTER 6

CONCLUSION

The detailed CFD model for the server is developed using 6SigmaET®. The system impedance curve is obtained for the model and when compared to other 1U servers, it follows the trend. We observe that At Idle fan speed, the air flow rate provided by 5 fans is 12.3 CFM. However, 3 fans, provide 10 CFM. The critical Temperature of PCH is 92.7°C. We also observe that the maximum temperature obtained with 3 fans is well below the critical temperature. Hence, safe to say we can reduce the fan power consumption. Hence, we can conclude that the CFD model can be used to optimize the airflow by reducing the fan use by 40% and achieve the required air cooling.

References

1. IDC: Data Center Investment is Critical to The Success of The Internet of Things. (n.d.). Retrieved May 05, 2016, from http://cloudtimes.org/2015/05/01/idc-data-center-investment-is-critical-to-the-success-of-the-internet-of-things/

2. Best Practices Guide for Energy-Efficient Data Center Design. (n.d.). Retrieved from http://energy.gov/sites/prod/files/2013/10/f3/eedatacenterbestpractices.pdf

3. "Development of detailed computational flow model of high end server and validation using experimental methods", Vijayalayan Pandiyan, The University of Texas at Arlington, December 2012

4. http://www.cisco.com/c/en/us/solutions/collateral/data-center-virtualization/unified-computing/white_paper_c11-680202.pdf

5. c220m3-sff-specsheet.pd.pdf

6. http://www.deltaww.com

7. SPECIFICATION FOR APPROVAL - partner.delta-corp.com. (n.d.). Retrieved from http://partner.delta-corp.com/Products/FanUploads/Specification/GFB0412EHS-DF00(rev00).pdf

8. http://www.cfdsupport.com/OpenFOAM-Training-by-CFD-Support/node263.html

9. Suhas V Patankar, "Computational fluid dynamics: Engineering Analysis and Application".

10. http://www.futurefacilities.com/software/et/6SigmaET.php

11. B.E. Launder and D.B. Spalding, "The Numerical Computation of Turbulent Flows", Appendix D of Computer Methods in Applied Mechanics and Engineering, 1974, pp. 269-289.

12. K.K. Dhinsa, C.J. Bailey, and K.A. Pericleous, "Low Reynolds Number Turbulence Models for Accurate Thermal Simulations of Electronic Components," Proc of the International Conference on Thermal and Mechanical Simulation and Experiments in Micro-Electronics and Micro-Systems, EuroSimE 2004.

Biographical Information

Neha Inamdar completed her undergraduate degree in Mechanical Engineering from the University of Pune, India in May 2012. She worked with Atlas Copco, India as a Sourcing Engineer for two years. She started working towards thesis under the guidance of Dr. Dereje Agonafer for Master's degree in December 2014. She also worked as an intern at Atlas Copco for seven months from June 2015 to December 2015.

Her research interests include Data Centre Cooling, Computational Fluid Dynamics, Thermal and Fluid Sciences.