USE OF A WINDBREAKER TO MITIGATE HIGH SPEED WIND LOADING ON A MODULAR DATA CENTER

by

PRASAD PRAMOD REVANKAR

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 2015

Acknowledgements

I would like to take this opportunity to thank my supervising professor Dr. Dereje Agonafer for his constant encouragement, support and guidance during the course of my research and studies at this University. The invaluable advice and support provided by him was the major driving force, which enabled me to complete my thesis.

I would like to thank Dr. Kent Lawrence and Dr. Haji-Sheikh for taking time to serve on my thesis committee. Also, I would like to thank Betsegaw Gebrehiwot for his invaluable support and timely inputs.

I am obliged to Ms. Sally Thompson and Ms. Debi Barton for helping me out in all educational matters. They have been very kind and supportive whenever I needed their help.

I would like to thank all my friends in the EMNSPC team and in the University for helping me throughout my time here at this University. Finally, I would like to thank my parents for their support, both emotionally and financially, without which I would not have been able to complete my degree.

April 21, 2015

Abstract

USE OF A WINDBREAKER TO MITIGATE HIGH SPEED WIND LOADING ON A MODULAR DATA CENTER

Prasad Pramod Revankar, MS

The University of Texas at Arlington, 2015

Supervising Professor: Dereje Agonafer

The data centers that are located in open regions are subjected to various environmental risks such as floods and very strong winds. As this wind blows over the data center, the pressure difference generated can have destructive effects on the data center. Various kinds of fences have been used as windbreakers to reduce the wind speed and divert the wind. A windbreaker basically acts as a barrier to the upstream wind and reduces the mean velocity of air downstream of the windbreaker, thereby reducing the wind loading on the objects situated behind the fence. The height, width, and void volume fraction of the windbreaker are main parameters that determine the level of wind speed reduction across the windbreaker. The aim of this computational investigation is to design a windbreaker for reducing sustained wind speed of 44.7 m/s (100 mph) to 9 m/s (20 mph) average velocity in the direction normal to wall, at a distance of 0.3 m (1 ft) from the wall of the modular data center (MDC).

The model of the windbreaker and an enclosure of the modular data center was created in FloTHERM 10.1. The windbreaker has a split wall configuration, in which two windbreakers, one on the ground the other on the roof of the data center, are used. Various parameters of the windbreaker such as height, void volume fraction and distance from the MDC are varied to study how these parameters affect the average normal-to-the-wall wind speed on a projected area of the MDC. The combination of these parameters that satisfy the criterion mentioned above are considered acceptable basic design parameters for the windbreaker. It is observed that, with the use of these windbreakers, there is a significant change in pressure that reduces the wind load induced damage. The design enables the use of existing modular data centers without the need for improving them to withstand high speed winds.

TABLE OF CONTENTS

Acknowledgements ii

Abstract iii

List of Illustrations viii

List of Tables ix

Chapter 1 INTRODUCTION 1

- 1.1 Modular Data Centers 3
  - 1.1.1 An Introduction 3
  - 1.1.2 Yahoo’s Chicken Coop Data Center 5

Chapter 2 WINDBREAKERS 8

- 2.1 Effectiveness of the Windbreak 10
  - 2.1.1 Effect of Height 10
  - 2.1.2 Effect of Distance 11
  - 2.1.3 Effect of Density 11
  - 2.1.4 Effect of Orientation 13
  - 2.1.5 Effect of Length 14

Chapter 3 COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS 15

- 3.1 Introduction to CFD Analysis 15
- 3.2 Governing Equations 16
- 3.3 Turbulence Modeling 19

3.3.1 LVEL Turbulence Model 19

3.3.2 K-Epsilon Turbulence Model 19

3.4 Meshing and Grid Constants 20

3.5 Smart Parts in FloTHERM 10.1 21

3.5.1 Cuboid 21

3.5.2 Resistance 21

3.5.3 Enclosure 22

3.5.4 Source 22

3.5.5 Monitor Points 22

3.5.6 Region 23

3.5.7 Command Center 23

Chapter 4 DESCRIPTION OF THE MODEL 25

4.1 Description of the Modular Data Center 25

4.2 Description of the Windbreaker 26

4.3 Description of the CFD Model 27

4.3.1 Dimensions of the Model 27

4.3.2 Model Setup 27

4.3.3 Boundary conditions 28

4.3.4 Meshing 29

4.4 Mesh Sensitivity Analysis 29

4.5 Scenarios Considered 31

Chapter 5 RESULTS AND CONCLUSION 32

5.1 For 5% void volume fraction 32

5.1.1 Velocity profile at 0.5m from MDC 33

5.1.2 Velocity profile at 2m from MDC 34

5.1.3 Velocity profile at 3.5m from MDC 35

5.2 For 10% void volume fraction 37

5.3 For 20% void volume fraction. 38

5.4 For 30% void volume fraction 39

5.5 For 40% void volume fraction. 40

5.6 Plot of Normalized Speed vs Windbreaker distance 40

5.6 Conclusion 41

Chapter 6 SCOPE AND FUTURE WORK 43

References 44

Biographical Information 45

List of Illustrations

Figure 1 Dell EPIC Modular Data Center for eBay [4] 4

Figure 2 Yahoo Compute Coop Data Center facility[5] 7

Figure 3 Windbreakers used at airports 9

Figure 4 Wind sped behind a windbreaker as percentage of incoming speed 9

Figure 5 Effect of density 12

Figure 6 Representation of a 3D grid 18

Figure 7 3D model of MDC 26

Figure 8 Split Wall configuration schematic 27

Figure 9 Mesh of the CFD model 29

Figure 10 Mesh Sensitivity Analysis 30

Figure 11 Half-model with boundary conditions 31

Figure 12 Lower wall at 0.5m from MDC 33

Figure 13 Upper wall at 0.5m from MDC 34

Figure 14 Lower wall at 2m from MDC 34

Figure 15 Upper wall at 2m from MDC 35

Figure 16 Lower wall at 3.5m from MDC 35

Figure 17 Upper wall at 3.5m from MDC 36

Figure 18 Particle track for 5% void volume fraction of windbreaker 36

Figure 19 Particle track for 20% void volume fraction of windbreaker 39

Figure 20 Graph of normalized speed vs windbreaker distance from MDC 41

List of Tables

Table 1 Saffir-Simpson Hurricane wind scale 2

Table 2 Dimensions of the model 27

Table 3 Mesh Sensitivity analysis... 30

Table 4 5% void volume fraction.. 32

Table 5 10% void volume fraction.. 37

Table 6 20% void volume fraction.. 38

Table 7 30% void volume fraction.. 39

Table 8 40% void volume fraction.. 40

Chapter 1

INTRODUCTION

Data Centers are a facility which store computer systems and other components, like data storage units, telecommunication systems, etc. The equipment in these data centers requires a certain controlled environment for its optimum functionality. This requires an efficient cooling system as well, since the huge amount of IT equipment within the data center produces a large amount of heat. So, the entire data center is an expensive facility that needs to be maintained and protected from any sort of damage or factors that would cause malfunctioning.

One of the major issues taken into consideration while designing a data center, is the environmental effects on the external structure. Earthquakes have the greatest effect on the structural integrity of a data center followed by wind loading. Strong, high speed winds can cause significant damage to the external structure or claddding of the data center. As the wind blows against the external structure, there is a build up of pressure against the external structure which is translated to the supporting structure. Also as the wind blows over the structure, it can generate an upward force on the roof. It can lead to localized damage to the roof, cladding or the support structure. In case of very high speed winds, which are caused by hurricanes or tornadoes, the wind loading can lead to lateral deformation of the structure.

Wind speeds are classified according to the Beaufort Scale which was developed by Sir Francis Beaufort in 1805. According to this scale, wind speeds in excess of 73.6 mph (32.924 m/s) are classified as hurricanes.[1] Hurricanes are classified by the Saffir-Simpson Hurricane Wind Scale. It is a 1 to 5 rating based on the hurricane's sustained wind speed. Wind speeds in excess of 96 mph (42.91 m/s) and within 110 mph (49.17 m/s) are classified as Category 2 Hurricanes. The following table shows the Saffir-Simpson Hurricane Wind Scale.[2]

Table 1 Saffir-Simpson Hurricane wind scale

Category	Sustained Wind Speed	Type of Damage
1	74 mph to 95 mph	Very dangerous winds will produce some damage: Well-constructed frame homes could have damage to roof, shingles, vinyl siding and gutters. Large branches of trees will snap and shallowly rooted trees may be toppled.
2	96 mph to 110 mph	Extensive damage caused: Well-constructed frame homes could sustain major roof and siding damage. Many shallowly rooted trees will be snapped or uprooted and block numerous roads.
3	111 mph to 129 mph	Devastating damage occurs: Well-built framed homes may incur major damage or removal of roof decking and gable ends. Many trees will be snapped or uprooted, blocking numerous roads.
4	130 mph to 156 mph	Catastrophic damage occurs: Well-built framed homes can sustain severe damage with loss of most of the roof structure and/or some exterior walls. Most trees will be snapped or uprooted and power poles downed. Fallen trees and power poles will isolate residential areas.
5	157 mph or higher	Catastrophic damage occurs: A high percentage of framed homes will be destroyed, with total roof failure and wall collapse. Fallen trees and power poles will isolate residential areas.

It is evident that wind speeds exceeding 74 mph itself can cause a substantial amount of damage to any building and any damage to an expensive facility like a data center would result in huge financial losses. The aim of this computational investigation is to design a windbreaker for reducing sustained wind speed of 44.7 m/s (100 mph) to 9 m/s (20 mph) average velocity in the direction normal to wall of the Modular Data Center.

1.1 Modular Data Centers

1.1.1 An Introduction

A modular data center system is a portable method of deploying data center capacity. They can be thought of as purpose-engineered modules and components to offer data center capacity with multiple power and cooling options. These modules can be shipped anywhere as required, according to customer's requirement. Modular data centers typically consist of standardized components, making them easier and cheaper to build.

Modular data centers come in two types of form factors. The more common type, referred to as containerized data centers or portable modular data centers, fits data center equipment (servers, storage and networking equipment) into a standard shipping container, which is then transported to a desired location. Another form of modular data center fits data center equipment into a facility composed of prefabricated components that can be quickly built on a site and added to as capacity is needed. For example, HP's version of this type of modular data center, which it calls Flexible Data Center, is constructed of sheet metal components that are formed into four data center halls linked by a central operating building. [3]

Modular data centers provide the option of rapid deployment directly to the required site, with pre-configured setup, supplied as a fully functional unit. This leads to savings in time, man power involved in construction and overall costs in shipping and delivering the equipment. Hence, modular data centers are a better alternative as compared to the traditional data centers.

Figure 1 Dell EPIC Modular Data Center for eBay [4]

The Dell EPIC MDC has room for 24 racks of IT gear and can have up to 50 kilowatts of power in a rack without melting, for a total of 1.1 megawatts. It uses outside air cooling and has evaporative cooling (using misting water to chill the air) for when the outside temperature gets too high. Dell has filled up sixteen of those racks with gear already for eBay as part of the rollout. [4]

1.1.2 Yahoo's Chicken Coop Data Center

There is an incredible similarity between a traditional chicken coop and Yahoo's 'chicken coop data centers. In this kind of design there are openings in the floor. The air from below the floor is drawn up through the coop keeping the chickens cool. The air movement also removes excess moisture. Similar principle is used in designing the data center, as this concept leads to ventilating a data center using a full-roof cupola system, which proves to be a great way to cool computing equipment. The first design was standard fare having a raised-floor white-space and forced-air cooling.

The company's second design, Yahoo Thermal Cooling (YTC) uses a different approach. The white-space in a YTC data center is considered the cool zone. Hot air exiting the server rack is captured in an enclosed space and forced up through an inter-cooler. What makes the YTC concept unique is the fact that server fans move the air. The entire structure acts as an air-handler, wherein the hot air is allowed to rise via natural convection. Also, the use of evaporative coolers during the hot summer months along with free cooling, reduces the need for chiller-systems and air handling equipment. The entry and exhaust of air is controlled by a louver system which works based upon the internal temperature with the data center. It also consists of fan modules, filter assemblies and evaporative (water) Inter-Cooling Modules. [5]

The Yahoo Chicken Coop design has three different cooling modes:

- Unconditioned Outside Air Cooling: When air temperature is between 70°F and 85°F (21C to 29C), air enters the data-center through louvered side walls, which is filtered and drawn through the servers by fans housed in the rack-mounted computing and networking devices. This now-hot exhaust air after passing through the servers moves up into the attic through natural convection. The exhaust air continues out of the data center through the adjustable louvers in the roof-length cupola.
- Outside Air Tempered by Evaporative Cooling: When the air temperature is above 85°F, air takes the same path as the previous case, except that shortly after entering the outer louvered walls, it is drawn through saturated media (Inter-Cooling Modules) in order to provide evaporative cooling to this incoming hot air.
- Mixed Outside Air Cooling: When the air temperature is below 70°F, especially during winter months, heated exhaust air is mixed with incoming outside air to maintain an air temperature of 70°F. This is achieved by recirculating fans and a control system which closes the louvers when the outside air temperature is below 70°F.[5]

A similar system is used in other data centers which use the Chicken Coop design. This design has proved to be very energy-efficient. It was found that, approximately 36 million gallons of water were saved per year with the chicken coop design, compared to conventional water-cooled chiller plant designs having comparable IT loads. Also, this design realized an almost 40 percent cut in the amount of electricity used relative to industry-typical legacy data centers. [6]

Figure 2 Yahoo Compute Coop Data Center facility[5]

Chapter 2

WINDBREAKERS

Windbreaks are barriers used to reduce wind speed and also to redirect the wind. They usually consist of trees and shrubs, but may also include perennial or annual crops and grasses, fences, or other materials. As a result of the reduction in wind speed due to the windbreak, the environmental conditions are modified in the region behind the windbreak, referred to as sheltered zone.. As wind blows against a windbreak, there is a high pressure zone created on the windward side (the side towards the wind) and a low pressure zone created on the leeward side, and large quantities of air move up and over the top or around the ends of the windbreak.

As far as the effects of wind on a structure is concerned, for mechanical damage and loads, the driving force is wind power. Wind power is defined as the square of wind speed. So, when wind speed is reduced to 25% of original, the wind power becomes 6.25% of its original value. Wind power is what you feel when you try and stand up in a strong wind. Clearly, even a small reduction in wind speed is enough to cause a dramatic reduction in wind power. With erosion the effect is even more pronounced as dust transport is proportional to wind speed cubed, or wind speed x wind speed x wind speed.

Figure 3 Windbreakers used at airports

A windbreak (also called a wind fence or wind shelter) can reduce wind speeds by over 50% of the incoming wind speed over large areas, and over 80% over localized areas. The figure shows reduction in wind speed behind a windbreak.

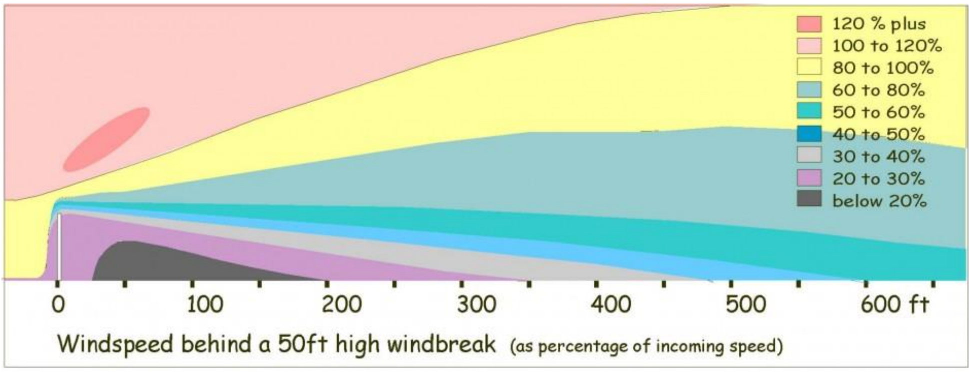

Figure 4 Wind speed behind windbreaker as percentage of incoming speed

2.1 Effectiveness of the Windbreak

The effectiveness of a windbreak in reducing the wind speed and altering the microclimate are determined by various characteristics of the windbreak structure. These characteristics include: height of the windbreak, distance of the windbreak from area to be sheltered, density of the windbreak, length and orientation.

2.1.1 Effect of Height

Windbreak height (H) is an important factor in determining the downwind area protected by a windbreak. This value varies from windbreak to windbreak. In farming applications, there are multiple row windbreaks. In this case the height of the tallest tree-row determines the value of H.

On the windward side of a windbreak, wind speed reductions are measurable upwind for a distance of 2 to 5 times the height of the windbreak (2H to 5H). On the leeward side (downwind side), wind speed reductions occur up to 30H downwind of the barrier. Within this protected zone, the structural characteristics of a windbreak, especially density, determine the extent of wind speed reductions.[7]

In the case of data centers, it is required that the wind speed is reduced only in an area immediately after the windbreak, that is just before the point of entry and exhaust of the air into and out of the data center, respectively. Hence, we do not need protection up to a large distance downwind. So the height H of the windbreak is chosen equal to the height of the data center itself.

2.1.2 Effect of Distance

Distance of the windbreak from area to be protected also plays an important role in reducing the wind speeds downwind. It is observed that for a fixed height, the area protected downwind is fixed. Although, placing the windbreaker very close to the data center would have the data center in the protected region, it would also choke the inlet and obstruct the outlet. While placing the windbreak too far away from the data center would not serve the purpose, that is reduced wind speed before the inlet and outlet of the data center. Therefore, it is necessary that the windbreaker is placed at an optimum distance from the data center in order to achieve its full benefit.

2.1.3 Effect of Density

Windbreak density also referred to as void volume fraction, is the ratio of the solid portion of the barrier to the total area of the barrier. When the wind is obstructed by a very dense windbreak, a low pressure develops on the leeward side. This low pressure area behind the windbreak pulls air coming over the windbreak downward, creating turbulence and reducing protection downwind. As void volume fraction of the windbreak increases, the amount of air passing through the windbreak increases, moderating the low pressure and turbulence, and increasing the length of the downwind protected area. While this protected area is larger, the wind speed reductions are not as great. By adjusting windbreak density different wind flow patterns and areas of protection are established.[7]

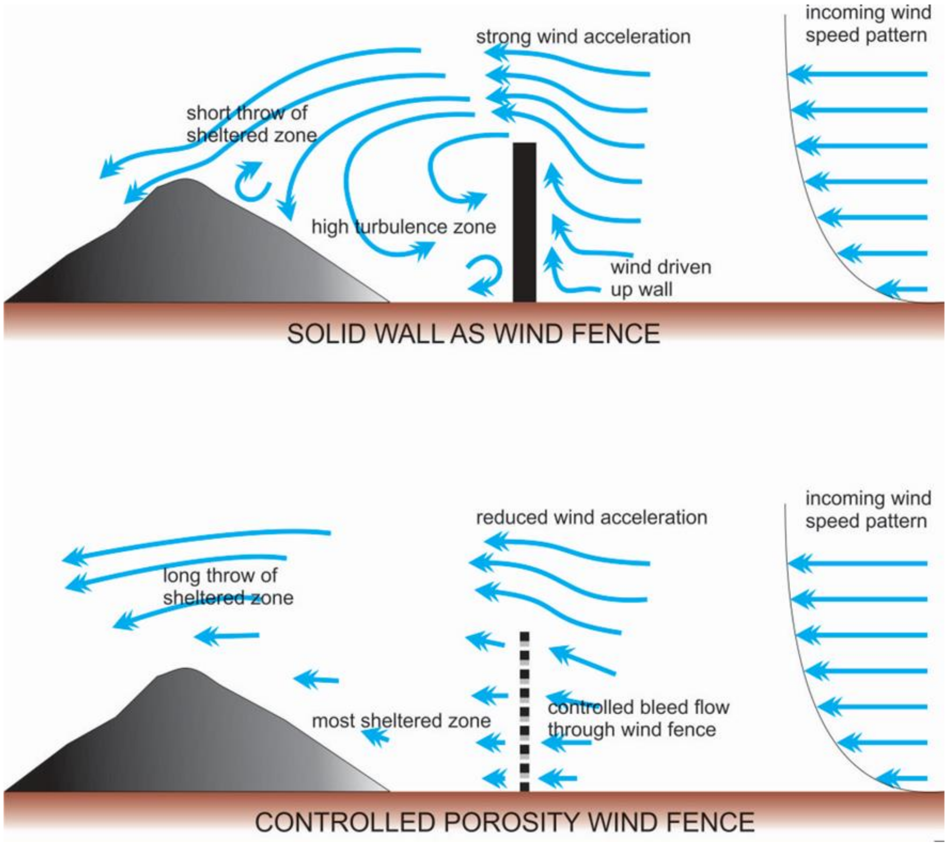

Figure 5 Effect of density

As windbreaks are mostly used in farming applications, while designing a windbreak, density should be adjusted to meet landowner objectives. A windbreak density of 40 to 60 percent provides the greatest downwind area of protection and provides excellent soil erosion control. To get even distribution of snow across a field, densities of 25 to 35 percent are most effective, but may not provide sufficient control of soil erosion. Windbreaks designed to catch and store snow in a confined area usually have several rows, and densities in the range of 60 to 80 percent. In case of farmsteads and livestock areas needing protection from winter winds require multiple row windbreaks with high densities. In these cases, wind speed reductions are greater but the protected area is smaller.[7]

In case of data centers, the effect of void volume fraction is studied to determine the optimum amount of void volume fraction required in the windbreak in order to obtain reduced velocity at the inlet and outlet of the data center. A solid wall with no void volume fraction, would totally block off an incoming wind, but this would not suit the purpose as it would not allow any air to enter the data center and the option of using the ambient air to cool the data center would not be available. Hence, some of void volume fraction is necessary in the windbreaker.

2.1.4 Effect of Orientation

The most effective orientation for a windbreak is to have it perpendicular to the prevailing winds. The purpose and design of each windbreak is unique, thus the orientation of individual windbreaks depends on the design objectives. Farmsteads and feedlots usually need protection from cold winds and blowing snow or dust. Orienting these windbreaks perpendicular to the troublesome winter wind direction provides the most useful protection. Field crops usually need protection from hot, dry summer winds, abrasive, wind-blown soil particles, or both. The orientation of these windbreaks should be perpendicular to prevailing winds during critical growing periods.

Although wind may blow predominantly from one direction for a season, it rarely blows exclusively from that direction. As a result, protection is not equal for all areas on the leeward side of a windbreak. As the wind changes direction and is no longer blowing directly against the windbreak, the protected area decreases. Again, individual placement depends on the site, the wind direction(s), and the design objectives.[7]

2.1.5 Effect of Length

Although the height of a windbreak determines the extent of the protected area downwind, the length of a windbreak determines the amount of total area receiving protection. For maximum efficiency, the uninterrupted length of a windbreak should exceed the height; by at least 10:1. This ratio reduces the influence of end-turbulence on the total protected area. The continuity of a windbreak also influences its efficiency. Gaps in a windbreak become funnels that concentrate wind flow, creating areas on the downwind side of the gap in which wind speeds often exceed open field wind velocities. Where there are gaps, the effectiveness of the windbreak is diminished. Lanes or field accesses through windbreaks should be located to minimize this effect or if possible avoided altogether.[7]

Chapter 3

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

3.1 Introduction to CFD Analysis

CFD is a branch of Fluid Dynamics which deals with the analysis of problems involving fluid flow and heat transfer. It uses numerical methods and algorithms to solve and analyze problems. Computational fluid dynamics is applied to simulate and analyze the behavior of fluids in various systems. The major advantage of numerical methods is that, the problem is discretized based on certain parameters and solved. A mathematical model is generated, which represents to actual physical system and then it can be solved and analyzed. In this case the study involves the effect of fluid (air) flow past the Modular Data Center and the Windbreaker and how this affects the velocity, pressure and other characteristics in the system.

CFD is concerned with the numerical simulation of fluid flow, heat transfer and related processes such as radiation. The objective of CFD is to provide the engineer with a computer-based predictive tool that enables the analysis of the air-flow processes occurring within and around different equipment, with the aim of improving and optimizing the design of new or existing equipment.[8]

3.2 Governing Equations

The numerical solution for most problems are obtained by solving a series of three differential equations, collectively referred to as the Navier-Stokes' Equations. These differential equations are the conservation of mass, conservation of momentum and conservation of energy.

But in this particular case, temperature is constant and the effect of flow is analyzed. Hence, only conservation of mass and conservation of momentum equations are solved.

In general form,

The conservation of mass is given by:

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

The conservation of momentum is given by:

$$\frac{\partial(\rho u)}{\partial t}+(\rho u.\nabla)u=\nabla.(\mu\nabla u)-\nabla p+\rho f$$

The solution domain is the region or space within which these differential equations are solved. The solutions are obtained by imposing certain boundary conditions for this solution domain. The boundary conditions for most problems include ambient temperature, pressure, wind conditions and other environmental conditions. Also, if there is heat transfer involved then, type of heat transfer, such as conduction, convection or even radiation are considered. The conditions at the domain wall are also specified, whether they are open, closed or symmetrical in nature. The fluid properties like density, viscosity, diffusivity and specific heat need to be specified. [9]

The governing equations for many problems are solved using numerical techniques like Finite Element Method, Finite Volume Method and Finite Difference Method. In FEM, the elements are varied and approximated by a function, in FVM the equations are integrated around a mesh element whose volumes are considered and in FDM the differential terms are discretized for each element.

In the CFD technique used in FloTHERM 10.1, the conservation equations are discretized by sub-division of the domain of integration into a set of non-overlapping, continuous finite volumes referred to as ‘grid cells’, ‘control cells’ or quite simply as ‘cells’. The governing equations are solved by considering the volume of the grid cells and the variables to be calculated are situated at the center of these grid cells.

The finite volume method is more advantageous than other computational methods as it the governing equations are conserved even on coarse grids and it also does not limit cell shape. A set of algebraic equations are used for discretizing the results, each of which relates the value of a variable in a cell to its value in the nearest-neighbor cells.

For example let T denote the temperature, this can be calculated using the algebraic equation:

$$T=\frac{C0T0+C1T1+C2T2+\cdots CnTn+S}{C0+C1+C2+\cdots Cn}$$

Where T0 represents temperature value in the initial cell, T1, T2….,Tn are values in the neighboring cells; C0, C1, C2,…, Cn are the coefficients that link the in-cell value to each of its neighbor-cell values. *S* denotes the terms that represent the influences of the boundary conditions.

These algebraic equations are solved for the variables like *T, u, v, w* and *p*. This means that if there are 'n' cells in the solution domain, a total of '5n' equations are solved.

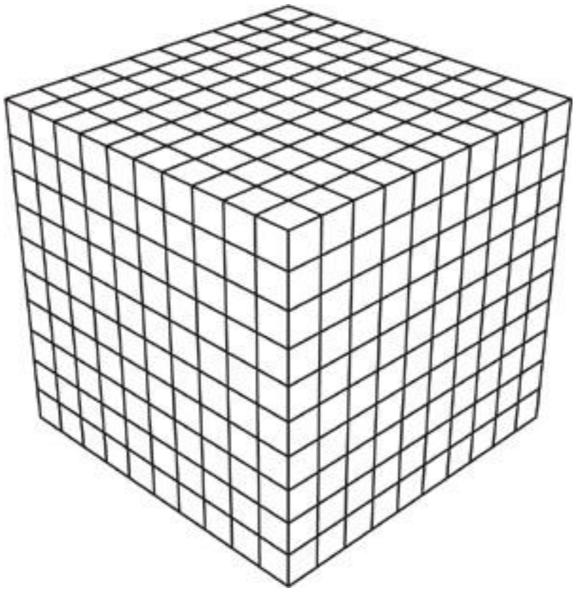

Figure 6 Representation of a 3D grid

3.3 Turbulence Modeling

A flow is said to be turbulent when the fluid undergoes irregular fluctuations or mixing. The velocity of the fluid at a point is continuously undergoing changes in both magnitude and direction, as opposed to laminar flow wherein the fluid moves in smooth paths or layers. Usually fluid with large Reynolds number are considered to be turbulent, while fluids with low Reynolds number are considered laminar. FloTHERM 10.1 uses two common methods to model turbulent flows: LVEL turbulence model and K-Epsilon turbulence model.

3.3.1 LVEL Turbulence Model

The LVEL turbulence model requires only few terms to determine the effective viscosity. They are nearest wall distance (L), the local velocity (VEL) and the laminar viscosity. In this model, Poisson's equation is solved initially to calculate the maximum length scale and local distance to nearest wall.

$$D = (|\nabla\emptyset|^2 + 2\emptyset)^{1/2}$$

$$L = D - |\nabla\emptyset|$$

Where $|\nabla\emptyset| = -1$ and $\emptyset = 0$ (which is boundary condition at the wall)

$\emptyset$ is the dependent variable. [10]

3.3.2 K-Epsilon Turbulence Model

The K-Epsilon turbulence model solves the governing equations along with another two additional equations, namely, the kinetic energy of turbulence (k) and the dissipation rate of kinetic energy turbulence (ε). It is also known as the two equation model and is used widely in turbulent flow modeling.[10] The two additional transport equations solved are:

- Kinetic Energy of turbulence equation (k)

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

- Dissipation rate of kinetic energy of turbulence (ε)

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

3.4 Meshing and Grid Constants

Grid constraints allow you to attach minimum grid requirements to a geometry so as to make sure that there is sufficient grid coverage wherever it is located in the solution domain. Grid constraints are used to specify minimum and maximum number of cells across a geometry.

Meshing is an important feature of any CFD software, since if the model created is not properly meshed, the results of the simulation would be inaccurate. The mesh needs to be fine in critical areas and can be coarse in areas of less importance. Keeping the mesh fine in critical areas would give the most accurate results. Also, a mesh sensitivity analysis can determine when the solution has reached grid independence. Grid independence is the point at which, addition of a large number grid cells has no further effect on the solution. FloTHERM 10.1 uses a Cartesian grid system and the values of any variable are calculated at the center of each grid cell. While meshing the model created in FloTHERM 10.1 there is an option of keeping the grid fine, medium or coarse. Also, localizing a grid is another option to improve the mesh. In this feature, the grid lines from two different regions do not interfere. The point where gridlines meet the edges of an object, they get truncated.

3.5 Smart Parts in FloTHERM 10.1

3.5.1 Cuboid

This is the most basic smart part in FloTHERM 10.1. It is used in representing most objects in the system. It is a solid block and can be used to represent any solid object like the external structure of the modular data center, a solid wall, etc. It also has the option to be collapsed to represent a plate.

3.5.2 Resistance

The resistance smart part is used to define region which acts as a barrier or a resistance to a flow. They can be collapsed, angled or non-collapsed depending on the requirement. They usually represent a porous media. The void volume fraction can be defined from the resistance library depending on the requirement. In this model the resistance smart part is used to represent a porous windbreaker. Different values of void volume fraction are given to the resistance to simulate a windbreaker with varying void volume fraction. In other applications, this smart part is used for modelling filters and vents.

3.5.3 Enclosure

The enclosure smart part is a hollow part which can used to define the outer boundaries of an object or system. It is cuboidal in shape and each side of the cuboid can have independent properties. We can remove certain sides of the enclosure and keep the remaining. In this manner we can use the enclosure smart part to simulate a wind tunnel, by keeping two the sides along its length, open. The thickness of the enclosure smart part can be specified or it can be kept as thin. It is also used to model racks, servers, etc.

3.5.4 Source

The source smart part is used to represent objects that require power to be defined. This smart part is used to simulate a wind source, with a fixed velocity assigned to it. The direction of the wind source can be defined as needed. The source area is made equal to the area of the open side of the enclosure, such that all the wind source velocity is channeled through the enclosure, thus acting as a wind tunnel.

3.5.5 Monitor Points

Monitor points are used to determine various parameters at critical points within the solution domain. They are usually used to monitor temperature or pressure at certain critical points within the system. In this case, monitor points are used to monitor pressure in regions close to the inlet and exhaust of the modular data center.

3.5.6 Region

This smart part is used for two main purposes. The first one being to create a refined mesh in a particular area of interest. A region can be created in a particular area where a finer mesh is required and the mesh density can be improved in that particular region using the Localize feature. Another use of the region is that it can be used in post processing. The values of different parameters like, velocity, pressure, speed, temperature can be determined within a particular region.

3.5.7 Command Center

The command center in FloTHERM 10.1 can be used to generate and solve different scenarios at the same time, enabling you to quickly see the effects of changing certain selected variables. This enables us to vary multiple parameters and see their individual or combined effect on the total system. Parametric and mesh sensitivity analysis can be performed using the command center. Mesh sensitivity analysis can be performed by varying the number of grid cells and studying its effect on the other parameters, by performing all the trials simultaneously. A parametric analysis can be performed for different components, for example, a parametric analysis of the fence is performed by varying parameters like distance from the MDC, void volume fraction and height.

Basically, the procedure that is followed while using the command center is that, a datum case is loaded as the project. Then in the command center, the parameters that are to be varied, called as the Input Variables are selected. This generates the different scenarios based on the number cases created. There is a graphical input tab which enables us to view what change has occurred in the model, due to the scenarios created by the input variables defined. The results that are required, for example, pressure, temperature, velocity, speed, etc. can be selected in the Output Variables tab. These can be viewed in the Scenario Table generated along with the input variables as well. There is also a tab called Solution Monitoring, for monitoring the solution and check for convergence.

Chapter 4

DESCRIPTION OF THE MODEL

The Computational Fluid Dynamics model is of a chicken coop modular data center and the two windbreakers, which are located at a certain distance in front of the inlet and exhaust of the MDC. Only the external structure of the MDC is modelled as this study is concerned with the wind loading on the external structure itself. The model was entirely created using FloTHERM 10.1 smart parts.

4.1 Description of the Modular Data Center

The modular data center modelled is a chicken coop modular data center, whose structure is similar to the Yahoo's Chicken Coop Data Center. It has two sections, the lower section which is the IT pod, which consists of all the IT equipment, filtration units, fans, etc. It also consists of the inlets for the ambient air to enter the data center, which is used for cooling purposes. The top section consists of the chicken coop or chimney, which acts as the exhaust for the hot air that rises from the hot aisle within the modular data center.

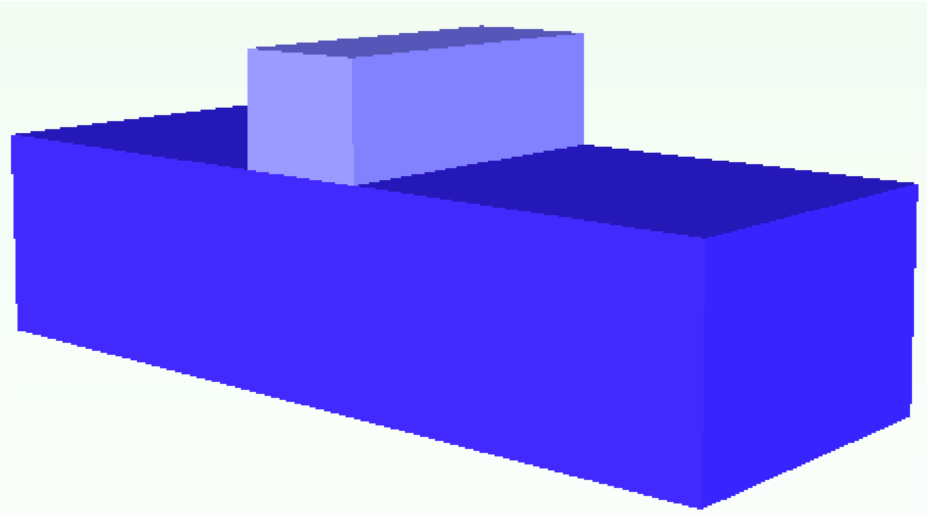

Figure 7 3D model of MDC

4.2 Description of the Windbreaker

The windbreaker used in this model is of a split wall type. One wall is placed on the ground and the second one is placed on top of the IT pod, in front of the chicken coop (exhaust). The height of the windbreaker is chosen equal to the height of the MDC. The windbreakers are modelled using the resistance smart part in FloTHERM 10.1. This allows the windbreaker to be modelled as a porous body with varying resistances. The initial case chosen was with 5% void volume fraction.

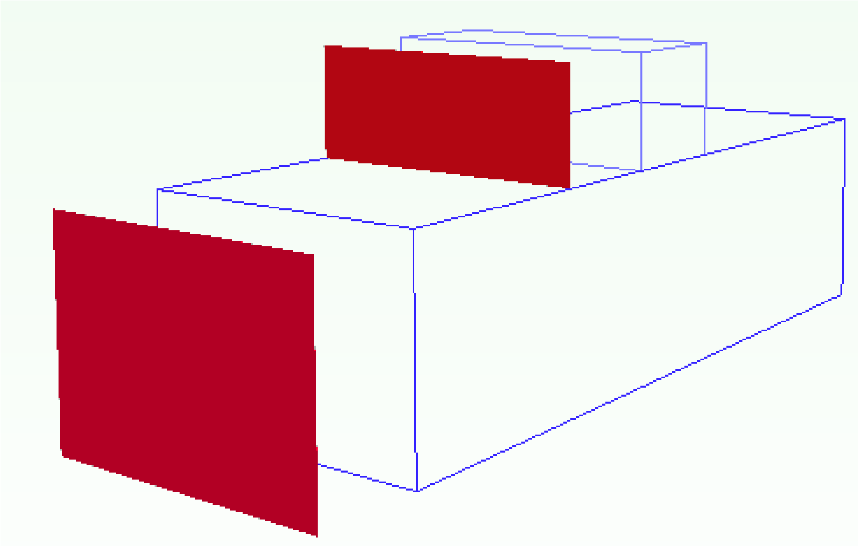

Figure 8 Split Wall configuration schematic

4.3 Description of the CFD Model

4.3.1 Dimensions of the Model

Table 2 Dimensions of the model

	Length	Width	Height
Lower portion of MDC	14160	5450	3740
Upper portion of MDC	2200	5450	2000
Lower portion of windbreaker	100	5450	3740
Upper portion of windbreaker	100	5450	2000

4.3.2 Model Setup

The model setup used for all the cases was as follows:

- Type of Solution was set to Flow Only
- Dimensionality was selected as 3-Dimensional

- Gravity was set to Normal, acting in –Y direction with Automatic value
- The flow type was set to Turbulent and Turbulence Model used was the LVEL K-Epsilon Model
- Ambient Pressure was 1 Atm
- Wind Velocity was set to 44.704 m/s originating from Source created using Source smart part in FloTHERM 10.1, direction of wind was in X direction

Note: The reason for selecting the Turbulent flow model was on account of very high fluid velocity (44.704 m/s) and also the Reynolds Number calculated was very high as well. The K-epsilon model is chosen as there are large empty volumes within the enclosure and the representation of turbulence is necessary. Since this model solves the two turbulence equations in addition to the governing equations as well, it proves to be the best model to use in this particular case.

4.3.3 Boundary conditions

The boundary conditions selected for the model are as follows, the ground or Y-low face of the solution domain has no slip condition. The Z-low face has the symmetric boundary condition. This is done to take advantage of the symmetry of the system. This saves the computational time by a large amount, since the number of grid cells are reduced by half the amount. The rest of the faces are all in the open condition.

4.3.4 Meshing

The mesh created is very fine in some locations and coarse in locations which do not affect the flow characteristics. In the regions where there is large amount of change in pressure and turbulence, the mesh is finer, while the regions within the MDC which does not affect the flow, the mesh is coarse.

The total mesh size is about 3.6 million cells and the maximum aspect ratio is 4.90. Since the model is symmetrical, a half model is created and meshed. Initially the computational time was very high due to the fine mesh, but the use of the half model reduces the computational time by almost 50 percent.

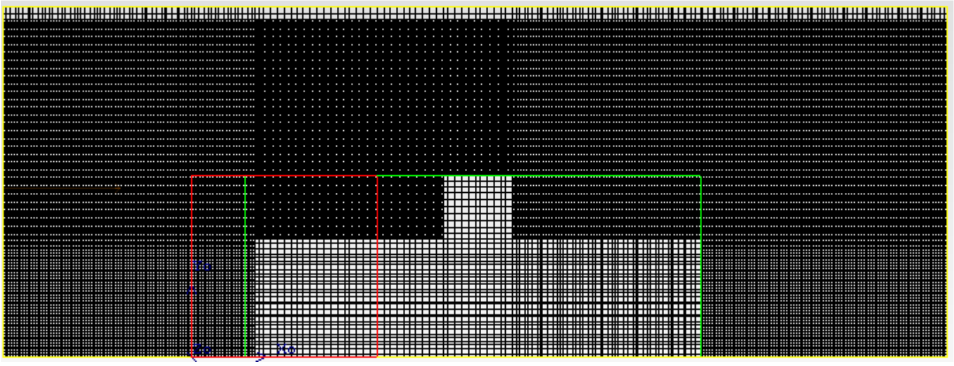

Figure 9 Mesh of the CFD model

4.4 Mesh Sensitivity Analysis

Mesh sensitivity analysis is carried out to check for uncertainties in the model. It is done to increase the understanding of the relationship between the input and output variables of the model. In the mesh sensitivity analysis carried out in this case, the input variable is the number grid cells is varied and its effect on the output variables like speed and pressure are studied.

The following mesh sensitivity analysis was carried out on the model.

Table 3 Mesh Sensitivity analysis

Cumulative total	Speed	% skewness
852681	9.03384	0
1878199	7.80365	13.62
2737552	7.91442	1.41
3604554	7.94467	0.37
4854640	7.95409	0.13
6580910	7.94887	0.06

Grid independence was achieved at 3.6 million cells with maximum aspect ratio of 4.9.

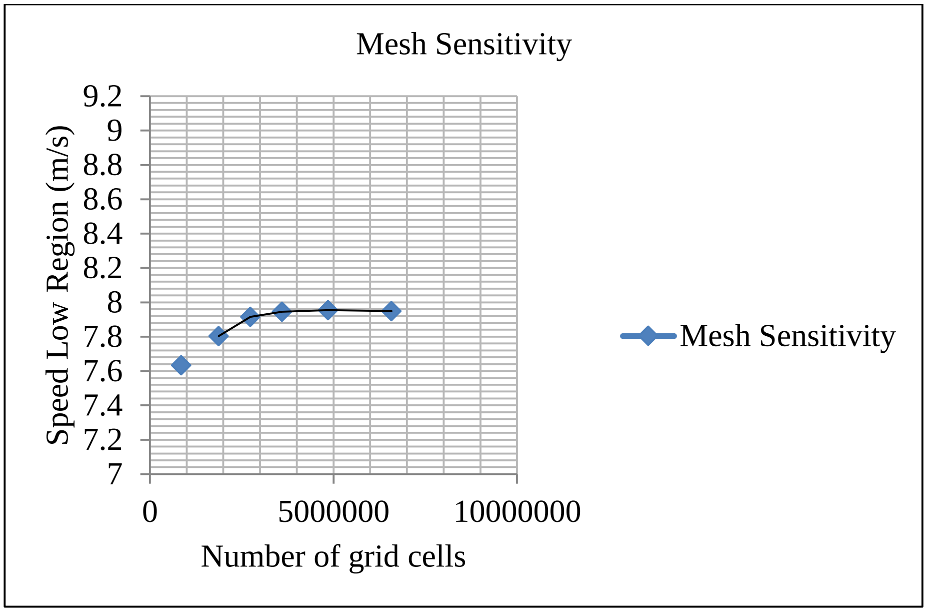

Figure 10 Mesh Sensitivity Analysis

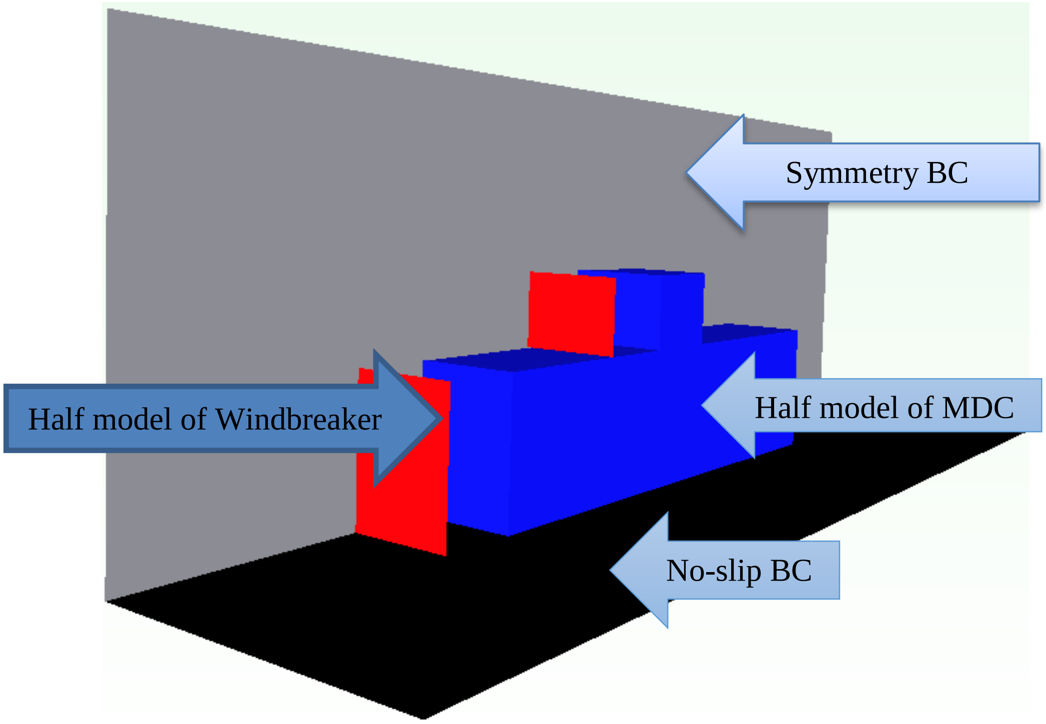

Figure 11 Half-model with boundary conditions

4.5 Scenarios Considered

The two main parameters that were considered in this investigation were Distance of the windbreakers from the MDC and void volume fraction of the Windbreakers. The initial case considered was wherein, both the upper and lower wall were taken at 2 m from the MDC. The distance was then varied in the steps of 0.5 m. A total of six cases were considered, at distances of 0.5,1, 1.5, 2.5, 3 and 3.5 meters. Void volume fraction was varied from 5%, 10%, 20% and 30%. Void volume fraction greater than 30% would not be serve the purpose as the wind speeds at the walls of the MDC would be higher than 20 mph, irrespective of the position of the windbreaker, which is not desirable.

Chapter 5

RESULTS AND CONCLUSION

The following results were obtained by varying the distance of the windbreaker from the MDC and the void volume fraction of the windbreaker. The results were monitored at a region which is 1 ft. (0.3 m) away from the MDC.

Assumption: The maximum wind speed that is acceptable at a distance of 1 ft. from the MDC is 20 mph (9 m/s).

5.1 For 5% void volume fraction

Table 4 5% void volume fraction

Scenario		1	2	3	0	4	5	6
Distance from MDC (m)		0.5	1	1.5	2	2.5	3	3.5
Lower Region	Mean Speed (m/s)	9.61	5.93	5.87	7.95	10.34	10.86	11.77
	Mean Pressure (Pa)	-1480	-1382	-1329	-1309	-1267	-1251	-1235
Upper Region	Mean Speed (m/s)	5.25	6.63	9.30	11.73	11.91	12.26	12.57
	Mean Pressure (Pa)	-1106	-1018	-944	-886	-807	-722	-628

For the lower region, maximum reduction in speed is observed when the windbreaker is placed at 1.5 m from MDC and for upper region, it is 0.5 m from MDC.

5.1.1 Velocity profile at 0.5m from MDC

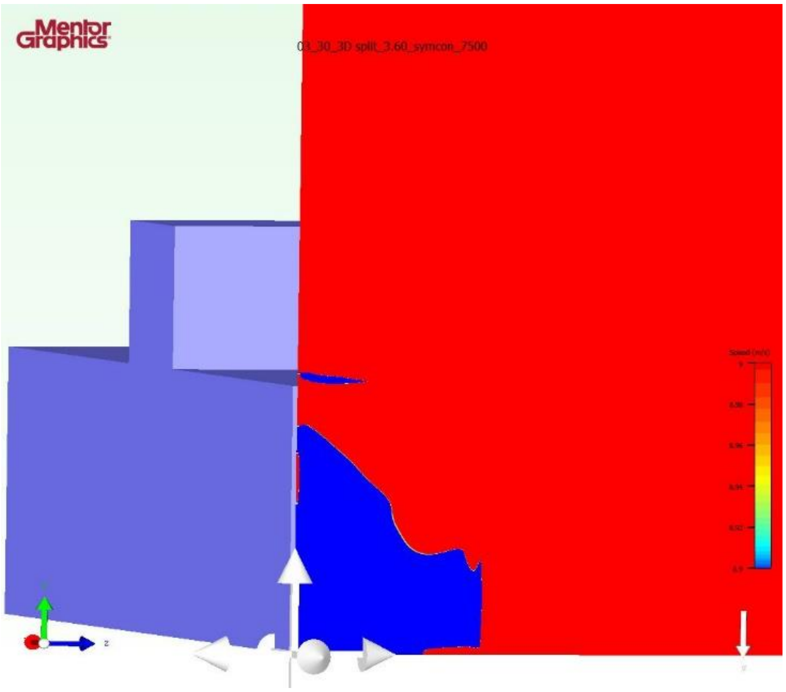

Figure 12 Lower wall at 0.5m from MDC

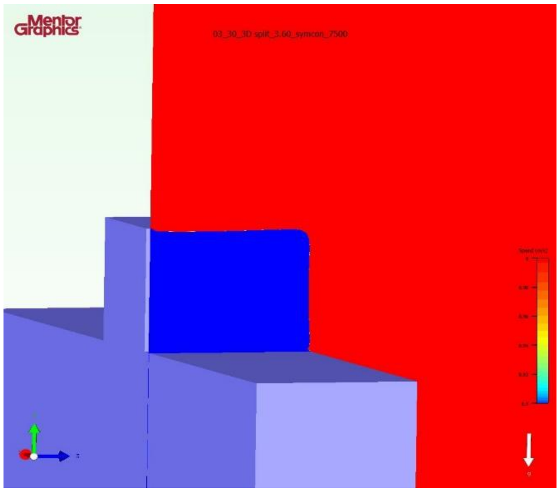

Figure 13 Upper wall at 0.5m from MDC

5.1.2 Velocity profile at 2m from MDC

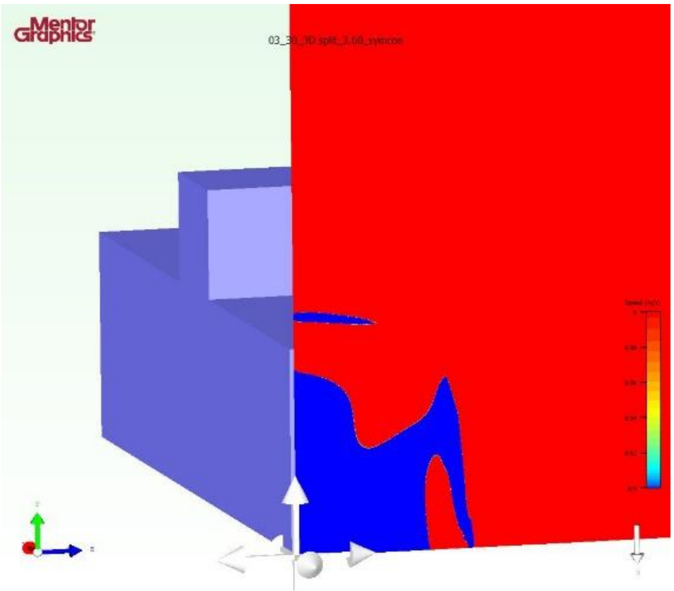

Figure 14 Lower wall at 2m from MDC

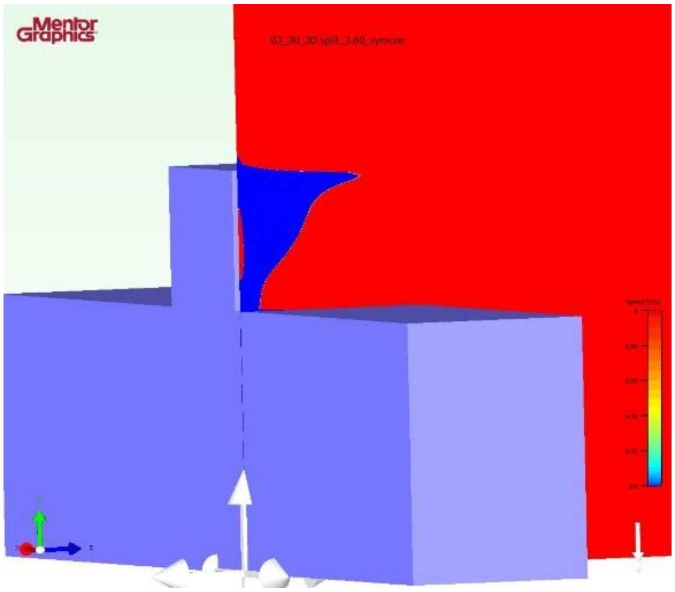

Figure 15 Upper wall at 2m from MDC

5.1.3 Velocity profile at 3.5m from MDC

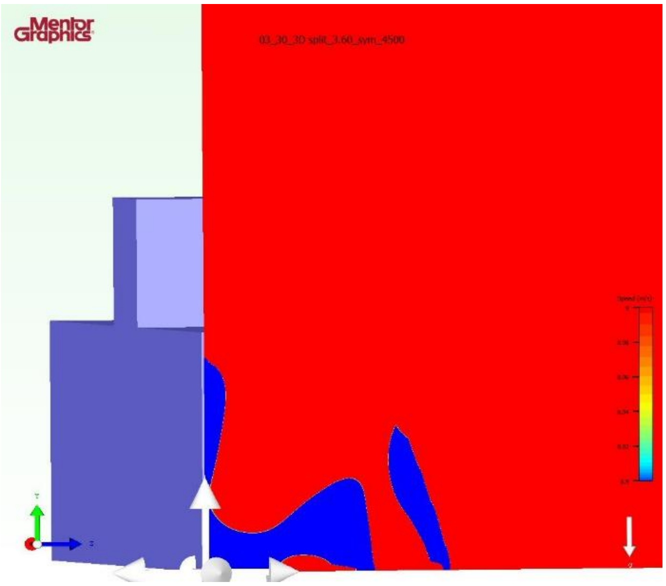

Figure 16 Lower wall at 3.5m from MDC

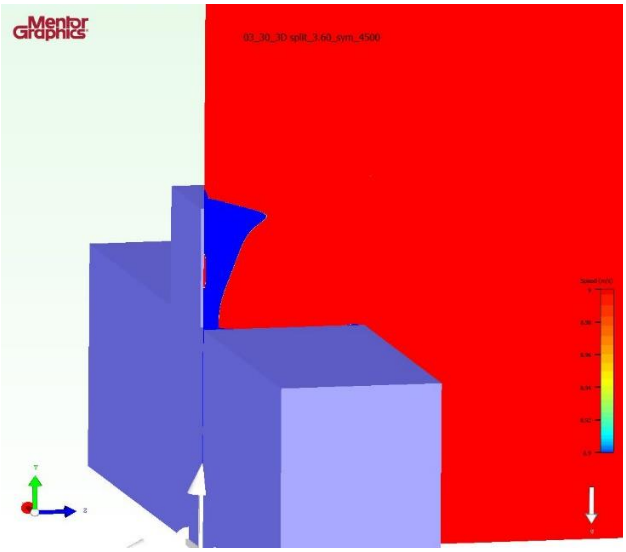

Figure 17 Upper wall at 3.5m from MDC

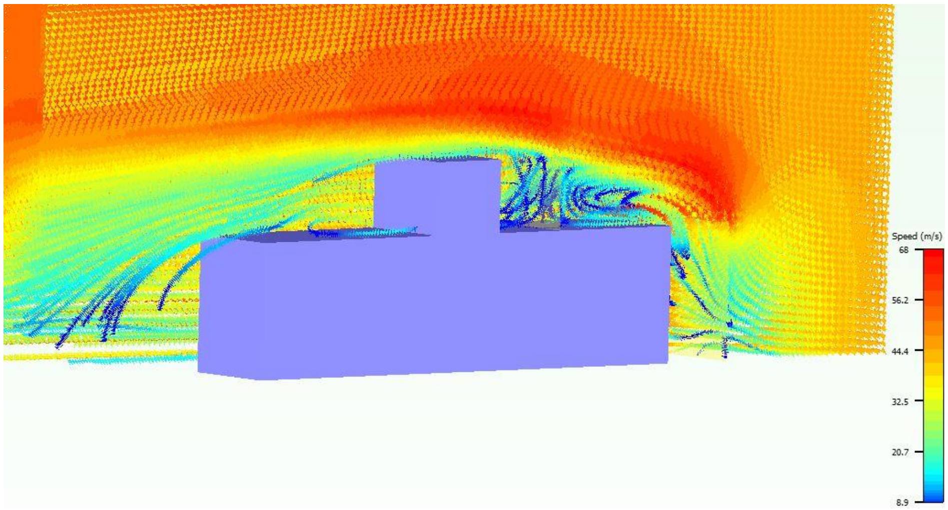

Figure 18 Particle track for 5% void volume fraction of windbreaker

5.2 For 10% void volume fraction

Table 5 10% void volume fraction

Scenario		1	2	3	0	4	5	6
Distance from MDC (m)		0.5	1	1.5	2	2.5	3	3.5
Lower Region	Mean Speed (m/s)	11.90	7.10	5.62	6.32	10.18	11.82	11.49
	Mean Pressure (Pa)	-1165	-1150	-1144	-1130	-1156	-1168	-1199
Upper Region	Mean Speed (m/s)	5.98	5.39	6.73	9.54	10.13	11.70	11.86
	Mean Pressure (Pa)	-845	-866	-866	-831	-799	-788	-806

For the lower region, best results are observed when the windbreaker is placed at 1.5 m from MDC and for upper region, it is 1 m from MDC.

5.3 For 20% void volume fraction.

Table 6 20% void volume fraction

Scenario		1	2	3	0	4	5	6
Distance from MDC (m)		0.5	1	1.5	2	2.5	3	3.5
Lower Region	Mean Speed (m/s)	21.27	10.58	7.61	5.69	4.68	5.05	7.05
	Mean Pressure (Pa)	-431	-690	-746	-775	-797	-807	-842
Upper Region	Mean Speed (m/s)	8.11	7.38	5.28	4.57	6.61	8.68	10.3
	Mean Pressure (Pa)	-523	-591	-598	-592	-579	-556	-524

It is observed that for the lower region least wind speed is attained when the windbreaker is placed at 3 m from MDC and for the upper region least wind speed is attained at 2 m from MDC.

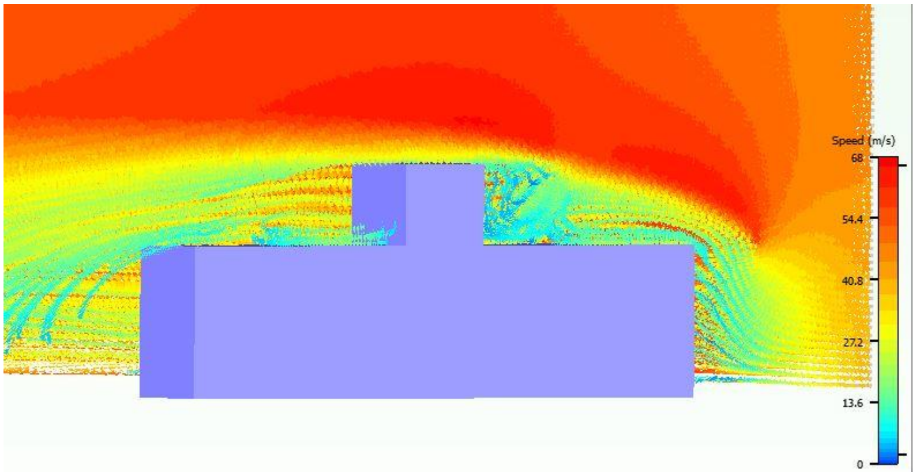

Figure 19 Particle track for 20% void volume fraction of windbreaker

5.4 For 30% void volume fraction

Table 7 30% void volume fraction

Scenario		1	2	3	0	4	5	6
Distance from MDC (m)		0.5	1	1.5	2	2.5	3	3.5
Lower Region	Mean Speed (m/s)	27.2	18.15	11.80	8.52	7.61	6.26	5.07
	Mean Pressure (Pa)	336	-132	-312	-361	-397	-424	-461
Upper Region	Mean Speed (m/s)	6.74	9.50	9.10	7.57	5.86	4.72	4.89
	Mean Pressure (Pa)	160	-375	-408	-422	-422	-416	-418

It is observed that for lower region, least wind speed is attained at 3.5 m from the MDC and for the upper region, least wind speed is at 3 m from MDC.

5.5 For 40% void volume fraction.

Table 8 40% void volume fraction

Scenario		1	2	3	0	4	5	6
Distance from MDC (m)		0.5	1	1.5	2	2.5	3	3.5
Lower Region	Mean Speed (m/s)	26.46	23.62	18.21	14.38	11.58	11.14	7.83
	Mean Pressure (Pa)	454	410	368	91	-71	-68	-38
Upper Region	Mean Speed (m/s)	14.98	8.33	12.23	12.18	12.36	11.96	10.44
	Mean Pressure (Pa)	-322	-267	-304	-299	-310	-316	-322

It is observed that for lower region, least wind speed is attained at 3.5 m from the MDC and for the upper region, least wind speed is at 3 m from MDC.

5.6 Plot of Normalized Speed vs Windbreaker distance

The speeds determined at both upper and lower regions of the MDC were normalized by dividing it by a factor, 9 (maximum acceptable speed). All the points lying within the shaded region are acceptable configurations.

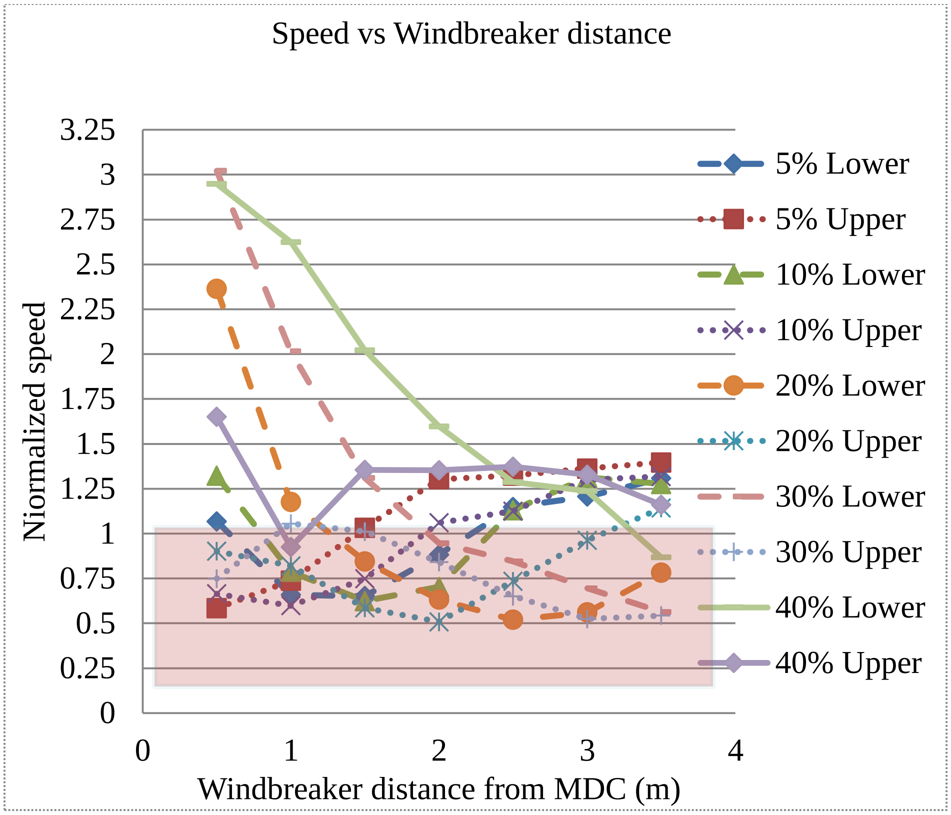

Figure 20 Graph of normalized speed vs windbreaker distance from MDC

From the above plot the acceptable configurations for the windbreaker where wind speed is less than 9m/s, were

- 5% at 1m
- 10% at 1m and 1.5m
- 20% at 1.5m, 2m, 2.5m and 3m
- 30% at 2m, 2.5m, 3m and 3.5m

5.6 Conclusion

The following conclusions can be drawn from this analysis:

- The steady state analysis was conducted and the flow of the air across the MDC can be analyzed for various parameters like speed, pressure, turbulence.
- There was a substantial reduction in wind speed due to the presence of the windbreakers.
- Wind speed reduction below the desired value was obtained at various distances and void volume fractions for both the upper and lower regions.
- Optimum configuration would be when the windbreaker with 30% void volume fraction was placed at a distance of 3.5 m from the MDC. This would be the optimum choice over the other options because building a windbreaker with 30% void volume fraction would be cheaper and less material would be required. Also having it at 3.5m away from the MDC would improve workability and provide ease of access around the MDC.

Chapter 6

SCOPE AND FUTURE WORK

- Transient analysis to conduct in-depth analysis of the low field. This would help to gain a better understanding of the flow around the MDC.
- Structural design of the windbreaker and also choice of material.
- 3D stress analysis of the windbreaker.
- Varying other parameters of the windbreaker such as height and orientation and study the combined effects of varying all parameters.

References

[1] [Online] Available: http://www.spc.noaa.gov/faq/tornado/beaufort.html

[2] [Online] Available: http://www.nhc.noaa.gov/aboutsshws.php

[3] “Modular Data Center”, Wikipedia, [Online]. Available: http://en.wikipedia.org/wiki/Modular_data_center

[4] [Online] Available: http://www.enterprisetech.com/2013/09/27/ebay-juices-utah-modular-data-center-fuel-cells-waste-heat/

[5] Yahoo! Compute Coop (YCC): A Next-Generation Passive Cooling Design for Data Centers, AD Robison, Chris Page, Bob Lytle

[6] [Online] Available: http://www.datacenterdynamics.com/design-strategy/yahoos-compute-coop-is-not-for-chickens/93474.fullarticle

[7] [Online] Available: http://extension.psu.edu/plants/plasticulture/production-details/windbreaks

[8] [Online] Available: http://www.bdi.aero/

Biographical Information

Prasad Pramod Revankar was born in Goa, India. He received his Bachelor's degree in Mechanical Engineering from Goa University, India in 2012. He completed his Master of Science degree in Mechanical Engineering at the University of Texas at Arlington in May 2015.

His primary research areas include Computational Fluid Dynamics. He has worked on the CFD analysis of the flow past a Modular Data Centers. The project he has worked on was designing a windbreaker to reduce high speed wind loading on modular data centers.

He joined the EMNSPC research team under Dr. Dereje Agonafer in November, 2014.