CFD BASED DESIGN AND MODELLING OF WIND FENCE TO MITIGATE HIGH-SPEED WIND LOADING ON A MODULAR DATA CENTER

By

DEVI PRASAD GORREPATI

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
December 2015

ACKNOWLEDGEMENTS

I would like to take this opportunity to express my deep gratitude to my supervising professor Dr. Dereje Agonafer for his immense encouragement, support and guidance throughout the course of my research at The University of Texas at Arlington. Also, I extend my gratitude to Mr. Thomas Craft from CommScope for his regular insights to enhance this work. The invaluable advice and the timely support provided by both Dr. Agonafer and Mr. Thomas have been very instrumental in the successful completion of this research.

I would like to thank Dr. Haji-Sheikh and Dr. Kent Lawrence for taking their time to serve on my thesis committee. Also, I would like to thank Betsegaw Gebrehiwot, Prasad Revankar, Framroz Bharucha, Kanan Pujara and other lab mates for their invaluable support while working at the EMNSPC labs.

I am very much obliged to Ms. Sally Thompson and Ms. Debi Barton for helping me with the administrative work and other educational matters at all times. They have been extremely kind and supportive at whatever point I required their assistance.

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.

November 9, 2015

ABSTRACT

CFD BASED DESIGN AND MODELLING OF WIND FENCE TO MITIGATE HIGH-SPEED WIND LOADING ON A MODULAR DATA CENTER

Devi Prasad Gorrepati, MS

The University of Texas at Arlington, 2015

Supervising Professor: Dereje Agonafer

A Modular Data Center (MDC) is a portable method of deploying a data center’s capacity. As an alternative to the traditional data center, an MDC can be placed anywhere data capacity is required. The purpose of this study is to reduce the damage or loss of performance caused to the data centers that use free cooling, by mitigating high-speed winds. The Modular Data centers which use free cooling and that are located in open regions are subjected to various environmental risks such as very high-speed winds. As this wind blows over these data centers, the pressure difference generated within and outside the enclosure can have a drastic effect on the free cooling. Therefore, by using a wind fence which basically acts as a barrier to the upstream wind and reduces the mean velocity of air downstream of the wind fence, we reduce the pressure difference created and also the wind induced loading on the objects situated behind the fence.

Although wind fences are used in many agricultural and farming practices, their usage pertaining to MDCs is very limited. The challenge is to reduce wind speed from 100 mph to 10 mph. This has been achieved by iteratively designing and analyzing a wind fence

using CFD simulations to come up with a few wind fence options that have defined properties such as height, perforation and location (distance from the inlet of MDC) of the wind fence.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF TABLES

Chapter 1

Introduction

A Data Center is a facility which stores 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. Since the IT equipment within the data center produces a large amount of heat this also requires an efficient cooling system. To conclude, 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.

The high-speed winds that usually occur in wide open spaces can cause significant damage and loss of cooling performance of the data center. As the wind blows over the data center there is a buildup of pressure difference within and outside of the data center, this affects the free cooling by restricting the airflow through the data center.

Wind speeds are classified according to the Beaufort Scale and according to this scale, wind speeds in excess of 73.6 mph (32.924 m/s) are classified as hurricanes. [1] Hurricanes are further classified by the Saffir-Simpson Hurricane Wind Scale. It is rated from 1 to 5 based on the sustained wind speed of the hurricane. 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, and 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 from the table that wind speeds exceeding 74 mph can cause a substantial amount of damage to the building. The aim of this computational investigation is to design a windbreaker for reducing Category 2 hurricane wind of 44.7 m/s (100 mph) to 4.5 m/s (10 mph) average velocity in the direction normal to the 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. [3]

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 the 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. [4]

Modular data centers are designed for rapid deployment, energy efficiency, and high-density computing to deliver data center capacity at a lower cost than traditional construction methods, and significantly reduce the construction time from years to a matter of months. [5] Hence, modular data centers are a better alternative as compared to the traditional data centers.

Figure 1: CommScope's DCU20 modular data center [1]

The CommScope's DCU20 Modular data center (MDC) has room for 20 racks of IT gear and can have up to 35 kilowatts of output power per rack. It uses outside air cooling and has chilled water evaporative cooling when the outside temperature gets too high. It has average annual Power Usage Effectiveness (PUE) of 1.03 – 1.06.

1.1.2 Yahoo's Chicken Coop Data Center

Yahoo's chicken coop data centers are pretty similar to the traditional chicken coop. This design has a raised floor with openings such that the air from below the floor is drawn up through the coop keeping the chickens cool. This movement of air also removes the excessive moisture accumulated in the enclosure. A similar principle has been used in designing the data center, as this concept leads to ventilating the data center using a full-

---

[1] http://www.commscope.com/catalog/enterprise/product_details.aspx?id=49244

roof cupola system, which proves to be a great way to cool computing equipment. [6] 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 are 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. [7]

The Yahoo Chicken Coop design has three different cooling modes:

1. <u>Unconditioned Outside Air Cooling:</u> When the 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.
2. <u>Outside Air Tempered by Evaporative Cooling:</u> 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.

3. 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. [7]

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. [8]

Figure 2: Yahoo Compute Coop Data Center Facility

Chapter 2

Wind fences

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 a sheltered zone. As the wind blows against a windbreak, there is a high-pressure zone created on the windward side 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 the wind on a structure are 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. [6]

Figure 3: Wind fences used at airports[2]

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 following figure shows a reduction in wind speed behind a windbreak. [9]

---

[2] http://www.bdi.aero/assets/img/featured/home/home-uk.jpg

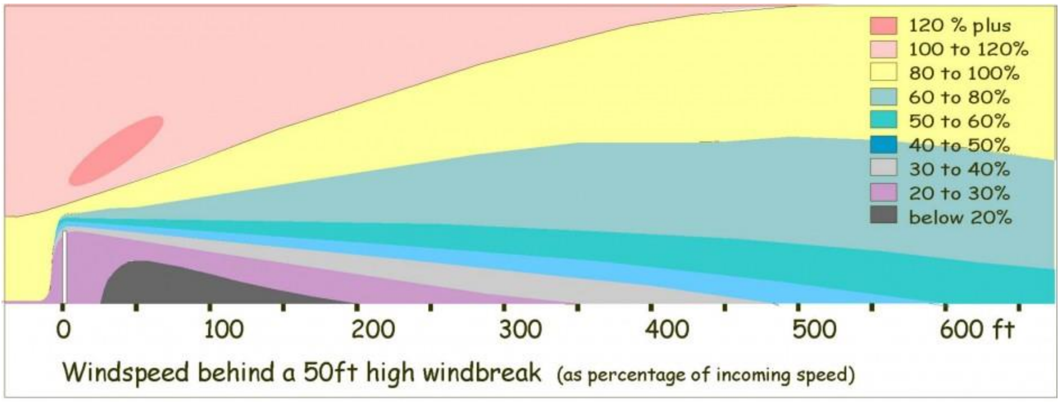


Figure 4: Wind speed behind a wind fence as percentage of incoming speed

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 the height of the windbreak, a distance of the windbreak from area to be sheltered, the density of the windbreak, length, and orientation. [10]

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. [11]

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 on the Leeward side. Therefore, the height H of the windbreak chosen equal to the height of the data center would be sufficient.

2.1.2 Effect of Distance

The 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 wind fence 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, which is reduced wind speed before the inlet and outlet of the data center. Therefore, it is necessary that the wind fence 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 or perforation, 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 perforation of the windbreak increases, the amount of air passing through the windbreak increases, moderating the low pressure and large wake vortices 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.

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 an 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 the 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. [11]

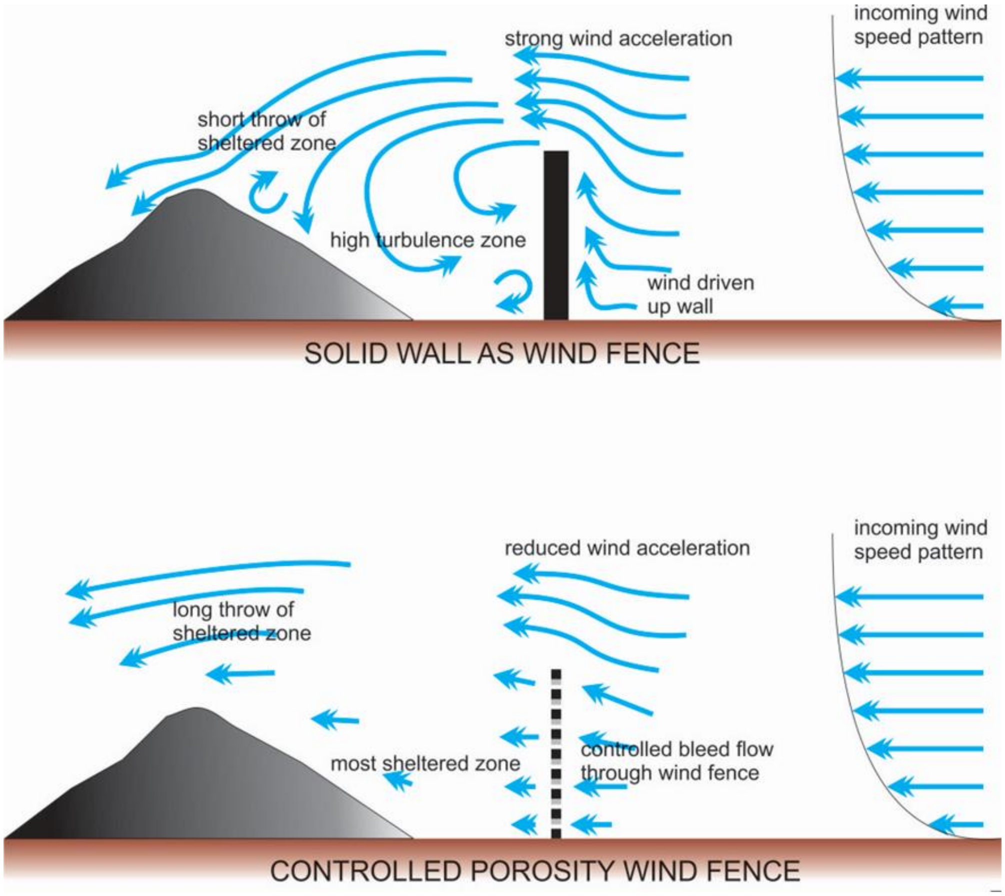


Figure 5: Effect of density

In the case of data centers, the effect of perforation is studied to determine the optimum amount of perforation required in the windbreak in order to obtain reduced velocity at the inlet and outlet of the data center. A solid wall with no perforation creates a negative or very low pressure right in front of the inlet which would not suit for this purpose since air would flow out of the inlet. Hence, some perforation is necessary for the wind fence to create a positive pressure gradient at the inlet.

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 are 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 the 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. [11]

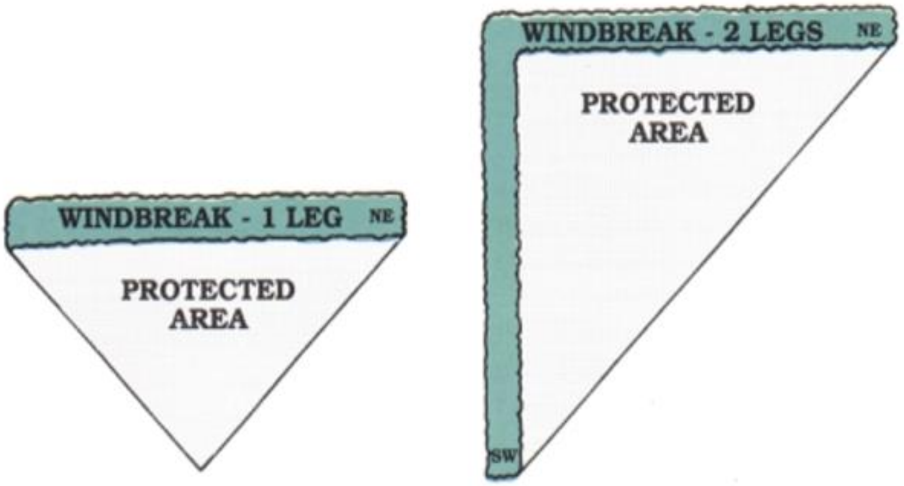


Figure 6: Protected area based on windbreak orientation[3]

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. [11]

In the case of data centers having a windbreak at least as wide as the modular data centers will provide the required protection from the wind that is directly impinging.

---

[3] http://nac.unl.edu/documents/morepublications/ec1763.pdf

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 an 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 Wind fence 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 the new or existing equipment. [12]

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, the 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. [6]

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, the conservation equations are discretized by subdivision 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 is 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:

$$T= \frac{C0T0+C1\mathrm{T}1+C2T2+\cdots CnTn+S}{C0+C1+C2+\cdots Cn}$$

Where T0 represents temperature value in the initial cell, T1, T2…, Tn is valued 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 is solved.

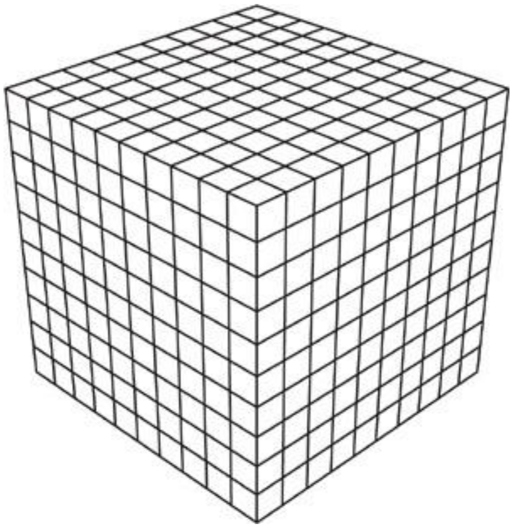


Figure 7: Representation of a 3D grid

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 is considered to be turbulent while fluids with low Reynolds number are considered laminar. FloTHERM 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 a 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 the nearest wall.

D = (|∇∅|2 +2∅)1/2

L= D-|∇∅|

Where |∇∅| = -1 and ∅ = 0 (which is boundary condition at the wall)

∅ is the dependent variable. [13]

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. [13] 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 the minimum and a 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, the addition of a large number grid cells has no further effect on the solution. FloTHERM 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, 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

3.5.1 Cuboid

This is the most basic smart part in FloTHERM. 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 a 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 perforation can be defined from the resistance library depending on the requirement. In this model, the resistance smart part is used to represent a porous wind fence. Different values of void volume fraction are given to the resistance to simulate a wind fence with varying void volume fraction. In other applications, this smart part is used for modeling filters and vents.

3.5.3 Perforated plate

The perforated plate smart part is used when we know the parameters of the perforated metal used. It would be impractical to model most grilles, screens or any other kind of planar perforated region in detail. The number of grid cells would simply be excessive. So FloTHERM gives the user the ability to model them as a region of pressure loss without modeling the perforations in detail.

3.5.4 Enclosure

The enclosure smart part is a hollow part which can be 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 of 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.

3.5.5 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.

3.5.6 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.

3.5.7 Region

This smart part is used for two main purposes. The first one is to create a refined mesh and the other is to define an 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 grid constraints and Localize feature. Another use of the region is that it can be used in post-processing, obtaining the fluid properties from the area of interest. The values of different parameters like, velocity, pressure, speed, and the temperature can be determined within a particular region.

3.5.8 Command Center

The command center in FloTHERM 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 changes have 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.

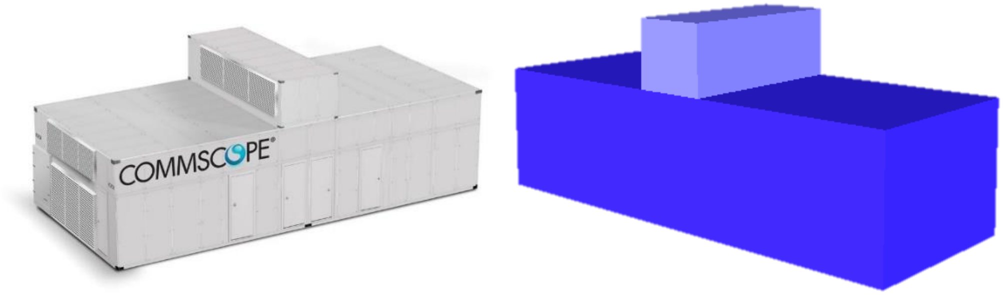
Chapter 4

Description of the CFD model

The Computational Fluid Dynamics model is of a chicken coop modular data center and the two wind fences, which are located at a certain distance in front of the inlet and exhaust of the Modular Data Center (MDC). Only the external structure of the MDC is modeled 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 modeled is a chicken coop modular data center, whose structure is similar to the DCU20. 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 inlet for the ambient air entering the data center, which is used for cooling. 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.



(a) (b)

Figure 8: (a) CommScope DCU20 MDC (b) Half-symmetric 3D model of DCU20 MDC

4.2 Description of the Wind fence

The wind fences used for this study are of two kinds, one is a Single wall and the other is a split wall type. In the case of the single wall, one wind fence as tall as the total MDC is erected in front of the inlet. Whereas, in the case of a split wall, one wall is placed on the ground which is as tall as the IT pod and the second one is placed on top of the IT pod which is as tall as the chicken coop (exhaust). The wind fences are modeled using the perforated plate smart part in FloTHERM. This allows the wind fence to be modeled as a porous body with varying perforations.

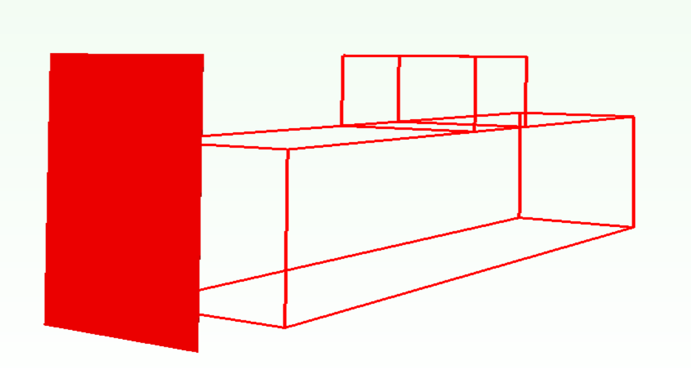


Figure 9: Single Wall configuration schematic

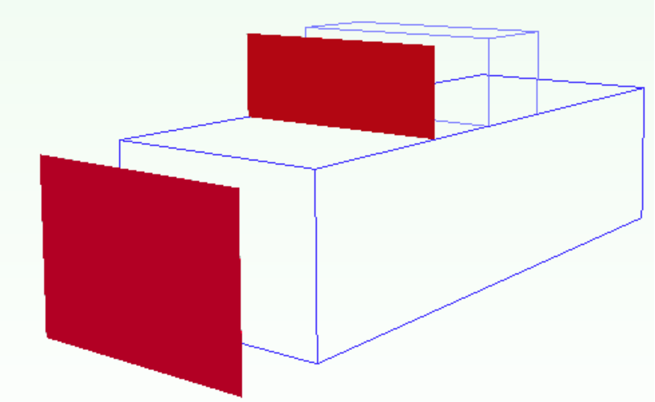


Figure 10: Split Wall configuration schematic

4.3 Description of the CFD Model

4.3.1 Dimensions of the Model

Table 2: Dimensions of the single wall model

	Length(mm)	Width(mm)	Height(mm)
Lower portion of MDC	14160	3150	3740
Upper portion of MDC	2200	3150	2000
Wind fence	100	3150	5740

Table 3: Dimensions of the split wall model

	Length(mm)	Width(mm)	Height(mm)
Lower portion of MDC	14160	3150	3740
Upper portion of MDC	2200	3150	2000
Lower portion of wind fence	100	3150	3740
Upper portion of wind fence	100	3150	2000

4.3.2 Model Setup

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

- Type of Solution: Flow Only
- Dimensionality: 3-Dimensional
- Gravity was set to Normal, acting in –Y direction with Automatic value (does not affect even when turned off since there is no temperature change)
- The flow type was set to Turbulent and the LVEL K-Epsilon Turbulence Model was used.
- Ambient Pressure: 1 Atm
- Default Radiant and Ambient Temperature: 35 ˚C (no significance for the analysis)

- Wind Velocity was set to 44.704 m/s (100 mph) 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 Domain and Boundary conditions

The size of the domain to properly model the wind loading has been referred from Best practices for Architectural (AEC) engineering analysis by Autodesk. [14]

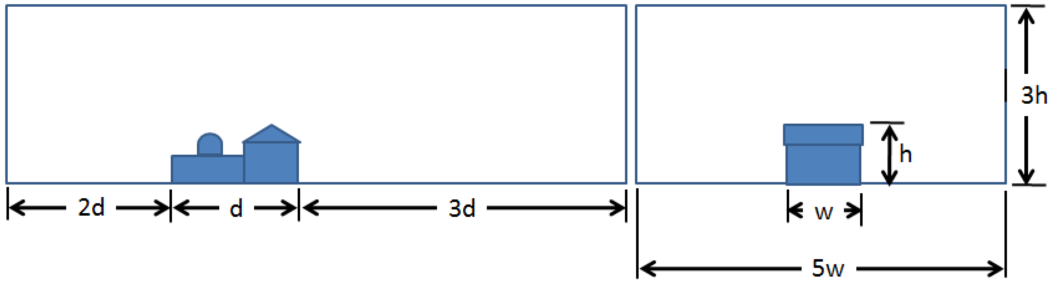


Figure 11: Recommended relative dimensions for environment geometry

The boundary conditions for the wind load analysis were also taken from the Autodesk website.

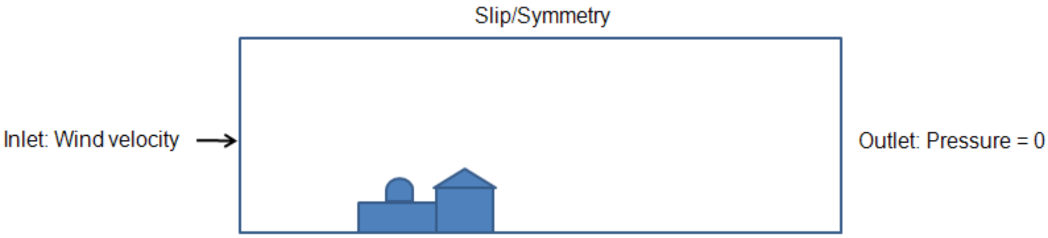


Figure 12: Recommended boundary conditions for wind loading analysis[4]

- To define the inlet of the air volume, assign the wind speed as a Velocity boundary condition.
- To define the outlet of the air volume, assign Static Gage Pressure = 0.
- If the air region simulates a free-space environment (not a wind-tunnel), assign Slip/Symmetry to the top and sides of the region. (Do not specify a condition on the ground plane, because the air does not physically move along the ground plane.) [14]

The boundary conditions selected for the model are as follows, the ground or Y-low face of the solution domain has no slip condition or wall. The Z-low face has the symmetric boundary condition. This is done to take advantage of the symmetry of the system and saving the computational time by a significant amount since the number of grid cells is reduced to half. The inlet has velocity boundary condition and the outlet has pressure boundary condition. The rest of the sides Z-high and Y-high have slip or symmetry boundary condition.

---

[4] http://help.autodesk.com/view/SCDSE/2015/ENU/?guid=GUID-2BF3C8A5-6D70-4F6A-A792-95A27C605E3D

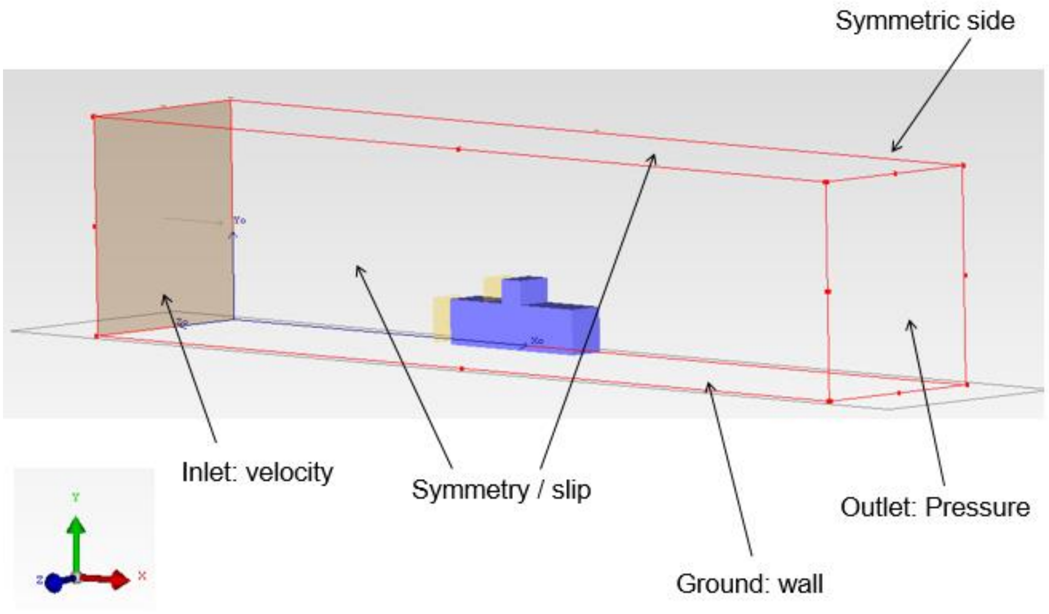


Figure 13: Boundary conditions for the CFD model

4.3.4 Meshing

A quadrilateral mesh was generated to represent the fluid domain. The mesh has been created in a way such that it is very fine where there is a large amount of change in pressure and other flow properties and coarser in regions that does not affect the flow characteristics. The total mesh size is about 3.7 million cells and the maximum aspect ratio is 1.96. Since the model is symmetric, a half model has been created and meshed. The principle meshing methodology used is the same for both single and split wall models.

Several regions have been created to limit or constraint the mesh to a definite size in that region. While performing grid independence study the entire mesh including the regions set to make the mesh fine were altered simultaneously. A cutout function has been used within the MDC to exclude those cells from the computation.

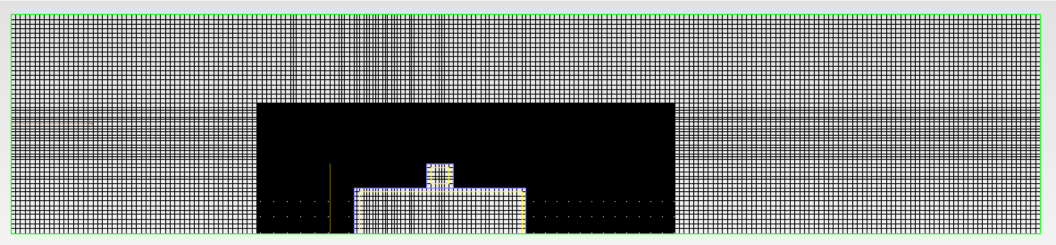
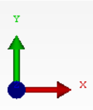


Figure 14: Mesh of the single wall CFD model



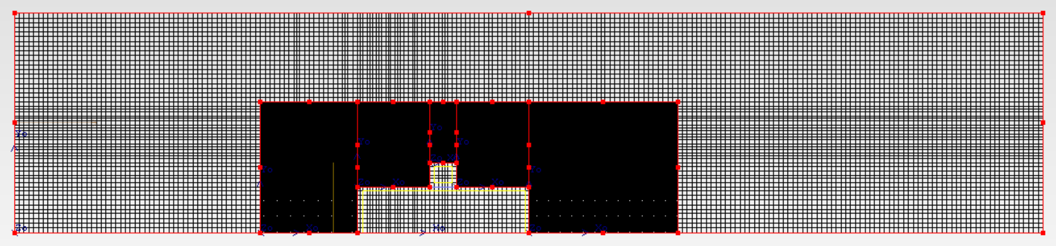


Figure 15: Regions considered for finer meshing

The regions considered for obtaining the fluid properties are shown below:

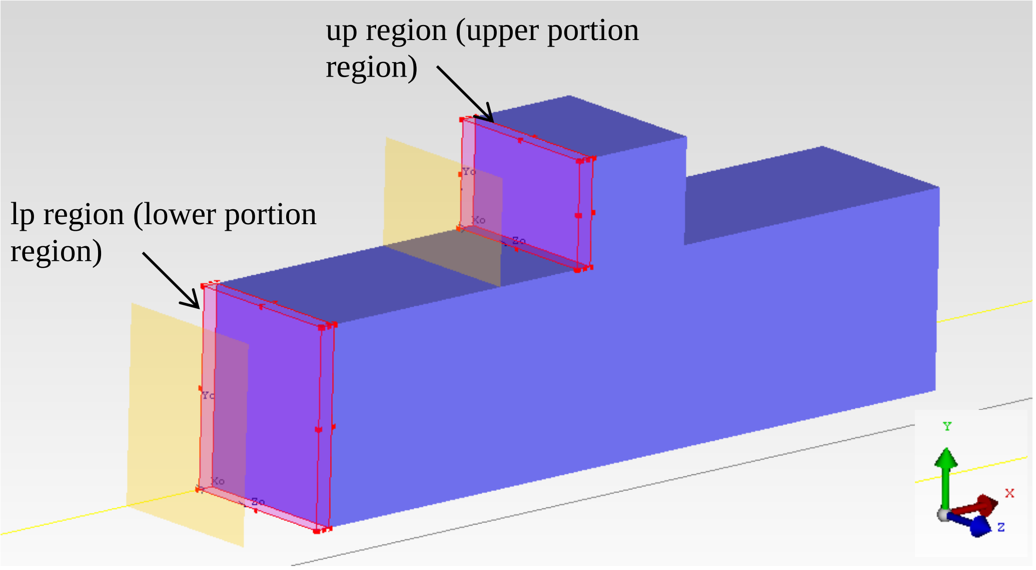


Figure 16: Regions considered for obtaining fluid properties in the CFD model

4.4 Mesh Sensitivity Analysis

Mesh sensitivity analysis has been 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, the input variable i.e., the number grid cells are varied and its effect on the output variables like speed and pressure are studied.

The following mesh sensitivity analysis was carried out on the single wall and split-wall model,

Table 4: Single wall mesh sensitivity analysis

Total Grid Size	lp region-Mean Region X-Velocity (m/s)
549684	1.58
1231338	2.47
1731516	2.14
2342579	2.14
3673456	2.41
4584522	2.39
5856943	2.19
6695220	2.43

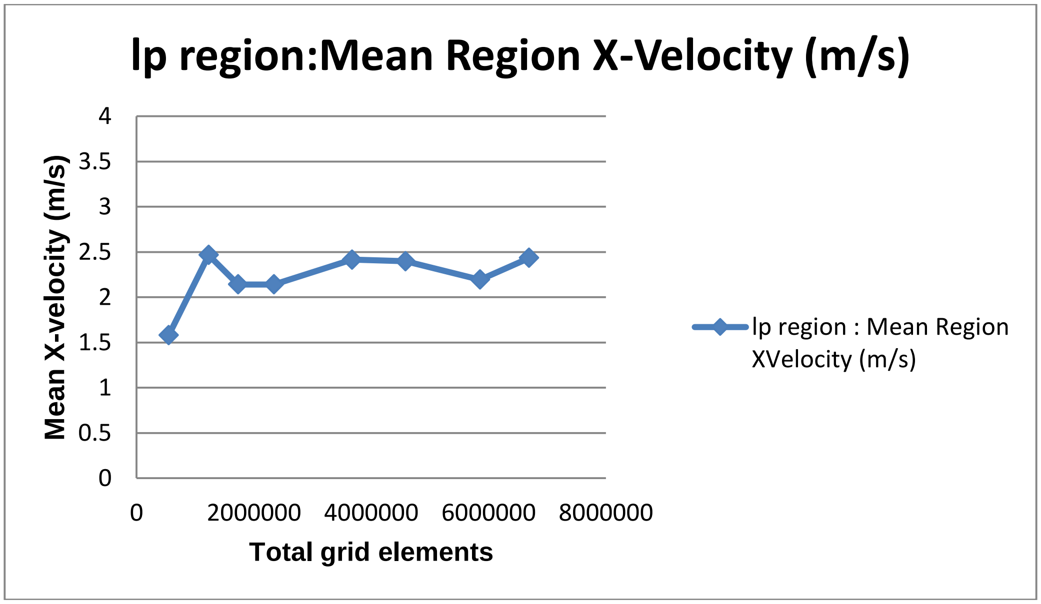


Figure 17: Mesh sensitivity analysis for single wall

Grid independence achieved at 3.7 million cells with a maximum aspect ratio of 1.9.

Note: At 5.8 million cells there seems to be an anomaly, hence that was ignored for the mesh sensitivity calculation above.

Table 5: Split wall mesh sensitivity analysis

Total Grid Size	lp region : Mean Region XVelocity (m/s)
316501	1.886
549684	1.543
741357	1.448
1231338	2.398
1549510	2.089
1731516	2.096
2364484	2.104
3673456	2.367
4584522	2.348
6695220	2.38

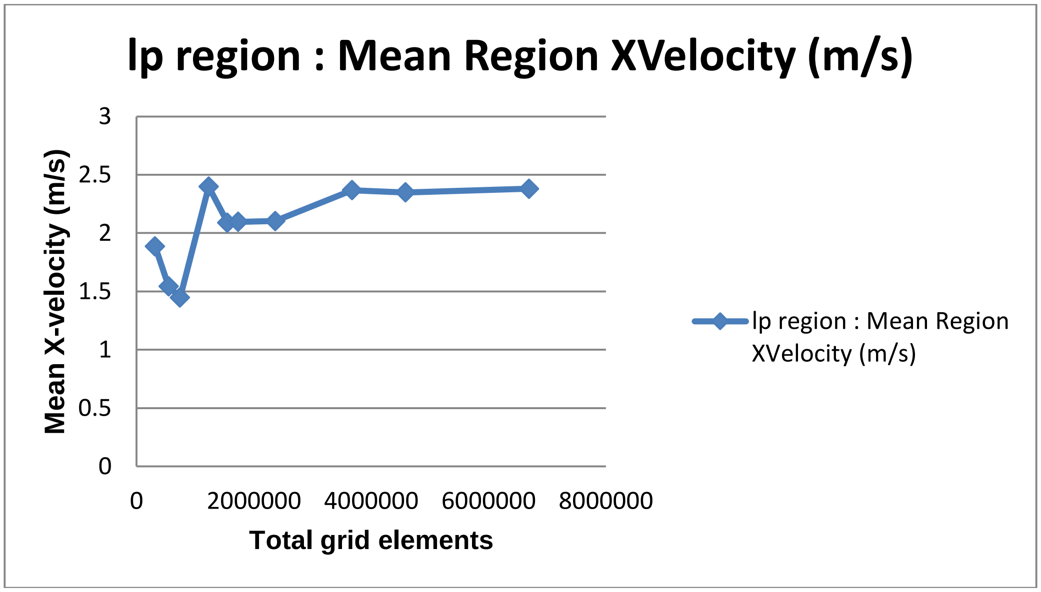


Figure 18: Mesh sensitivity analysis for split wall

Grid independence for the split wall was achieved at 3.6 million cells with a maximum aspect ratio of 1.9.

4.5 Scenarios Considered

The two main parameters that were considered in this investigation were a distance of the wind fences from the MDC and perforation of the wind fences. The initial case for both single and split walls were considered at a distance of 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. Perforation was varied from 20% to 50% with an increment of 10% i.e., 20%, 30%, 40% and 50%

Google's experiments showed that fences with lower porosity (more blockage) tend to reduce aerodynamic loads on heliostats, however, the difference is not significant and probably not worth the extra cost. Lower porosity fences require more material to build and more substantial installations since they have to withstand much higher wind loads. The flow visualization tests showed that a solid wall (a 0% porosity fence) was not effective at reducing flow velocity in a heliostat field and that a 40-50% open area (i.e. 50-60% blockage) fence seems to have a good amount of load-reducing capability. [15] Hence, the perforation scenarios were limited to 50% perforation. Further reading about Google's porosity study can be obtained at http://www.google.org/pdfs/google_heliostat_wind_tunnel.pdf

The pictures below show the convention followed for the wall advancement. As wall moves behind from the datum position (2m) it is called negative wall advancement and when the wall moves forward towards the MDC from the datum position (2m) it is called positive wall advancement.

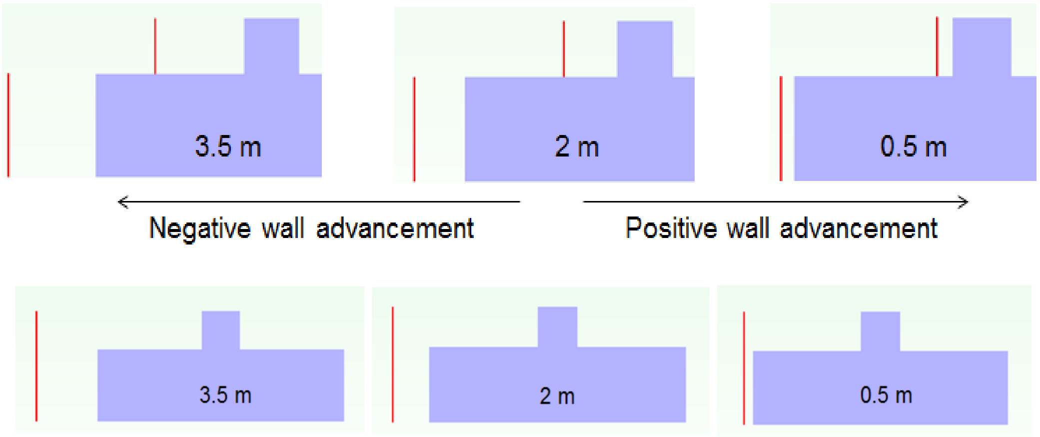


Figure 19: The convention followed for the wall advancement scenarios

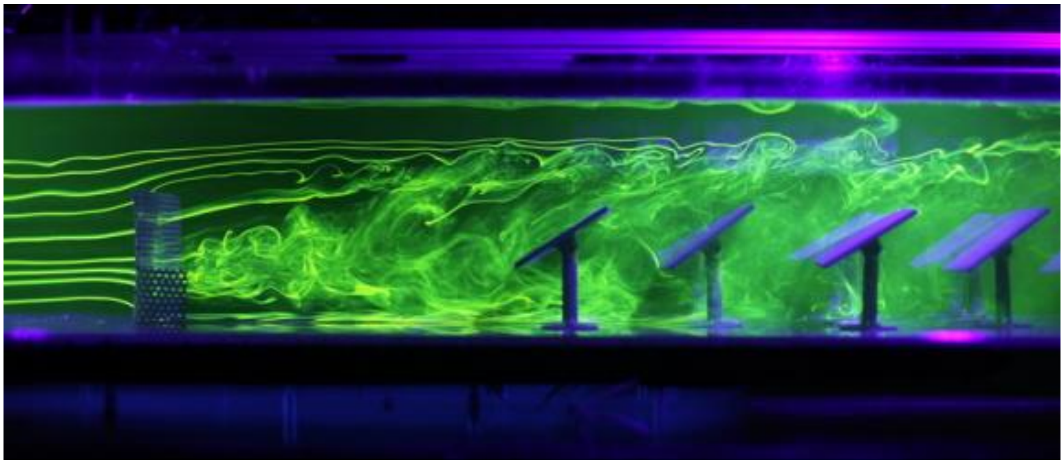


Figure 20: Flow visualization study conducted by google on porous fences[5]

An upstream fence (50% open area), as tall as heliostat overall height, creates disruptive flow immediately, lowering the flow velocity for all heliostats in the field. [16]

---

[5] https://www.google.org/pdfs/google_heliostat_flow_visualization.pdf

Chapter 5

Results and Conclusion

The following results were obtained by varying the distance of the wind fence from the MDC and the perforation of the wind fence. The parameters such as mean x-velocity, mean speed, and mean pressure were obtained from a region which is 1 ft. (0.3 m) away from the MDC.

5.1 Single Wall Results

5.1.1 Single wall X-velocity at the inlet

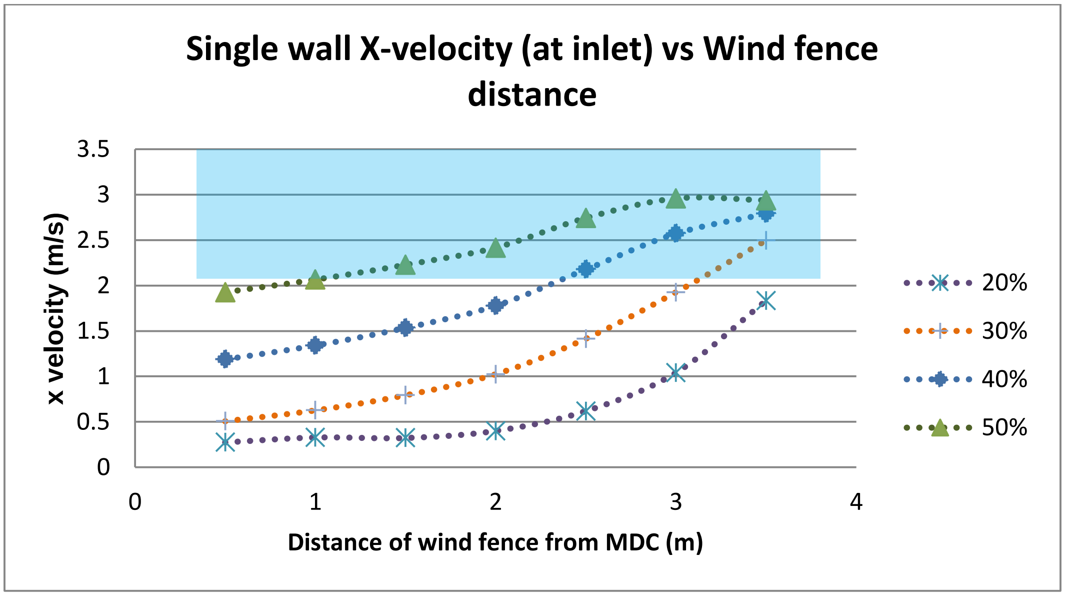


Figure 21: Single wall X-velocity (at inlet) vs Wind fence distance

The x- velocity at the inlet in the single wall case is very low (approx. 6.7 mph) for the perforations considered. Since we intend to supply the air to the inlet we must have a considerable positive x-velocity. Therefore, on considering 2 m/s or 4.5 mph as a considerable positive x-velocity, the region highlighted shows all the cases that suit this condition.

5.1.2 Single wall X-velocity at the exhaust

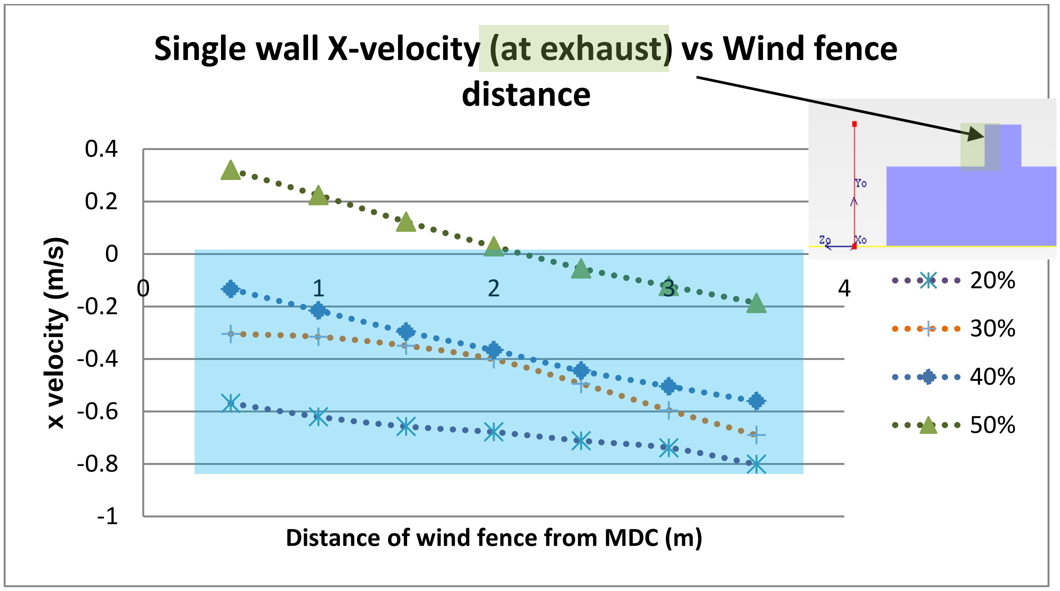


Figure 22: Single wall X-velocity (at exhaust) vs Wind fence distance

The x-velocity at the exhaust in the single wall case is very low (around 1 mph) as shown in the graph above, however having a negative velocity at the exhaust would facilitate more air to flow outward from the exhaust. Therefore, the region highlighted in the graph determines all the fence locations and perforations that facilitate a negative x velocity at the exhaust.

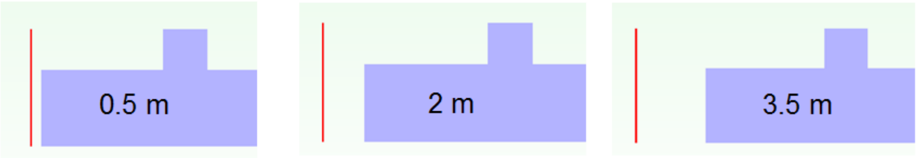


Figure 23: Schematic figure to understand the wall distance from MDC

5.1.3 Single wall Mean Pressure at the inlet

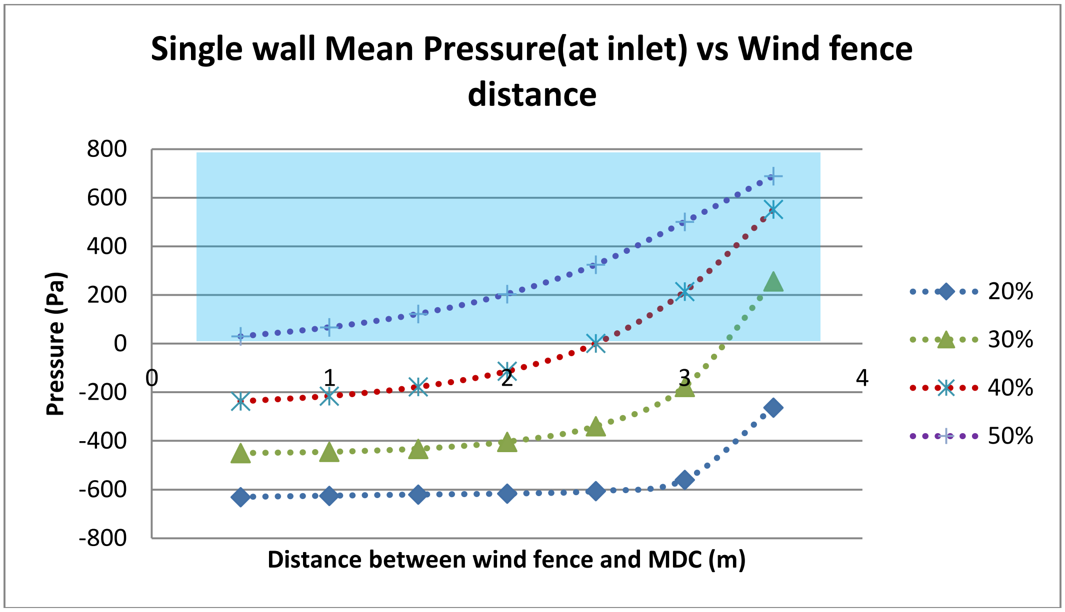


Figure 24: Single wall Pressure (at inlet) vs Wind fence distance

The pressure at the inlet in the single wall case with respect to the wind fence distance is shown above. Since we intend to supply the air to the inlet we must have a positive pressure gradient. Therefore, the region highlighted on the graph with a colored box shows all the cases that provide a positive pressure gradient at the inlet of the MDC.

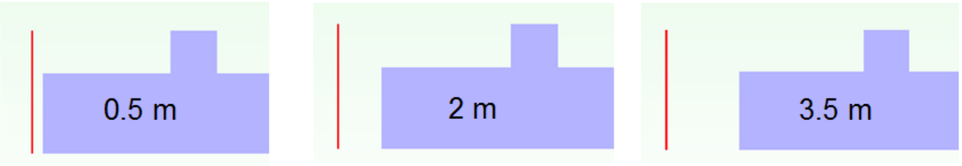


Figure 25: Schematic figure to understand the wall distance from MDC

5.2 Split Wall Results

5.2.1 Split wall X-velocity at the inlet

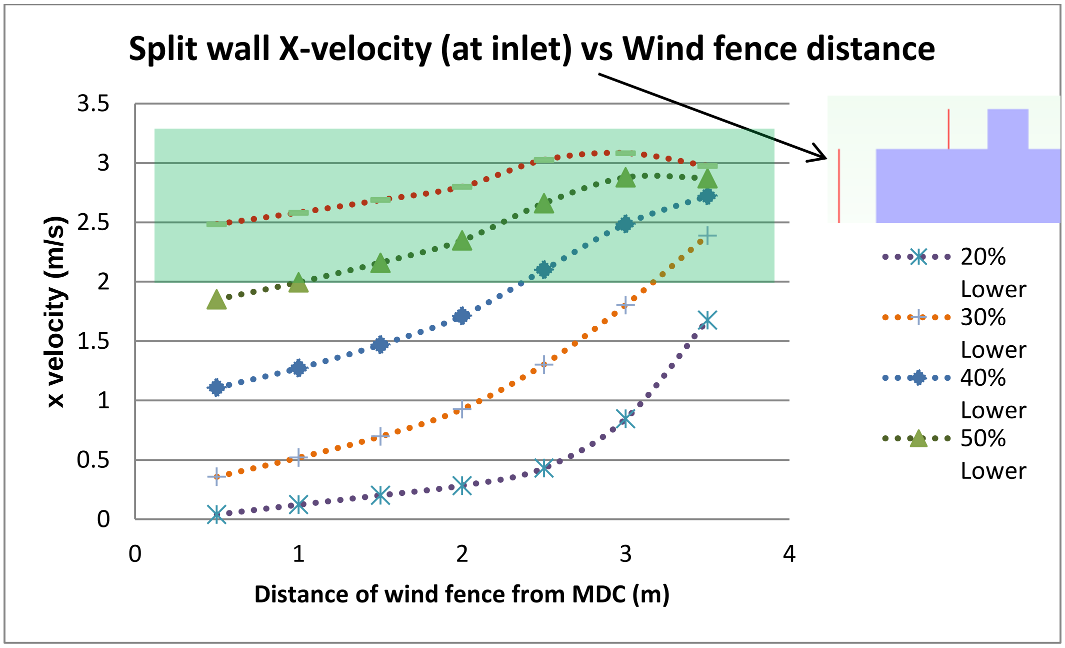


Figure 26: Split wall X velocity (at inlet) vs Wind fence distance

The x- velocity at the inlet in the split wall case is very low (approx. 6.7 mph) as well for the perforations considered. Since we intend to supply the air to the inlet we must have a considerable positive x-velocity. Therefore, on considering 2 m/s or 4.5 mph as a considerable positive x-velocity, the region highlighted shows all the cases that suit this condition.

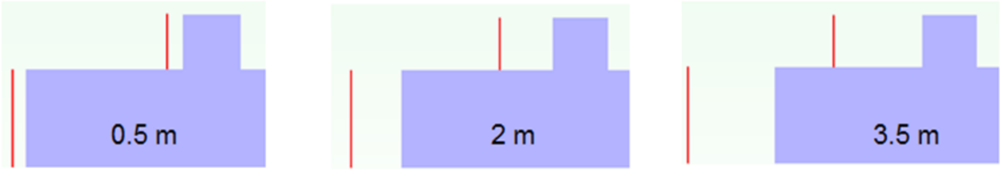


Figure 27: Schematic figure to understand the wall distance from MDC

5.2.2 X-velocity at the exhaust

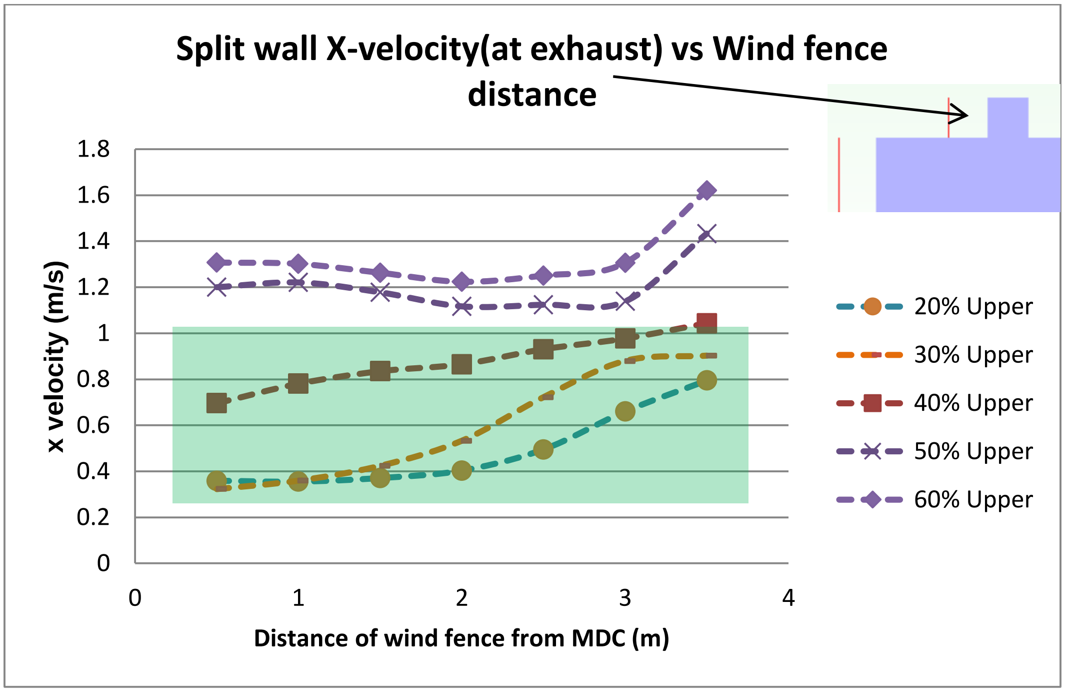


Figure 28: Split wall X velocity (at exhaust) vs Wind fence distance

The x-velocity at the exhaust in the split wall case for various perforations and wind fence locations is as shown in the graph above. On having the wind velocity hitting at the exhaust lower than that at the inlet, more flow through the MDC can be achieved. Therefore considering 1 m/s (2.24 mph) which is less than the lowest inlet velocity i.e., 2m/s (4.5 mph) as beneficial for the flow through MDC, the region highlighted in the graph shows all the cases that suit the condition.

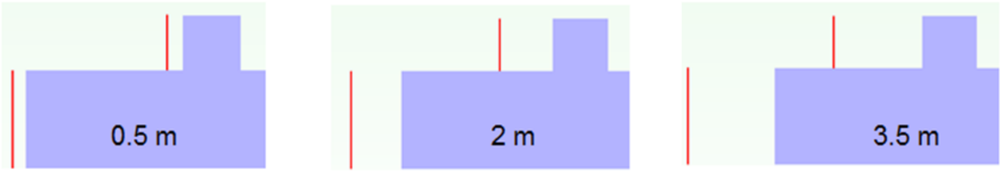


Figure 29: Schematic figure to understand the wall distance from MDC

5.2.3 Split wall Mean Pressure at the inlet

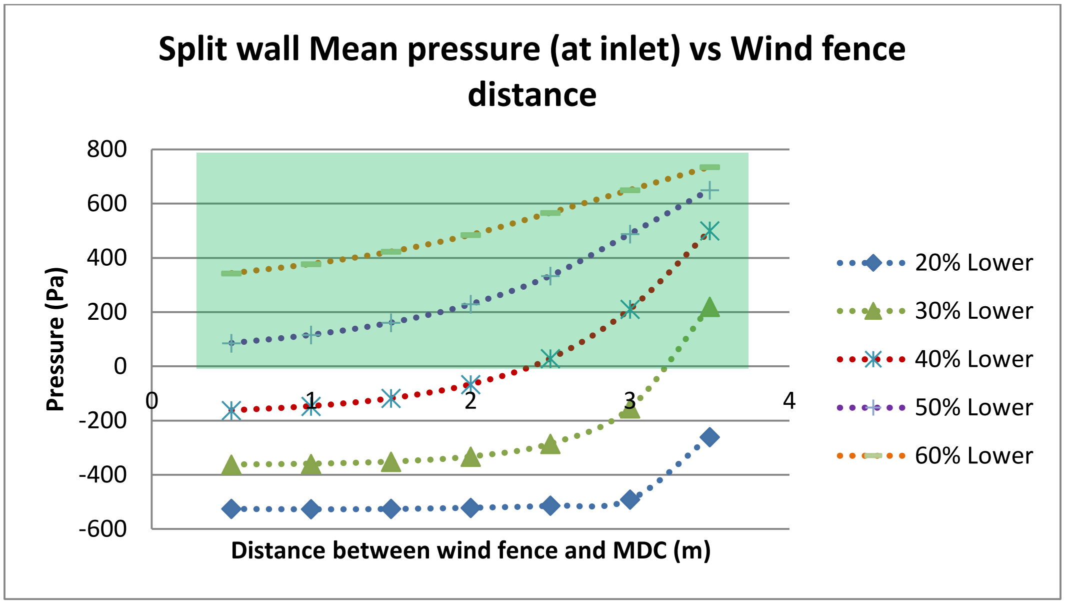


Figure 30: Split wall Mean pressure (at inlet) vs Wind fence distance

The pressure at the inlet in the split wall case with respect to the wind fence distance is shown above. Since we intend to supply the air to the inlet we must have a positive pressure gradient at that location. Therefore, the region highlighted on the graph with a colored box shows all the cases that provide a positive pressure gradient at the inlet of the MDC.

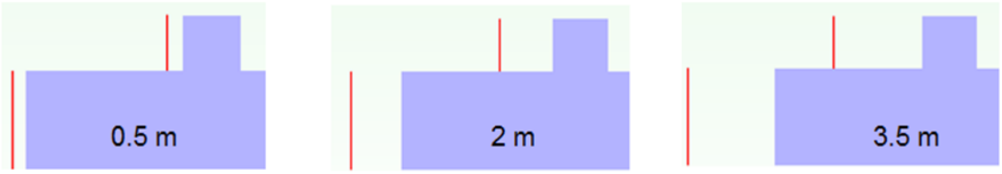


Figure 31: Schematic figure to understand the wall distance from MDC

5.2.4 Split wall Mean Pressure at the exhaust

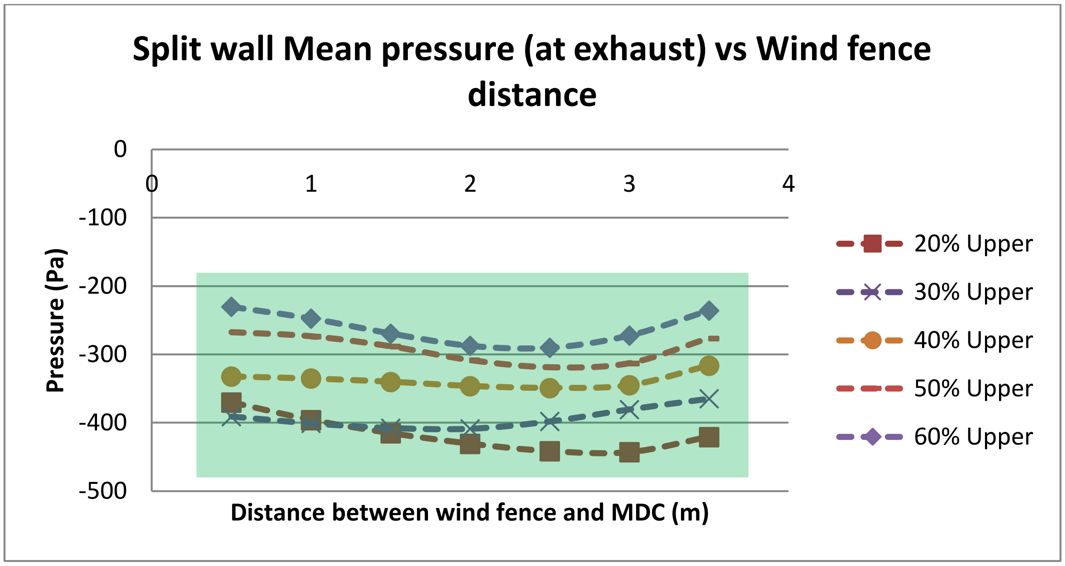


Figure 32: Split wall Mean pressure (at exhaust) vs Wind fence distance

The pressure at the exhaust in the split wall case with respect to the wind fence distance is shown above. Since we intend to vent out the air from the exhaust it is beneficial to have a negative pressure gradient in that region. Therefore, the region highlighted on the graph with a colored box shows all the cases that provide a negative pressure gradient at the exhaust of the MDC. More the negative pressure gradient at the exhaust, the easier it is to vent out air.

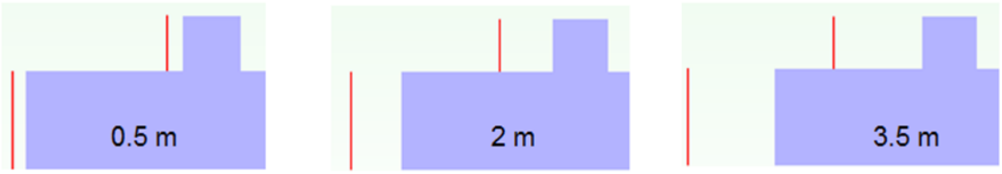


Figure 33: Schematic figure to understand the wall distance from MDC

In the CFD analysis carried out both the lower and the upper wall of the wind fence were moved simultaneously in positive and negative wall advancement scenarios, therefore the next two graphs show the comparison of the X-velocity and Pressure at both inlet and exhaust vs each perforation and wind fence location on the same graph.

5.2.5 Comparison of Split wall X-velocity (at inlet and exhaust) vs Wind fence distance

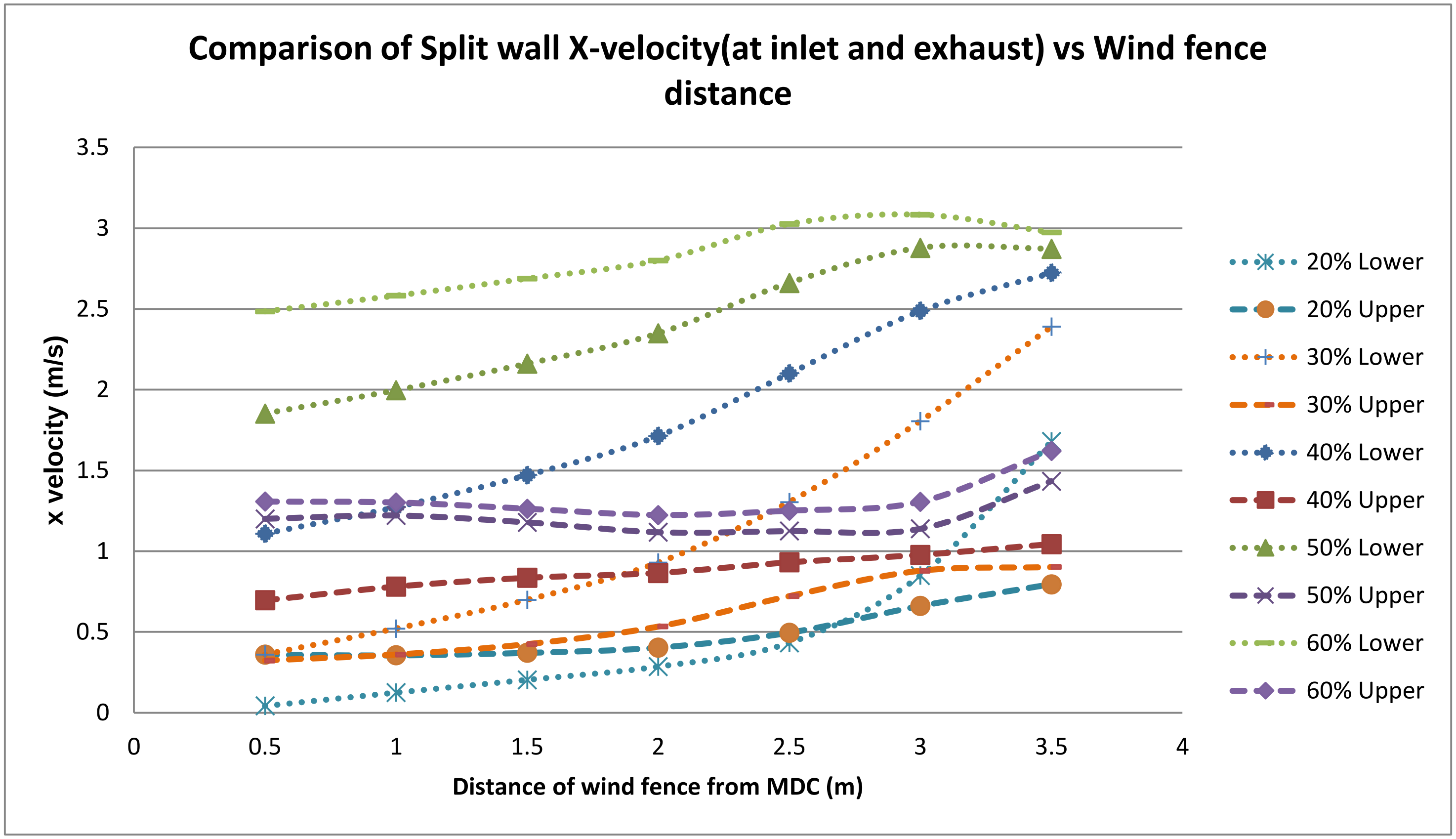


Figure 34: Comparison of split wall x-velocity (at inlet and exhaust) vs Wind fence distance

5.2.6 Comparison of Split wall mean pressure (at inlet and exhaust) vs Wind fence distance

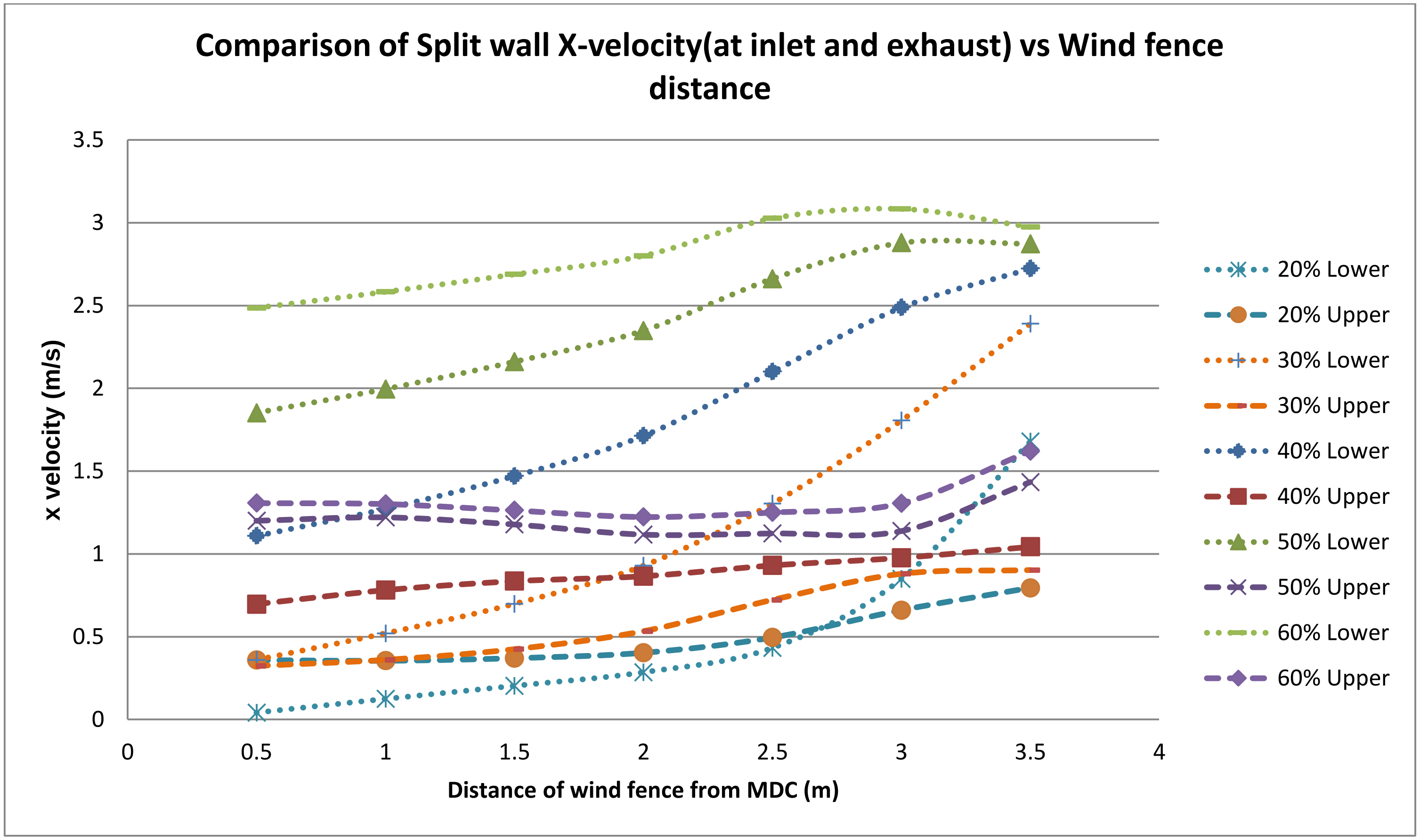


Figure 35: Comparison of Split wall mean pressure (at inlet and exhaust) vs Wind fence distance

5.3 Over View of the Results

5.3.1 Single wall summary

Best cases considering x-velocity at inlet alone:

- 50% : 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 40%: 2.5m, 3m, 3.5m
- 30%: 3m, 3.5m

Best cases considering x- velocity at exhaust alone:

- 50%: 2m, 2.5m, 3m ,3.5m
- 40%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 30%:0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 20%:0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m

Best cases considering mean Pressure at inlet alone:

- 50%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 40%: 2.5m, 3m, 3.5m
- 30%: 3.5m

Final best cases considering X velocity at inlet and exhaust and mean pressure at the inlet:

- ✓ 50%: 2m, 2.5 m, 3m, 3.5m
- ✓ 40%: 2.5m, 3m, 3.5m
- ✓ 30%: 3.5m

5.3.2 Split wall summary

Best cases considering x-velocity at inlet alone:

- 60%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 50%: 1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 40%: 2.5m, 3m, 3.5m
- 30%: 3.5m

Best cases considering x- velocity at exhaust alone:

- 40%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m
- 30%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 20%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m

Best cases considering the mean pressure at inlet alone

- 60%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 50%: 0.5m, 1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 40%: 2.5m, 3m, 3.5m
- 30%: 3.5m

Best cases considering mean pressure at the exhaust alone

- 60%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 50%: 0.5m, 1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 40%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 30%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m
- 20%: 0.5 m ,1 m, 1.5m, 2m, 2.5m, 3m, 3.5m

Final best cases considering X velocity at inlet and exhaust and the mean pressure at the inlet and exhaust when both the lower wall an upper wall perforation and location are identical:

- ✓ 40%: 2.5m, 3m
- ✓ 30%: 3.5m

Chapter 6

Recommendations and Future Work

6.1 Recommendations

Based on the literature survey and some primary calculations the following conclusions can be drawn:

- Height of the fence should be as tall as the data center, any taller than that would be redundant
- Minimum length of the fence should be as wide as the data center

For Single Wall wind fence configuration, the following combination of the perforation and wind fence location has to be used to obtain the best wind load reduction without hindering the free cooling system of the MDC.

- ✓ 50%: 2m, 2.5 m, 3m, 3.5m
- ✓ 40%: 2.5m, 3m, 3.5m
- ✓ 30%: 3.5m

For Split Wall wind fence configuration, the following combination of the perforation and wind fence location has to be used to obtain the best wind load reduction without hindering the free cooling system of the MDC.

- ✓ 40%: 2.5m, 3m
- ✓ 30%: 3.5m

6.2 Future Work

- Fluid-structure interaction with detailed modeling of the fence such as supporting pole structure could be analyzed to extensively study the best material and perforation (hole) sizing for various configurations of the fence.
- The transient analysis could be conducted for in-depth analysis of the flow field. This would help to gain a better understanding of the flow around the MDC

REFERENCES

[1] S. F. Beaufort, "National Weather Service," 1805. [Online]. Available: http://www.spc.noaa.gov/faq/tornado/beaufort.html.

[2] S.-S. Team, "National Weather Service," [Online]. Available: http://www.nhc.noaa.gov/aboutsshws.php.

[3] J. Rath, "Data Center Knowledge," 20 October 2011. [Online]. Available: http://www.datacenterknowledge.com/archives/2011/10/20/dck-guide-to-modular-data-centers-why-modular/.

[4] B. Pitchaikani, "Data Center Knowledge," 8 September 2011. [Online]. Available: http://www.datacenterknowledge.com/archives/2011/09/08/strategies-for-the-containerized-data-center/.

[5] B. WORTHEN, "The Wall Street Journal," 19 April 2011. [Online]. Available: http://www.wsj.com/articles/SB10001424052748704336504576259180354987332.

[6] T. P. Morgan, "Enterprise Tech," 27 September 2013. [Online]. Available: http://www.enterprisetech.com/2013/09/27/ebay-juices-utah-modular-data-center-fuel-cells-waste-heat/.

[7] P. P. Revankar, "UTA Libraries," May 2015. [Online]. Available: https://uta-ir.tdl.org/uta-ir/bitstream/handle/10106/25141/Revankar_uta_2502M_13217.pdf?sequence=1

[8] M. P. Kassner, "Data Center Dynamics," 6 March 2015. [Online]. Available: http://www.datacenterdynamics.com/design-strategy/yahoos-compute-coop-is-not-for-chickens/93474.fullarticle.

[9] C. P. L. AD Robison, "Recovery Act: Yahoo! Compute Coop (YCC): A Next-Generation Passive Cooling Design for Data Centers," Yahoo! Inc., 2011.

[10] "WeatherSolve," [Online]. Available: http://weathersolve.com/how-windbreaks-work/.

[11] O. Vigiak, "Spatial modeling of wind speed around windbreaks," Catena, vol. 52, p. 16, 2003.

[12] "Penn State Extension," [Online]. Available: http://extension.psu.edu/plants/plasticulture/production-details/windbreaks.

[13] R. Udakeri, Comparison Of Cooling Performace Of Overhead And Underfloor Supply With Rear Doorheat Exchanger In High-Density Data Center Clusters, The University of Texas at Arlington, 2008.

[14] L. G.-L. a. D. B. S. D. Agonafer, "The LVEL Turbulence Model for," Application of CAE/CAD, ASME, vol. 18, 1996.

[15] "Autodesk knowledge Network," Autodesk, 10 December 2014. [Online]. Available: http://help.autodesk.com/view/SCDSE/2015/ENU/?guid=GUID-2BF3C8A5-6D70-4F6A-A792-95A27C605E3D.

[16] "Google," [Online]. Available: https://www.google.org/pdfs/google_heliostat_wind_tunnel.pdf.

BIOGRAPHICAL INFORMATION

Devi Prasad Gorrepati was born in Nizamabad, India. He received his Bachelor's degree in Aerospace Engineering from SRM University, India in May 2013 and his Master of Science degree in Mechanical Engineering from the University of Texas at Arlington in December 2015.

During his masters program his research area majorly comprised of Computational Fluid Dynamics under the guidance of Dr. Dereje Agonafer. He has indulged himself in various industry collaborated projects and gained extensive experience working in Laboratory environment. His interest in having a hands-on approach led him to help other lab mates with their projects and also retrofitting old servers to be used as workstations for various computational analysis

He worked as a Teaching Assistant for CAD Lab from his second Semester till the time he graduated and during his third semester he also worked as a Graduate Teaching Assistant under Dr. Daejong Kim for Kinematics and Dynamics.in The University of Texas at Arlington