

Numerical Simulation of Environmental Flow over Buildings
for Renewable Energy Application

by

Guoyi Li

A Thesis Presented in Partial Fulfillment
of the Requirements for the Degree
Master of Science

Approved April 2015 by the
Graduate Supervisory Committee:

Huei-Ping Huang, Chair
Taewoo Lee
Erica Forzani

ARIZONA STATE UNIVERSITY

May 2015

ABSTRACT

For the increasing concerns of influence on environment by fossil-electricity generation, application of renewable energy becomes one of the most focused issues in society. Based on the limitation on urban environment, wind turbines, which can be mounted on rooftop or between buildings, are regarded as a feasible way for wind energy generation. This study presents wind flow simulations in a large-scale environment with certain dimension buildings. Different inlet velocity boundary conditions are tested firstly, and the non-uniform inlet boundary condition shows better agreement with realistic situation. Turbulence intensity is set to be 10% for comparison consistency. The k-epsilon turbulence model is regarded as a better simulation for this certain condition. After that, three different structures, which include single building, pristine double building and modified circular gap double building systems, are tested in this environment condition. The result shows 18.8% velocity increasing on the top of single building system. Pristine double building systems are tested with 4 different gap distances, and building with 10 meters gap achieved the best velocity condition, which 32.8% velocity increasing and 11.8% improvement comparing to single building system, respectively. But the location of maximum velocity moves to the gap and the maximum velocity on the rooftop of double building system is approximately 5.1% lower than single building system. Based on previous study, modified circular gap double building system is created with 10 meters gap. Comparing result with single building system, modified circular gap system achieves higher improvement for wind flow, whose improvement of velocity increasing in the gap and on the rooftop of building are 47.1% and 3.0%, respectively. As a result, the modified circular gap double building can be regarded as a high efficiency system of environmental wind flow over buildings for renewable energy system.

DEDICATION

To my mother and father

ACKNOWLEDGMENTS

It is my honor to take this opportunity to thank Dr. Huei-Ping Huang, my advisor. I would not complete this thesis on time without his guidance and support. I appreciate his valuable suggestions and patient help. And I also thank Dr. Taewoo Lee and Dr. Erica Forzani to be my committee members.

I acknowledge all the journals, books and websites that are cited in this thesis. Such information gives me the basic methods and directions in this study.

I wish to thank my mother, father, friends and family for the support and help, which make me keep forwarding to my dream.

TABLE OF CONTENTS

	Page
LIST OF TABLES	iv
LIST OF FIGURES.....	vii
CHAPTER	
1 INTRODUCTION	1
2 BACKGROUD AND BASIC MODEL SETUP	5
2.1 Simulation Modeling.....	5
2.2 ANSYS FLUENT Setup.....	6
2.3 Introduction of Different Variables in Simulation	10
3 SENSITIVITY TESTS ON THE CHOICES OF TURBULENCE MODELS AND MESH RESOLUTION.....	12
3.1 Turbulence Model Comparison.....	12
3.2 Mesh Sensitive Testing.....	16
3.3 Introduction to Non-uniform Inlet Velocity.....	20
3.4 Turbulence Intensity Testing	23
4 SIMULATIONS WITH TWIN BUILDINGS.....	26
4.1 Double building introduction	26
4.2 Mesh Sensitivity Test.....	27
4.3 Distance Comparison	30
5 INTRODUCTION TO MODIFIED DOUBLE BUILDING SYSTEM, MESH SENSITIVITY STUDY AND IMPROVEMENT STUDY.....	36
5.1 Introduction to Modified Double Building System	36
5.2 Mesh Sensitivity Study.....	37
5.3 Improvement Study.....	40
6 SUMMARY	44
REFERENCES.....	46

APPENDIX	Page
A NON-UNIFORM INLET VELOCITIES WITH MATLAB CODES.....	47
B NON-UNIFORM INLET VELOCITIES WITH C PROGRAM CODES	48

LIST OF TABLES

Table	Page
3.1. Investigation Data for Different Mesh Levels (Uniform Inlet) with Single Building	16
3.2. Mesh Sensitivity Study for Non-uniform Inlet Velocity.....	22
3.3. Turbulence Intensity Testing	24
4.1. Investigation for Different Mesh Levels (Non-uniform Inlet) with Double Building.....	27
5.1. Mesh Study for Modified Double Building System.....	38

LIST OF FIGURES

Figure		Page
1.1	(a) Picture of Three Wind Turbines Between The Two Buildings of Bahrain World Trade Center, Which Cover 11% to 15% Energy Consumption of The Twin Building. (b) Six Wind Turbines Mounted on The Global Institute of Sustainability building That Show the Strategy of ASU to Reduce Usage of Non-renewable Energy.....	3
2.1	(a) The Basic Geometry of Single Building with the Environment Around It. (b) The Symmetry Single Building Structure, Which is Used in the Simulation.....	7
2.2	Detail Setup for Mesh Sizing in FLUENT.....	7
2.3	Details in Standard $k-\epsilon$ Turbulence Model Setup, with the 2-equation Model and the Standard Model Constants.....	8
2.4	(a) The Inlet Boundary Condition Setup Details for the First Simulation Model. (b) The Outlet Boundary Condition Setup Details for the First Simulation Model. (c) No Slip Stationary Wall for Other Boundaries.....	9
2.5	Setup of Solution Method in FLUENT.....	10
2.6	MATLAB Plot of Non-uniform Inlet Velocity with One-Seven Power Law.....	11
3.1	(a) Plot of Residual, Continuity and Velocity Curves with Respect to Iteration Times for $k - \epsilon$ Turbulence Model. (b) Plot of Residual, Continuity and Velocity Curves with Respect to Iteration Times for Spalart-Allmaras Turbulence Model.....	13
3.2	(a) Contour of Horizontal Velocity for $k-\epsilon$ Turbulence Model. (b) Unfilled Contour of Horizontal Velocity for $k-\epsilon$ Turbulence Model for More Clear Observation of Acceleration Field and Backward Vortices Field.....	15
3.3	(a) Contour of Horizontal Velocity for Spalart-Allmaras Turbulence Model. (b) Unfilled Contour of Horizontal Velocity for Spalart-Allmaras Turbulence Model for More Clear Observation of Acceleration Field and Backward Vortices Field.....	16
3.4	(a) Contour of Velocity Filed for the Mesh Level of 177807 Elements. (b) Contour of Velocity Filed for the Mesh Level of 383978 Elements.....	18

Figure	Page
3.5	(a) Plot for Maximum Velocity with 6 Different Mesh Levels. (b) Plot for Acceleration Factors with 6 Different Mesh Levels.....19
3.6	Interpreted UDFs to Get Non-uniform Inlet Velocity.....20
3.7	Setup of Velocity Inlet with Non-uniform Inlet Velocity.....20
3.8	(a) Plot of Velocity on the Symmetry Plane That Is 90 Meters Away From the Inlet Boundary with Uniform Inlet Velocity. (b) Plot of Velocity on the Symmetry Plane That Is 90 Meters Away From the Inlet Boundary with Non-uniform UDF Inlet Velocity.....22
3.9	(a) Velocity Contour of 178709 Elements. (b) Velocity Contour of 351042 Elements. (c) Velocity Contour of 459626 Elements.....23
3.10	(a) Velocity Contour with Highest Mesh Level for 5% Turbulence Intensity. (b) Velocity Contour with Highest Mesh Level for 10% Turbulence Intensity. (c) Velocity Contour with Highest Mesh Level for 15% Turbulence Intensity. (d) Velocity Contour with Highest Mesh Level for 20% Turbulence Intensity.....25
4.1	Structure of Double Building System, Wind Flow Inlet Is x-direction (15m).....27
4.2	Plot for Maximum Velocity with 4 Different Mesh Levels.....28
4.3	(a) Front View of Velocity Field of Double Building System with 1 st Mesh Level. (b) Front View of Velocity Field of Double Building System with 2 nd Mesh Level. (c) Front View of Velocity Field of Double Building System with 3 rd Mesh Level. (d) Front View of Velocity Field of Double Building System with 4 th Mesh Level.....30
4.4	Comparison of Maximum Velocities for Single Building System and Double Building System with Four Different Gap Widths.....31
4.5	(a) Velocity Contour of Double-building System with 5 Meters Width Gap from Parallel View of Inlet Velocity. (b) Velocity Contour of Double-building System with 5 Meters Width Gap from Vertical View of Inlet Velocity. : (c) Velocity Contour of Double-building System with 10 Meters Width Gap from Parallel View of Inlet Velocity. (d) Velocity Contour of Double-building System with 10 Meters Width Gap from Vertical View of Inlet Velocity. (e) Velocity Contour of Double-building System with 15 Meters Width Gap from Parallel View

Figure	Page
	of Inlet Velocity. (f) Velocity Contour of Double-building System with 15 Meters Width Gap from Vertical View of Inlet Velocity. (g) Velocity Contour of Double-building System with 20 Meters Width Gap from Parallel View of Inlet Velocity. (h) Velocity Contour of Double-building System with 20 Meters Width Gap from Vertical View of Inlet Velocity.....33
4.6	Comparison Maximum Velocity over Roof of Building with Single Building Condition and Double Building Condition, Which Contains 4 Different Gap Widths (5 Meters, 10 Meters, 15 Meters and 20 Meters).....35
5.1	Structure of Modified Double Building System with Circular Shape Gap.....37
5.2	Plot for Maximum Velocity with 4 Different Mesh Levels.....38
5.3	(a), (b), (c), (d) Are Front View of Velocity Field of Modified Circular Double Building System with 1 st Mesh Level, 3 rd Mesh Level, 4 th Mesh Level, 5 th Mesh Level, Respectively.....40
5.4.	Maximum Velocity in Modified Circular Double Building System Compared with Maximum Velocities from Single and Pristine Double Building Systems.....41
5.5	(a), (b) Are Front Views of Pristine Double Building Systems and Circular Gap Double Building Systems, Respectively. (c), (d) Are Side Views, Parallel to the Inlet Velocity, of Pristine Double Building Systems and Circular Gap Double Building Systems, Respectively.....42
5.6	Maximum Velocity on Rooftop in Modified Circular Double Building System Compared with Maximum Velocities from Single and Pristine Double Building System.....43

CHAPTER 1

INTRODUCTION

With increasing public concern on producing of greenhouse gas and global warming, it is necessary to generate energy without fossil fuel to reduce greenhouse gas emissions. Our energy strategy for the future life should maximize efficiency of energy generation and minimize the negative effects of energy production on people's lives and environment. One of the feasible ways is utilization of carbon-free energy sources, which produces energy with much less carbon dioxide. Carbon-free energy sources include wind energy, solar energy, geothermal energy, low-head hydropower, hydrokinetics and nuclear power. Among the provided carbon-free energy sources, wind energy is suited to the environment of urban. Especially, urban buildings, to some extent, will contribute to wind energy generation by increasing the inlet wind speed of building-mounted wind turbines. When considers the specific situation that vertical wind direction to wind turbines, the power generated can be expressed as follow

$$P_{Turbine} = \frac{1}{2} \rho C_p A V^3$$

Where $P_{Turbine}$ is the wind turbine power, ρ is the wind density, regarded as air (kg/m^3). C_p is the coefficient of performance, A is the total swept area of turbine blades (m^2). From this equation, it can be observed the cubic relationship between wind speed and wind power generation, which indicates the considerable benefit of study on how to increase the inlet velocity of building-mounted wind turbine.

Due to the benefits introduced, many urban buildings have one or a group wind turbines to provided energy to cover a part of buildings' daily energy consumption. In Manama, Bahrain, There is a world tallest twin building, Bahrain World Trade Center (BWTC), which has a height of 240 meters (50 floors). The two buildings are linked by three skywalks, and each skywalk is mounted a 225 KW wind turbine as shown in Figure 1.1a. As a result, the BWTC has a 675 KW wind energy capacity that can cover 11% to 15% of its totally power consumptions. Another example is shown in figure 1.1b that a group of wind turbines are mounted on the rooftop edge of

the Global Institute of Sustainability building located in Arizona State University Tempe campus. These six wind turbines, under average condition, provide electricity energy to 36 computers in order to reduce the consumption of non-renewable energy of daily maintenance of Arizona State University. Because of raising attention on urban wind energy utilization, many studies on this area have been done and some of them have indicated the further work.

Dursun (2012) in his work presents the benefits of urban roof-mounted wind turbine energy generation system, regarding buildings as a concentrator or an accelerator of wind. The author introduces the concept of Building Augmented Wind Turbines (BAWTs), which can make use of wind energy mentioned above. In study of Islam Abohela (2013), for the vertical wind direction to a single flat roof building, the percentage of increase in power is 31.3%. For more realistic situation, Islam puts the building into a group of other buildings to achieve the situation in urban environment and finds it will lead to yielding the maximum 48% power than freestanding buildings. On the other hand, Lin Lu (2007) considers the effect of distance of two buildings for a wind turbine mounted between them. In conclusion, the increase of in power goes up with the distance decreasing. For the detail setup in wind flow simulations, S. Murakami (1993) have compared the differences of turbulence models, including standard $k - \varepsilon$, standard DSM (ASM) and LES, for a bluff body, and concludes no model can simulate the wind flow over and between buildings perfectly, but standard $k - \varepsilon$ turbulence model is one of the most realistic models and have been used widely. For the other important issue, Robert N. Meroney (1999) states that adapted grids can be regarded as an efficient ways to get more detail information for the flow simulation. As mentioned in the study of Meroney, it is necessary to do the mesh sensitivity testing in wind flow simulation. In addition, many simulation techniques have been used in this specific area. In order to find the differences among them in simulation of large-scale wind flow over buildings, Bernd M. Leid (1997) investigates concentration and flow distributions in the vicinity of U-shaped buildings with different simulation techniques, including ABC, ASMUS, DASIM, FLUENT ($k - \varepsilon$ turbulence model) and MISKAM. Bernd compares the each result with

the data from the wind tunnel experiment and concludes that FLUENT shows the most consistent result with the data from wind tunnel test.



(a)



(b)

Fig 1.1: (a) Picture of Three Wind Turbines Between The Two Buildings of Bahrain World Trade Center, Which Cover 11% to 15% Energy Consumption of The Twin Building. (b) Six Wind Turbines Mounted on The Global Institute of Sustainability building That Show the Strategy of ASU to Reduce Usage of Non-renewable Energy

In this thesis, the wind flow over buildings will be simulated in ANSYS FLUENT based on the previous study to this specific problem and the best location will be determined by the conclusion from numerical results. The basic dimension of structure model is 20 meters in length, 20 meters in width and 30 meters in height, which can be presented as L: W: H=1:1:1.5. The boundary dimension is determined as 200 meters in length, 200 meters in width and 150 meters in height. Meshing independence will be verified based on dimension above. An acceleration factor α is introduced to investigate acceleration of wind, which shows the ratio of wind speed increasing:

$$\alpha = \frac{v'}{v}$$

Where v' is the maximum velocity around the build, and v is the inlet velocity in the same location with maximum velocity point in this system. As the cubic relationship mentioned above, the ratio of power increase is defined as:

$$R = \alpha^3$$

These two variables, α and R , are used to be the evidence of the energy benefit in different simulation variables and different structures of buildings. Stable mesh level is determined as the variance of maximum velocity is less than 1%, which means the variance of power increasing is less than a negligible number, 0.001%. When the stable mesh level is determined, effect of turbulent intensity is evaluated. Though S. Murakami has evaluated some turbulence models, he does not consider the Spalart-Allmaras model. The comparison of $k - \varepsilon$ turbulence model and Spalart-Allmaras model will be provided in this thesis. After then, implement the non-uniform inlet velocity instead of uniform velocity and compare the results from both types inlet definition. With the more realistic non-uniform inlet velocity, effect on different distances of twin building will be tested with the same building dimension. In order to achieve higher power increasing, two original design structures of double building wind turbine system, which are defined Modified Twin Buildings (MTBs), will be investigated and compare the power increase of them with original twin building to show the benefits of these two MTBs structures. As a result, the MTBs designed in this thesis become constructive ways to increase efficiency of building mounted wind turbines.

CHAPTER 2

BACKGROUND AND BASIC MODEL SETUP

2.1 Simulation Modeling

With the numerical iteration method, Computational Fluid Dynamics (CFD) can be used to model the real situation that wind flow around buildings. In this thesis, ANSYS FLUENT is used to simulate different wind flow situations with different variables.

In wind flow simulation, turbulence model and mesh level are two major important concerns, which influents on the convergence and stability of simulation. As mentioned before, several kinds of turbulence models are tested. Based on the provided models by FLUENT, the standard $k - \varepsilon$ turbulence model is operated most of simulations in this thesis, except for one comparison testing on Spalart-Allmaras model. For the standard $k - \varepsilon$ turbulence model, k is the turbulence kinetic energy and ε is the rate of dissipation. As Lin Lu mentions, the relationship between these two parameters is derived as:

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

And

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

As result,

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Where μ is air density, μ_t is turbulent viscosity; ρ is fluid density, σ_k and σ_ε are the turbulent Prandtl numbers for k and ε , respectively. C_μ , $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ are constants, G_k is the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, u_i is mean velocity components along x, y and z axes. S_k , S_ε are

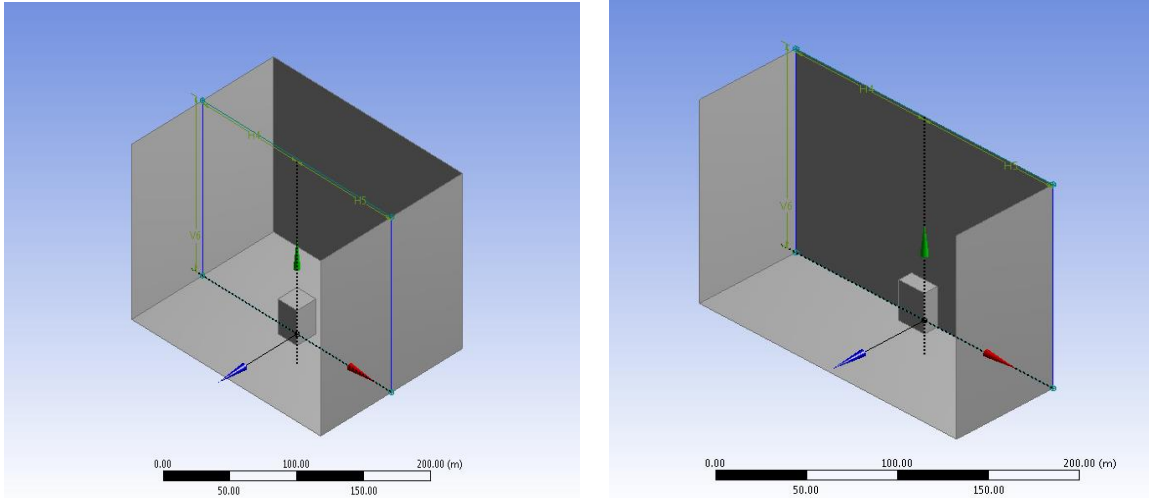
user-defined source terms and Y_M is the contribution of the fluctuating dilatation on compressible turbulence to overall dissipation rate.

In order to testing the mesh sensitivity to this system, 3 major mesh levels are compared and modified. The modified method is to utilize mesh adaption option in FLUENT. In detail, due to the most important concern is wind velocity around buildings, velocity gradient adaption is applied. For each mesh level, a general rule is to use 10% of the maximum gradient when setting the value for Refine Threshold.

2.2 ANSYS FLUENT Setup

There are two major setups in FLUENT: Mesh setup and Solution setup. In this paragraph, details in of the basic setup will be introduced step by step. The main concern is controlling variables in certain simulation system and increasing reliability of result comparisons.

Because meters describe dimensions of buildings in this problem, the first issue is to set the unit of the geometry as meter. To create the geometry of building, method in this thesis is that build the building firstly and fix this structure by freeze tool, then create the around environment based on dimensions described in INTRODUCTION part. With the body operation, the fixed part of whole geometry is cut and the set the rest part of structure to fluid instead of solid. Final step is to cut structure along the symmetry plane to save simulation data space and increase iteration speed. The basic geometry and modified geometry are shown in Figure 2.1.



(a)

(b)

Figure 2.1: (a) The Basic Geometry of Single Building with the Environment Around It. (b) The Symmetry Single Building Structure, Which is Used in the Simulation

With the meshing monitor in FLUENT, the sizing setup is shown in Figure 2.2 and program controlled inflation is applied. In order to get the relative high resolution of mesh, it is necessary to choose a high inflation growth rate. So, growth rate is set to be 2 in this whole simulation. On the other hand, all boundaries should be named, including the body, to make an accessible definition to the solution method for next step.

Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine

Figure 2.2: Detail Setup for Mesh Sizing in FLUENT

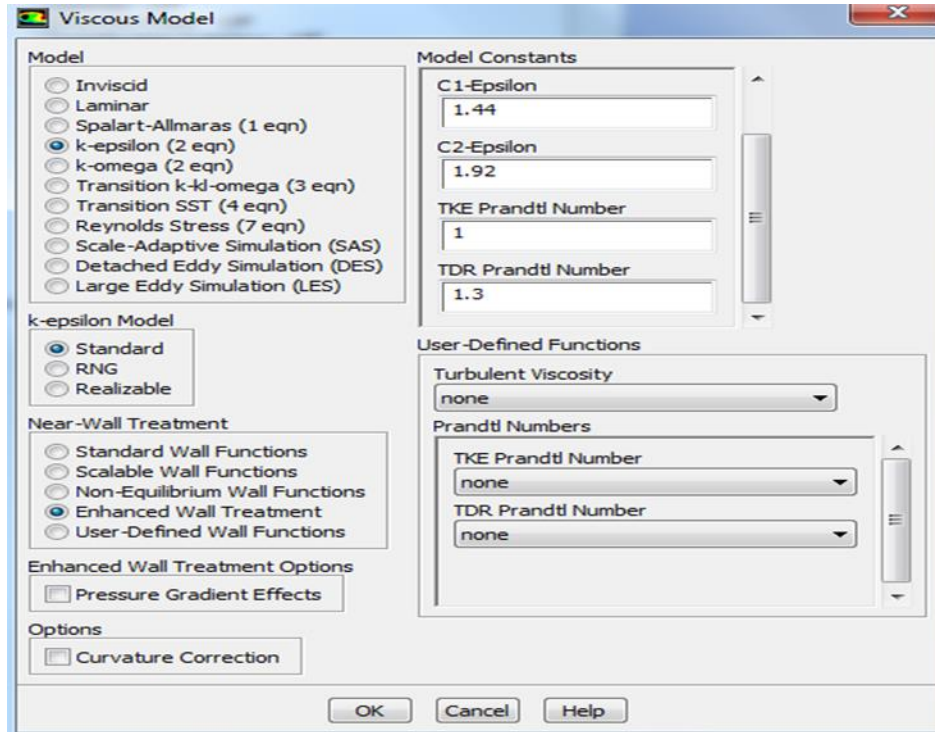
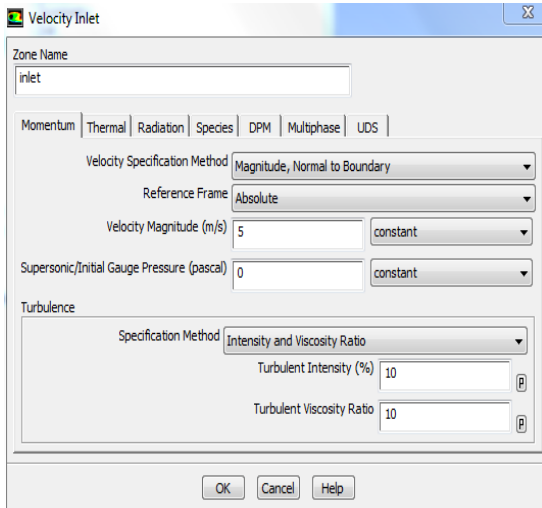


Figure 2.3: Details in Standard $k-\epsilon$ Turbulence Model Setup, with the 2-equation Model and the Standard Model Constants

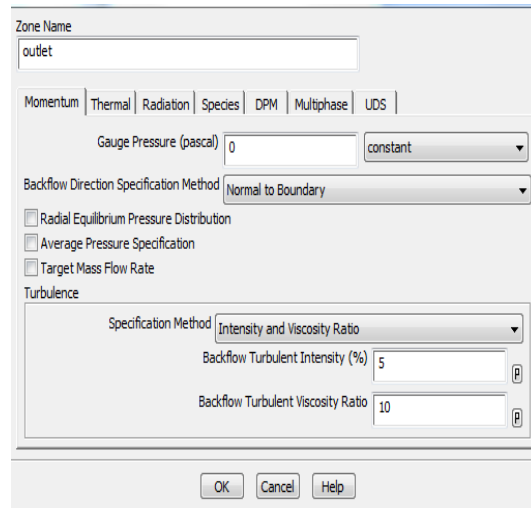
Before entering FLUENT, double precision and serial processing options are applied, with 3D dimension is enabled automatically to this system. Among the viscous models, standard $k-\epsilon$ turbulence model is applied. Because the turbulence layer near the building boundaries is an important concern in this study, the enhanced wall treatment is enabled to get better results of the boundary layers. Detail set up is shown in Figure 2.3. For the material of flow, air is used and density and viscosity are set to be 1.225 kg/m^3 , $1.7894 \times 10^{-5} \text{ kg/(m} \cdot \text{s)}$, respectively. The inlet boundary condition and outlet boundary condition are set as shown in Figure 2.4 (a) and (b), which is the situation of the first simulation. And other walls are set to be no slip stationary wall. As an example, the setup of front wall is demonstrated in Figure 2.4 (c). SIMPLE scheme and Green-Gauss Node Based gradient, whose gradient method is suggested for tetrahedral meshes, are applied for solution methods and other detail is shown in the Figure 2.5. For the residuals, set all residuals to be 0.001. The system will not to be converged in most situations, because the kinetic energy will not as stable as we expect due to the large-scale geometry and relative coarse

mesh. Simulation is stopped when the residual curves are all relative stable. Detail analysis on the result precision will be discussed in further part.

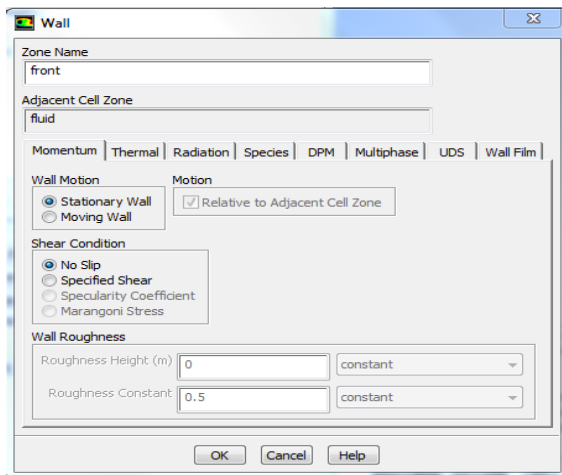
The last procedure is the solution initialization and hybrid initialization is applied, then the number of iterations is set to be 2000. After running the program, all data should be reserved and one simulation is completed.



(a)



(b)



(c)

Figure 2.4: (a) The Inlet Boundary Condition Setup Details for the First Simulation Model. (b) The Outlet Boundary Condition Setup Details for the First Simulation Model. (c) No Slip Stationary Wall for Other Boundaries

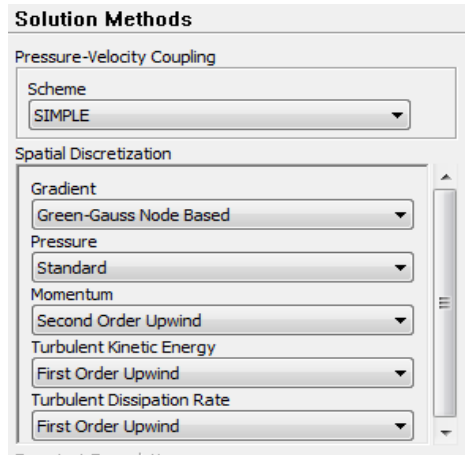


Figure 2.5: Setup of Solution Method in FLUENT

2.3 Introduction of Different Variables in Simulation

Variables that are tested in simulations will be introduced in this part. The first variable is inlet turbulence intensity. Few studies show the relationship between inlet turbulence intensity and wind speed acceleration. So, this investigation would be a reference for the further work on wind turbine study. Secondly, uniform and non-uniform inlet velocities are compared with different mesh level. As the uniform inlet velocity is set to be 5 m/s, One-seven power law is implemented with the inlet velocity, which is at the same height of building roof, set to be 5 m/s as well. The governing equation is shown below:

$$\frac{\bar{u}}{u_1} = \left(\frac{z}{z_1}\right)^p$$

Where u_1 is the known mean wind speed at a height z_1 , and \bar{u} is the desired wind speed at some other height. For the One-Seven Power Law, constant p is assumed to be $1/7$, which is mentioned to be one of the most realistic situations for wind flow around buildings. Based on the original equation and basic concern of simulation, it is necessary to keep the non-uniform velocity located on the building roof same as uniform inlet velocity. The velocity located on building roof is regarded as location of maximum velocity after accelerating. So, u_1 is equal to 5 m/s, which is the same as uniform inlet velocity and z_1 is equal to 30 meters, which is the height of buildings in the system. To the specific situation in this system, the equation is modified as:

$$u = 5 \left(\frac{z}{30} \right)^{1/7}$$

MATLAB codes are in the ANNEX 1. Based on this equation, C program is compiled and load in the ANSYS FLUENT as User Defined Function (UDF). The C programing code is also collected in ANNEX 1. The plot of non-uniform inlet velocity is demonstrated in Figure 2.6. The last variable is distance. Distance between buildings is major concern factor of acceleration rate of wind speed. Distances comparisons are applied to two building analysis and availability certification of original design building structures.

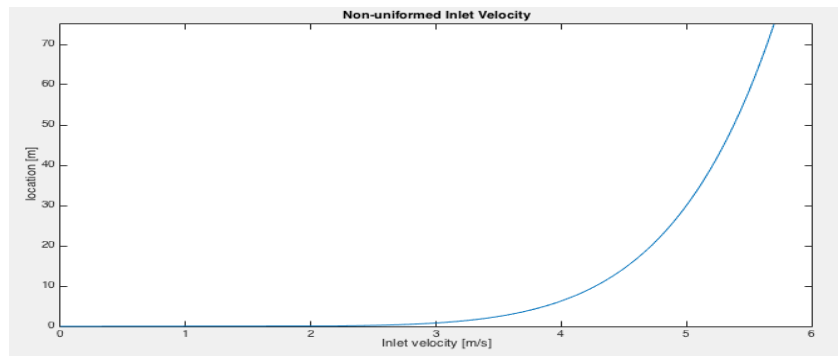


Figure 2.6: MATLAB Plot of Non-uniform Inlet Velocity with One-Seven Power Law

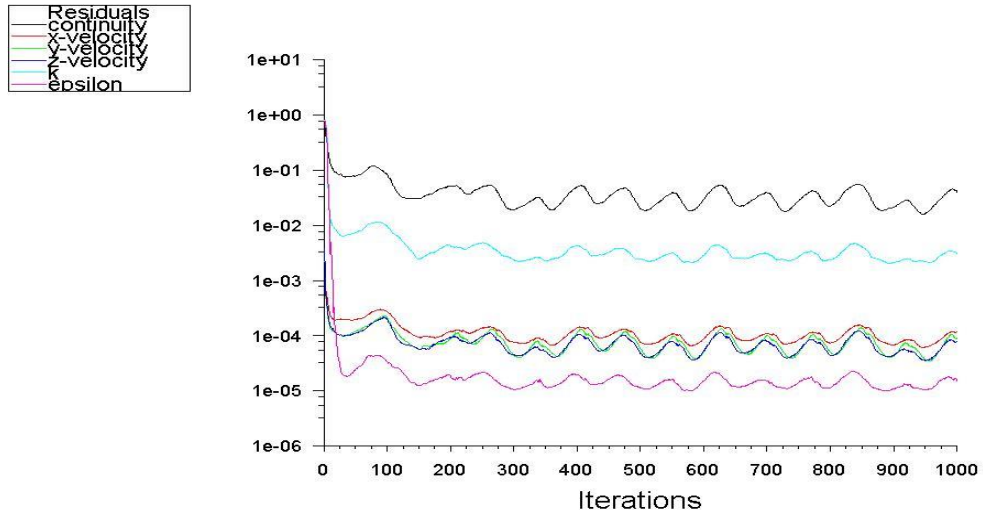
CHAPTER 3
SENSITIVITY TESTS ON THE CHOICES OF TURBULENCE MODELS AND MESH
RESOLUTION

3.1 Turbulence Model Comparison

Based on previous study mentioned in Chapter 1 and Chapter 2, standard $k - \varepsilon$ turbulence model is one of the most realistic turbulence models, compared with some other turbulence models. However, difference of standard $k - \varepsilon$ turbulence model and Spalart-Allmaras turbulence model was not compared. Spalart-Allmaras turbulence model is a relatively simple one-equation model. It is a less memory-intensive model. In this comparison, the basic idea is to compare the near wall vortices and speed acceleration field, which are regarded as the most important concerns for renewable energy application. On the other hand, iteration convergence should be considered as the evidence of accuracy.

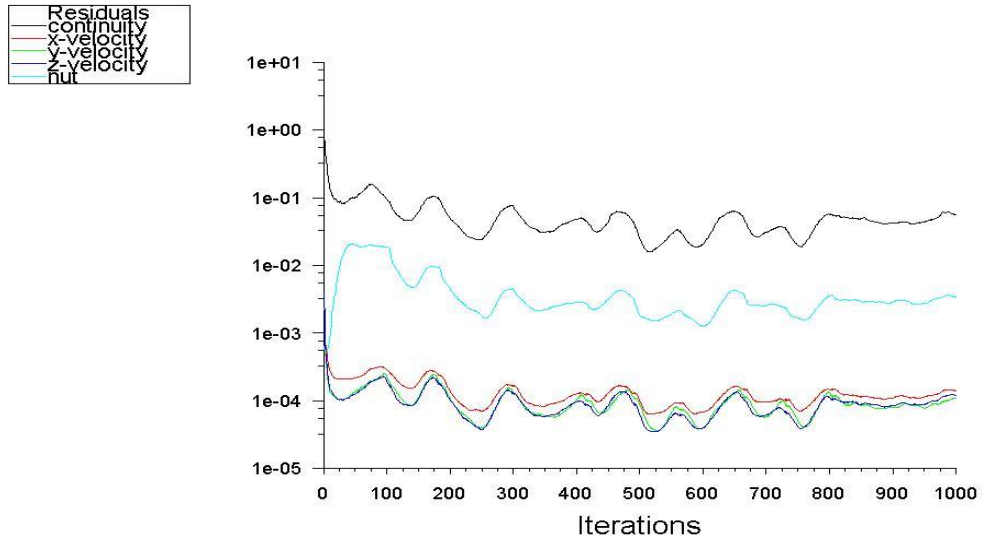
In ANSYS FLUENT, setups for two different models are exactly the same. First of all, mesh level is set to be 177685 for the symmetric part and turbulence intensity is set to be 10%. For the inlet velocity, to simplify the comparison, uniform inlet velocity is implemented, which is set to be 5 meters per second. In order to compare convergence effectiveness, total number of iterations are counted when relative steady for both models.

Firstly, from the plots of convergence curve in the same mesh level in Figure 3.1, convergence curves of standard $k - \varepsilon$ turbulence model and Spalart-Allmaras turbulence model are at a similar stable level, and variance of residual of standard $k - \varepsilon$ turbulence model is relative small. However, After 800 iteration times, the convergence curve of Spalart-Allmaras turbulence model shows a much better convergence effect than $k - \varepsilon$ turbulence model.



Scaled Residuals Feb 03, 2015
ANSYS Fluent 14.5 (3d, dp, pbns, ske)

(a)



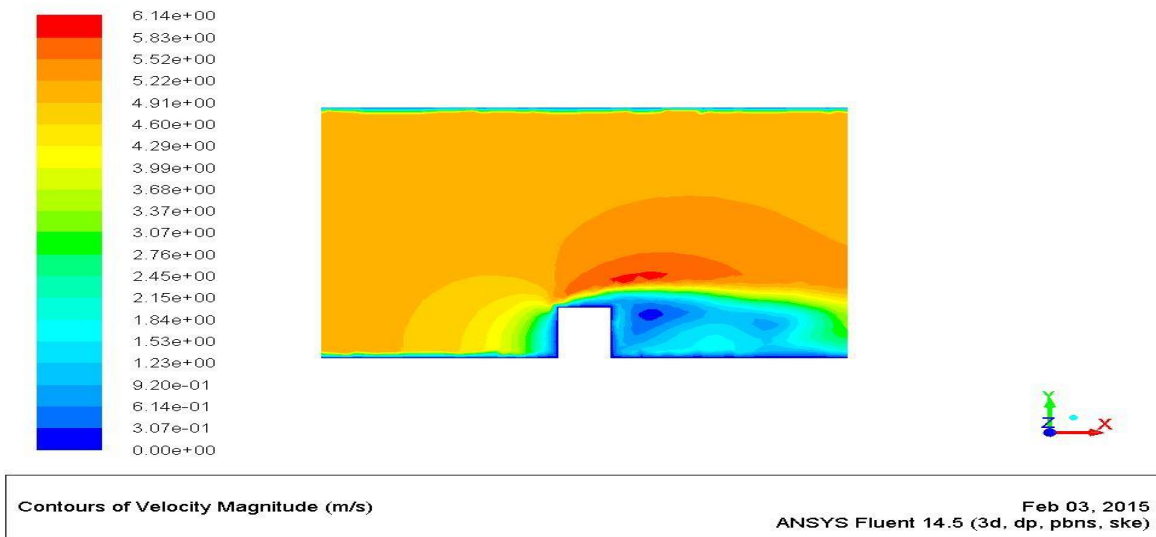
Scaled Residuals Feb 03, 2015
ANSYS Fluent 14.5 (3d, dp, pbns, S-A)

(b)

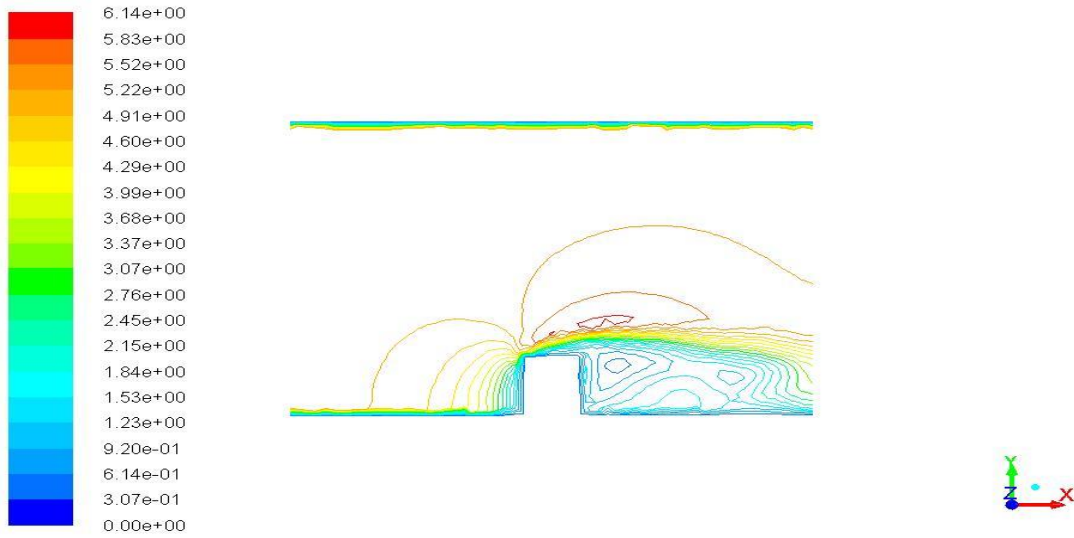
Figure 3.1: (a) Plot of Residual, Continuity and Velocity Curves with Respect to Iteration Times for $k - \epsilon$ Turbulence Model. (b) Plot of Residual, Continuity and Velocity Curves with Respect to Iteration Times for Spalart-Allmaras Turbulence Model

For the same mesh resolution level, it can be observed that the major parts of wind flow contours are similar, which are shown in Figure 3.2 and Figure 3.3 for $k - \varepsilon$ turbulence model and Spalart-Allmaras turbulence model respectively. The maximum velocities after wind speed acceleration for $k - \varepsilon$ turbulence model and Spalart-Allmaras turbulence model are 6.14 m/s and 6.40 m/s, respectively. The most important point is that the contours for velocity over and behind the build in $k - \varepsilon$ turbulence model are much more smooth and clear than Spalart-Allmaras turbulence model, which can describe the shapes and thickness of turbulence layer, acceleration field and back vortices field more effectively.

In conclusion, even though Spalart-Allmaras turbulence model is quite stable and shows good convergence, this model does not show accuracy in computing the shear flow, separated flow (yellow part over the building) and decaying turbulence (located at rear of building). Because this simulation aims at the acceleration field (height, location and associated turbulence boundary layers), $k - \varepsilon$ turbulence model is more suitable, which can describe turbulence layer and wind flow acceleration more effectively. As result, $k - \varepsilon$ turbulence model is used to investigate following situations for renewable energy application with wind flow speed acceleration over buildings.



(a)

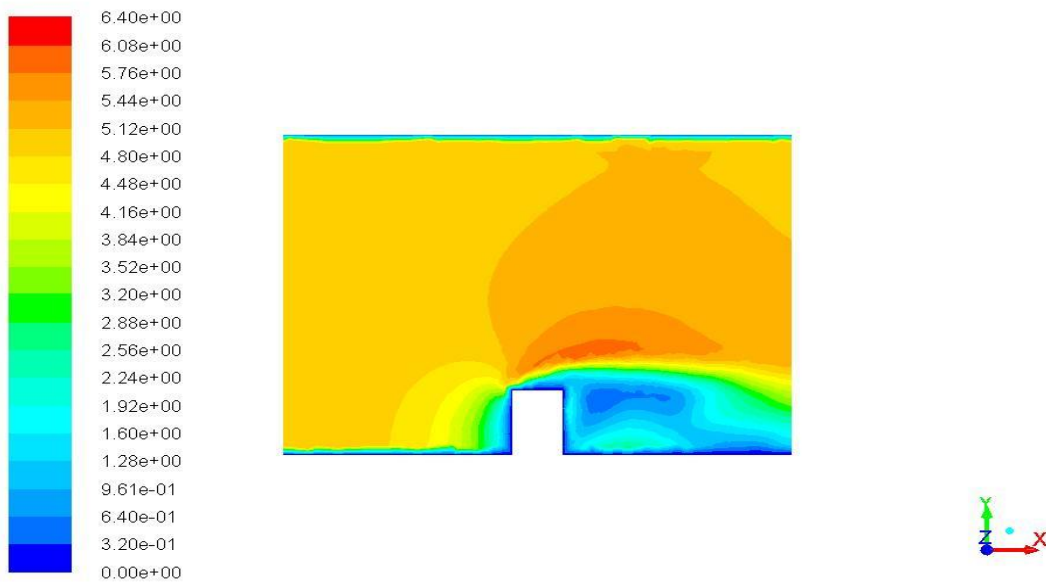


Contours of Velocity Magnitude (m/s)

Feb 03, 2015
ANSYS Fluent 14.5 (3d, dp, pbns, ske)

(b)

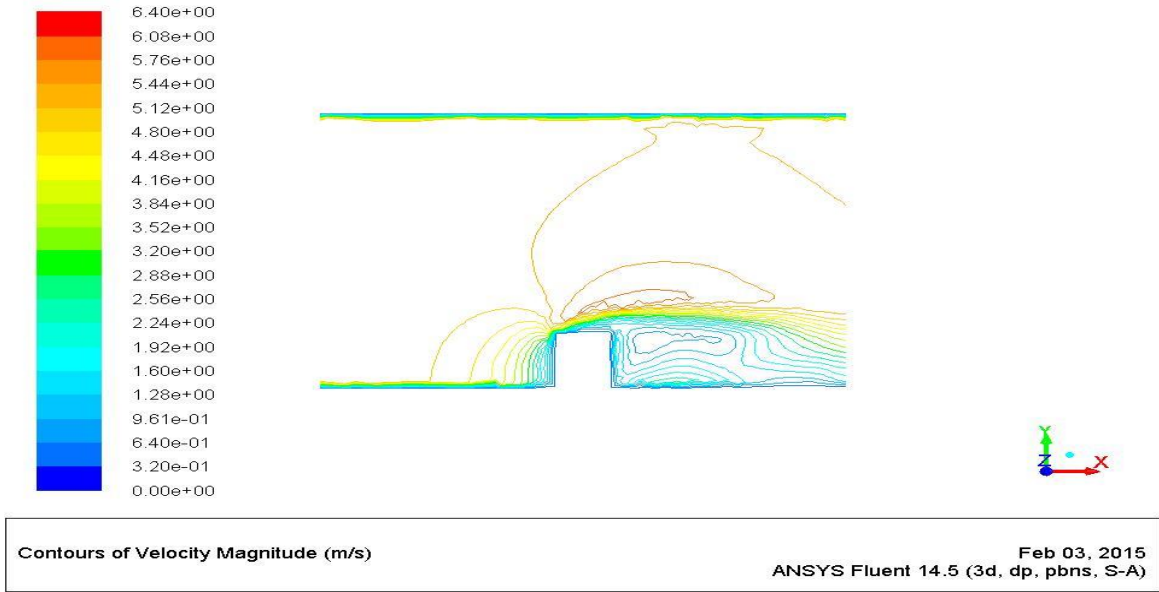
Figure 3.2: Contour of Horizontal Velocity for k-ε Turbulence Model. (b) Unfilled Contour of Horizontal Velocity for k-ε Turbulence Model for More Clear Observation of Acceleration Field and Backward Vortices Field



Contours of Velocity Magnitude (m/s)

Feb 03, 2015
ANSYS Fluent 14.5 (3d, dp, pbns, S-A)

(a)



(b)

Figure 3.3: (a) Contour of Horizontal Velocity for Spalart-Allmaras Turbulence Model. (b) Unfilled Contour of Horizontal Velocity for Spalart-Allmaras Turbulence Model for More Clear Observation of Acceleration Field and Backward Vortices Field

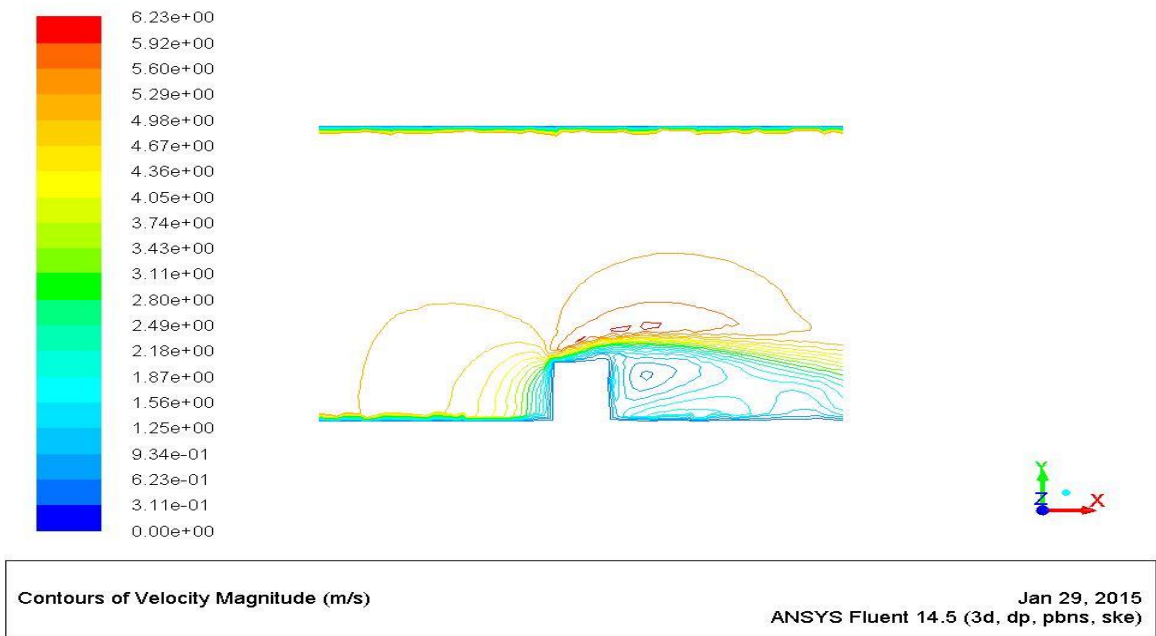
3.2 Mesh Sensitive Testing

For the turbulence model in large-scale wind flow, mesh sensitivity should be concerned as a fact of iteration convergence. Before testing on different variables in simulation system, it is necessary to make sure the result is independent from mesh levels. Because application of wind energy is aimed at the speed acceleration of wind around buildings, the maximum speeds in steady state of the system are compared with different mesh level. On the other side, turbulence intensity and inlet velocity are controlled to be the same, which are set to be 10% and 5m/s uniform inlet velocity, respectively. And the original data of such tests are provided in table 3.1.

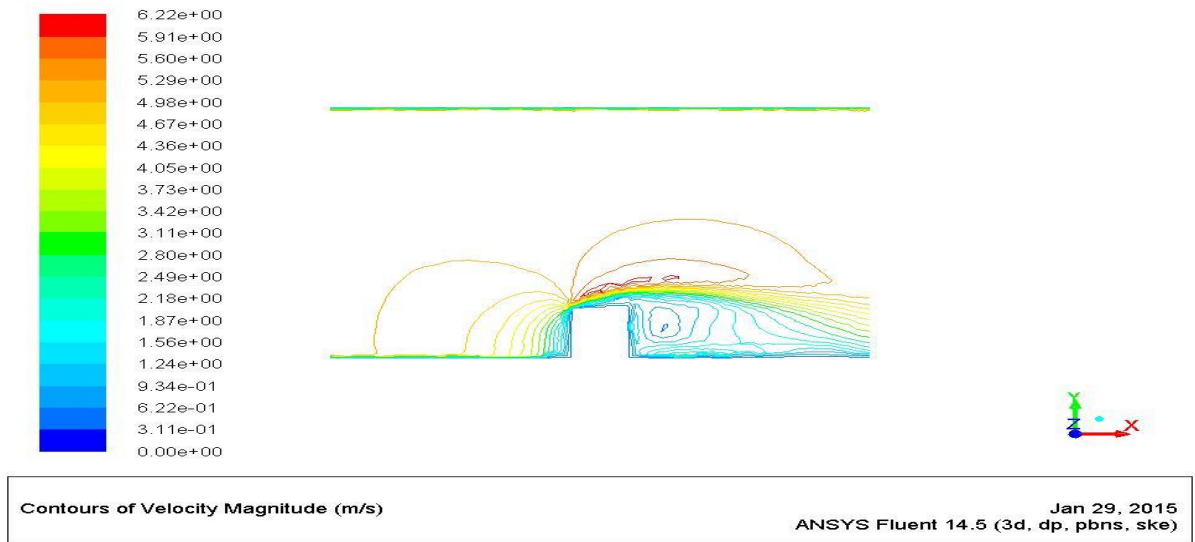
Table 3.1: Investigation Data for Different Mesh Levels (Uniform Inlet) with Single Building

Simulation Number	Mesh level (Tetrahedral cells)	Turbulence intensity	Inlet velocity [m/s]	Maximum velocity [m/s]	Acceleration factor α
1	163953	10%	5	6.242676	1.2485352
2	177807	10%	5	6.227428	1.2454856
3	383978	10%	5	6.224184	1.2448368
4	582638	10%	5	6.255041	1.2510082
5	591717	10%	5	6.291282	1.2582564
6	729505	10%	5	6.284508	1.2569016

Contours for two main mesh levels are shown in Figure 3.4. It can be observed that velocity fields are similar in the 2 mesh levels. On the other hand, locations and shapes of acceleration fields are similar. However, shapes of backward vortices fields are extremely different among these two mesh levels. The phenomenon indicates that locations and shapes of acceleration fields have been independent from provided mesh levels, but backward vortices fields are sensitive to different mesh levels. For the main concern is velocity acceleration field of wind flow over building, it is acceptable to analysis system based on such mesh levels. But this setup of system should not be used in simulations for studying backward vortices fields, which calls for higher mesh level or higher order algorithm.



(a)

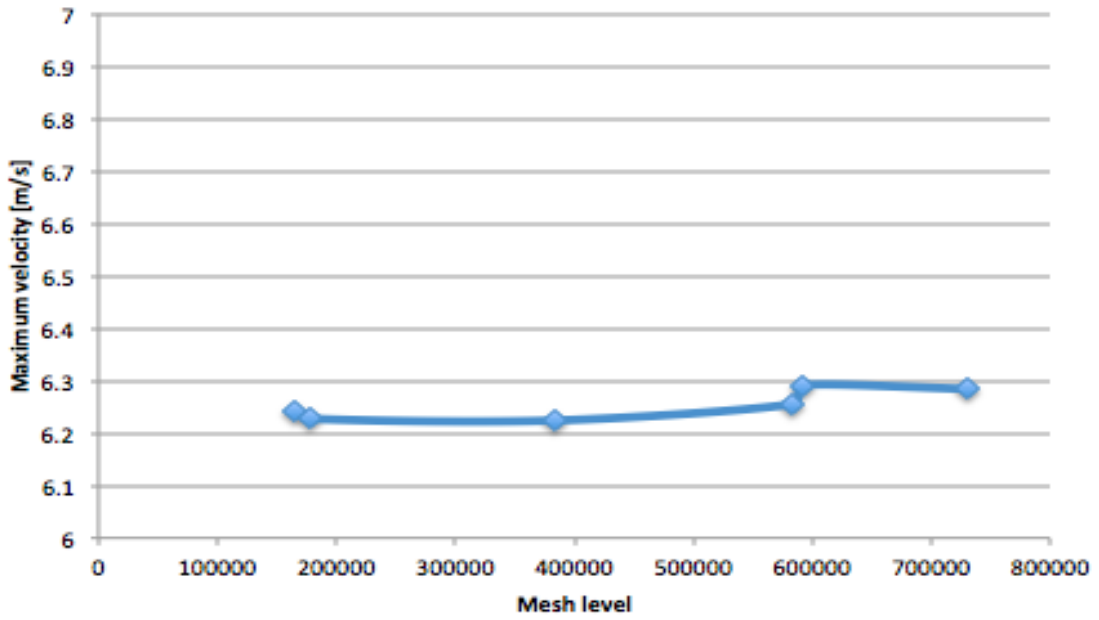


(b)

Figure 3.4: (a) Contour of Velocity Filed for the Mesh Level of 177807 Elements. (b) Contour of Velocity Filed for the Mesh Level of 383978 Elements

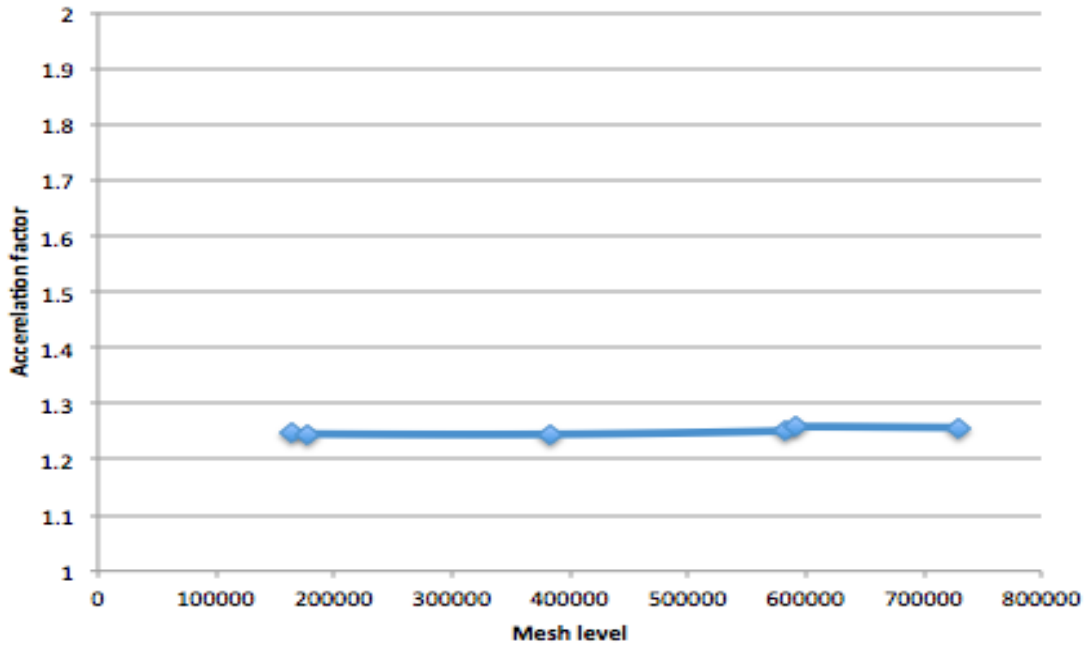
Value of maximum velocities and acceleration factors from different mesh levels, which are regarded as the main concerns of this simulation, have been shown in Figure 3.5. It can be observed that variance of maximum velocities values are in a very small range. Among all mesh levels; the maximum variance is 1.078 %, which is small enough to be ignored. On the other hand, the variances of acceleration factor are also small enough that can be ignored that those from maximum velocities, which is no more than 1.079%. Because the simulation is based on large-scale turbulence wind flow, with the 3 dimensional cells, efficiency of simulation should be considered as one of main concerns. It will take a lot memory space when calculating, which lead to decrease speed of calculate significantly and lower the efficiency of simulation. At the same time, relative fine intervals will be out of memory of computer in current CFD simulation laboratory due to the install memory (RAM) limitation. Combining all concerns mentioned above, there are two major variances in this mesh sensitivity test for single building, which are different mesh levels and dissimilarity of backward vortices. Both of them are acceptable for this study, which emphasizing on comparisons of different building situations. And the data from this setup in FLUENT is reliable for discussions and comparisons. It is reasonable to use the finest mesh level to test other variables for the wind flow over a single building with the last mesh level.

Maximun velocities for different mesh levels



(a)

Acceleration factors for different mesh levels



(b)

Figure 3.5: (a) Plot for Maximum Velocity with 6 Different Mesh Levels. (b) Plot for Acceleration Factors with 6 Different Mesh Levels

3.3 Introduction to Non-uniform Inlet Velocity

For more realistic simulation, based on one seven power low mentioned before, User defined Functions (UDFs) are used to achieve this specific inlet boundary velocity condition. In order to study the difference between uniform and non-uniform inlet velocities, testing is based on defined environment without buildings to investigate velocity curves at where the front wall of building should be located. Compiling the C program as shown in ANNEX 1, this inlet velocity functions keep velocity, which locates at the same horizontal level of building roof. In the case, the roof of building is regarded as the reference point that achieves the maximum velocity in original single building system, approximately. FLUENT can interpret this program and screenshot has been shown in Figure 3.6. When determine inlet velocity, parameter defined in the program should be chosen, which is shown in figure 3.7.

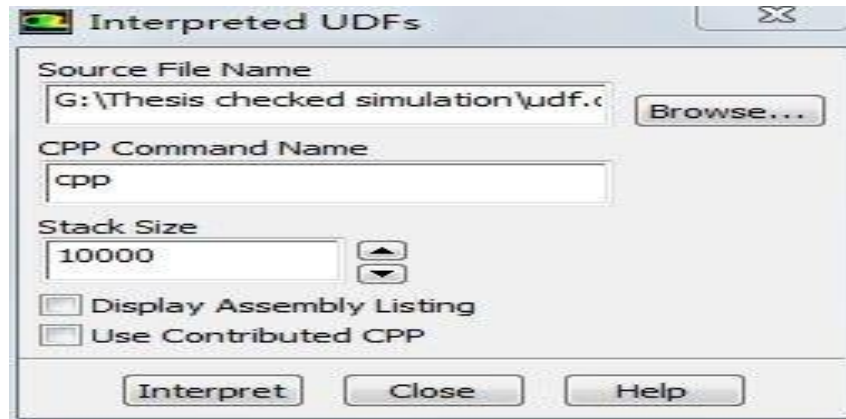


Figure 3.6: Interpreted UDFs to Get Non-uniform Inlet Velocity

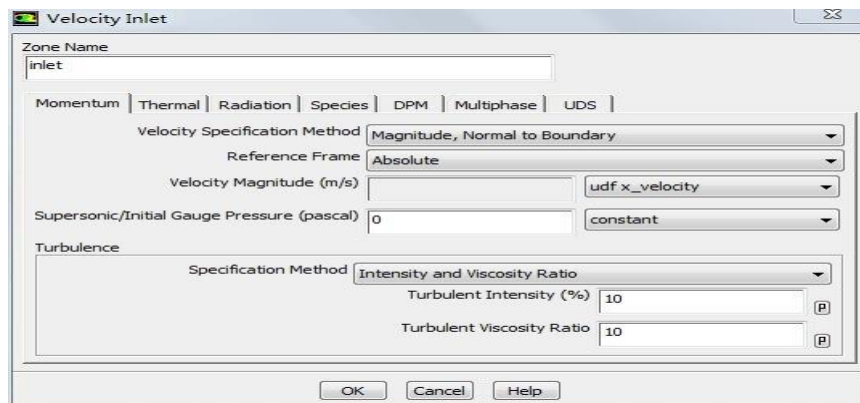
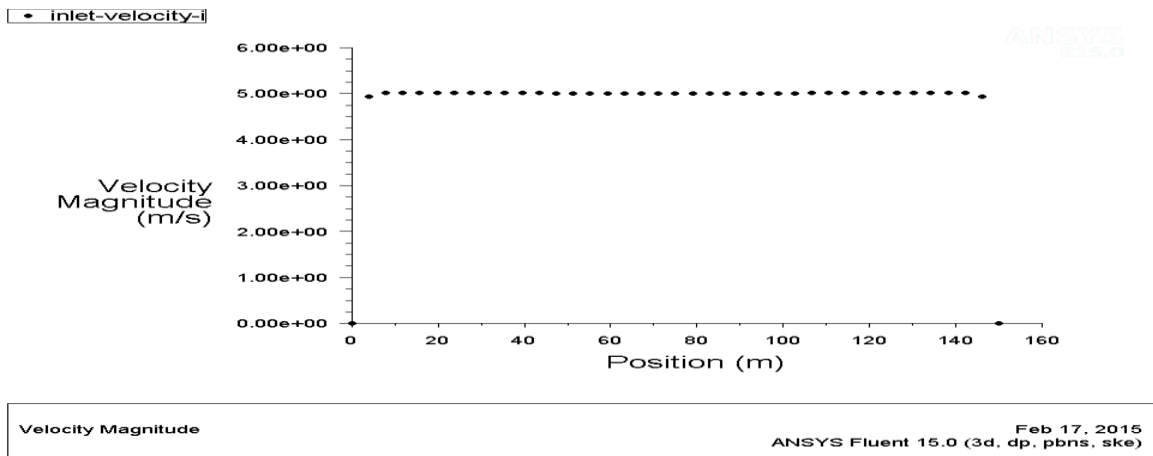
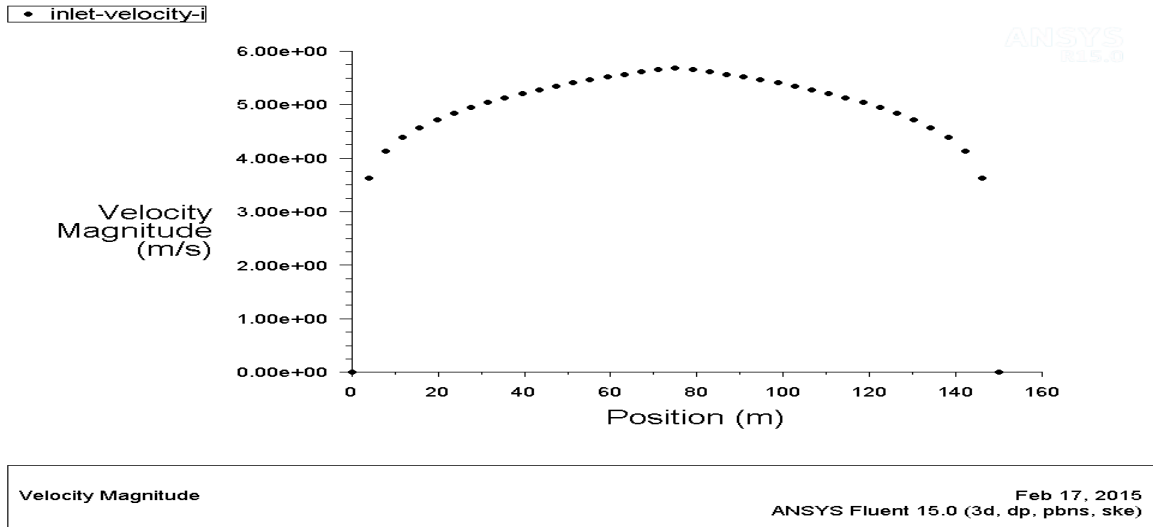


Figure 3.7: Setup of Velocity Inlet with Non-uniform Inlet Velocity

After completing both simulations for uniform and non-uniform inlet velocities, the lines, which are 90 meters away from inlet boundary, are plotted for both inlet boundaries respectively. Two plots are shown in Figure 3.8. It can be observed that, for the uniform inlet, the wind flow velocity keeps uniform (5 m/s) except for a thin layer near upper and bottom, where the velocity lows down. This is because in the setup of simulation, stationary boundary walls are defined so that the simple wind flow over boundary just causes a relative thin turbulence layer. Another interpretation is that the length from inlet to the front wall of building is not long enough for wind flow changing to more realistic shape. In more detail, the for such inlet distance, the viscosity of fluid, which is regarded as air, is relative small. So, in order to achieve a realistic inlet velocity, the distance between inlet boundary and front wall should be elongated. However, this is not a feasible method for large-scale wind flow simulation, because such elongation will take much more computational space. On the other hand, velocity from non-uniform inlet is a relative smooth curve, which is much more realistic based on statement in Chapter 1 and Chapter 2. So, in next step of simulation, for the relative short distance from inlet boundary to front wall of building, non-uniform velocity is used as inlet velocity boundary condition. What's should be done before implement this technique is analysis mesh sensitivity to identify result from non-uniform inlet simulation is independent from mesh levels. Procedures are exactly the same as mesh sensitive testing for uniform inlet velocity boundary.



(a)



(b)

Figure 3.8: (a) Plot of Velocity on the Symmetry Plane That Is 90 Meters Away From the Inlet Boundary with Uniform Inlet Velocity. (b) Plot of Velocity on the Symmetry Plane That Is 90 Meters Away From the Inlet Boundary with Non-uniform UDF Inlet Velocity

Based on non-uniform inlet velocity, which is more realistic, mesh sensitive testing is implemented with the same procedures as uniform inlet velocity and the data from simulations is shown in Table 3.2 and Figure 3.9. In this part, 3 mesh levels have been tested. It can be observed that the variances of maximum velocities are extremely small, which are all within 0.001 m/s (0.0168% to the maximum velocity). It indicates the maximum velocity after wind flow acceleration is independent from mesh level. On the other hand, with iterations converging automatically, both velocity field over roof of building and vortices field backward of building share high similarity, which is different from result of uniform inlet condition. It means such simulation setup is useful for investigate wind flow over building, not only for velocity field over roof of building, but also for vortices field backward of building.

Table 3.2: Mesh Sensitivity Study for Non-uniform Inlet Velocity

Simulation Number	Mesh level (Tetrahedral cells)	Turbulence intensity	Impending velocity to roof of building [m/s]	Maximum velocity [m/s]	Acceleration factor α
1	178709	10%	5	5.942227	1.1884454
2	351042	10%	5	5.942313	1.1884626
3	459626	10%	5	5.943198	1.1886396

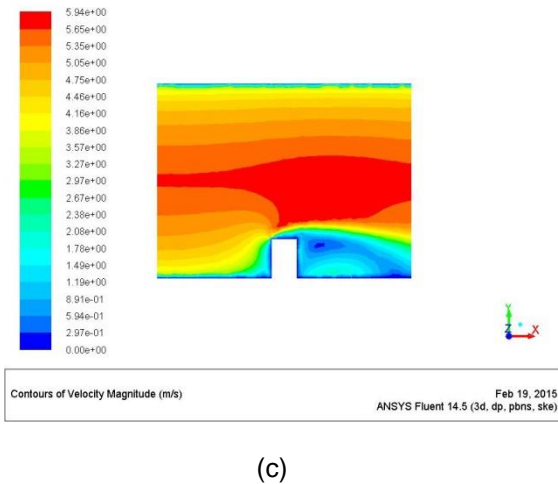
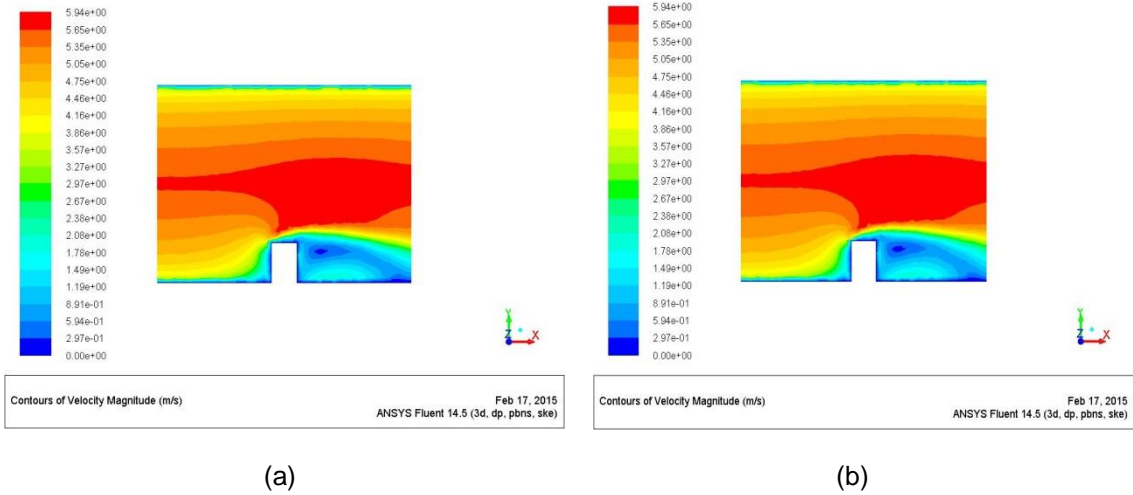


Figure 3.9: (a) Velocity Contour of 178709 Elements. (b) Velocity Contour of 351042 Elements. (c) Velocity Contour of 459626 Elements

3.4 Turbulence Intensity Testing

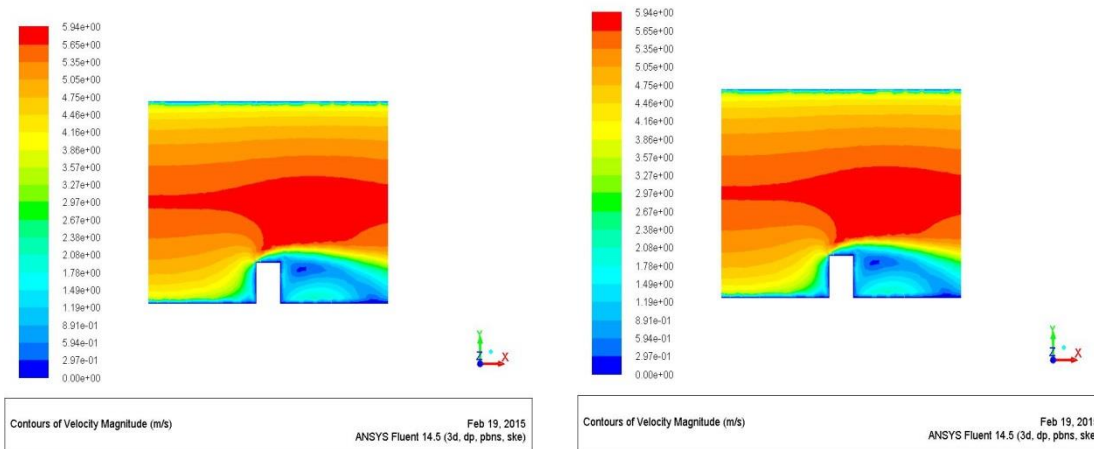
Inlet boundary conditions are major effects that will lead to variance of results. Except for inlet velocity condition, inlet turbulence intensity is concerned as the other variables, which may influence on simulation of large-scale wind flow. For different condition of different fluid flow field, turbulence intensity changes a lot. Before finalizing boundary conditions for this CFD simulation, turbulence intensity should be determined. With the relative high inlet velocity of wind flow, this system is regarded as high-turbulence case. For common estimations of turbulence intensity, typically the value is between 5% and 20% [15]. In order to investigate if inlet turbulence intensity

values will influence on simulation system, comparison tests are implemented with 5%, 10%, 15% and 20%, respectively, with highest mesh level in non-uniform inlet velocity condition.

Table 3.3: Turbulence Intensity Testing

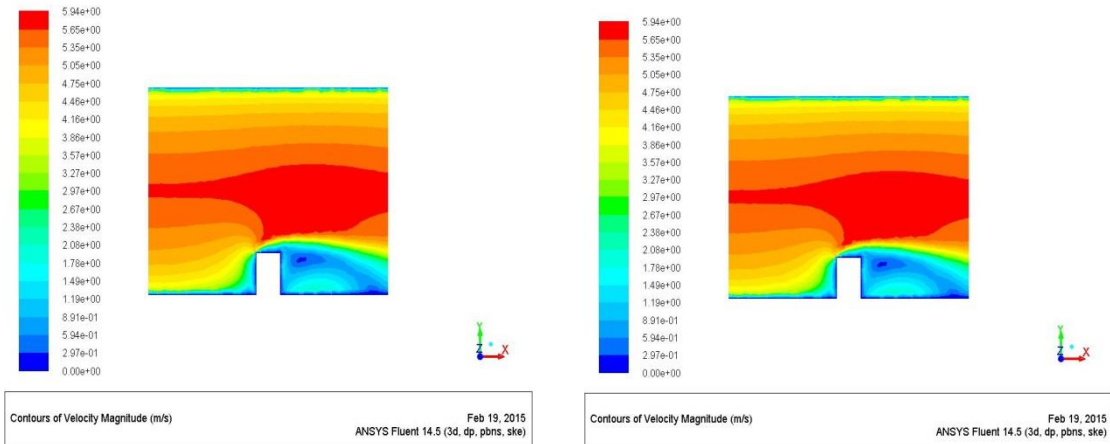
Simulation Number	Mesh level (Tetrahedral cells)	Turbulence intensity	Impending velocity to roof of building [m/s]	Maximum velocity [m/s]	Acceleration factor α
1	459626	5%	5	5.943107	1.1886214
2	459626	10%	5	5.943198	1.1886396
3	459626	15%	5	5.943168	1.1886336
4	459626	20%	5	5.943242	1.1886484

Maximum velocities and acceleration factors are provided in Table 3.3. The variances of both maximum velocities and acceleration factors are within 0.001, which are small enough to be ignored. So, it can be concluded that both parameters are insensitive to turbulence intensities within range from 5% to 20%. On the other hand, from the contours of velocity fields shown in Figure 3.10, these four inlet turbulence intensity inlet boundary conditions showed high consent compared to each other. Based on setup of thesis, it is obvious all concerned major parameters, which include maximum velocity, acceleration factor and velocity field around location of maximum velocity, are independent from the inlet turbulence intensity within range of 5% to 20% completely. In order to achieve consistency, inlet turbulence intensity is defined as 10% in further simulation.



(a)

(b)



(c)

(d)

Figure 3.10: (a) Velocity Contour with Highest Mesh Level for 5% Turbulence Intensity. (b) Velocity Contour with Highest Mesh Level for 10% Turbulence Intensity. (c) Velocity Contour with Highest Mesh Level for 15% Turbulence Intensity. (d) Velocity Contour with Highest Mesh Level for 20% Turbulence Intensity

CHAPTER 4

SIMULATIONS WITH TWIN BUILDINGS

4.1 Double building introduction

Based on previous study, since wind flow over single building achieves an acceleration field over building, which can be regarded as an effective way for renewable energy application, double-building system is another feasible method to get even larger acceleration rate. There will be two kinds of velocity acceleration fields in double-building system. In detail, wind flow speed will be increased over the building and between the gaps of buildings. For example, Bahrain World Trade Center takes advantage of acceleration of gap between twin buildings and benefits huge electricity energy from wind flow. In order to get investigate how double-building influences on wind flow and how the acceleration rate (acceleration factor) compares to those from single building system, FLUENT setup for double-building system should be same as single between, completely. Regarding that dimension of single building is 20 meters long, 20 meters width and 30 meters height, set the initial gap between two buildings as 15 meters. The 3-dimension view of this system is shown in figure 4.1. After certification of mesh independence, widths of gaps are concerned as main variance in this system and several different widths will be implemented for maximum velocity comparison. Through study of different widths of gaps, it can be found the most efficient width that achieves highest wind flow acceleration rate in this setup of system. All double-building systems are simulated with the non-uniform inlet velocity for realistic situation. The only different between single-building and double-building systems is complete structure is used in double-building system, instead symmetry part in testing for single-building system.

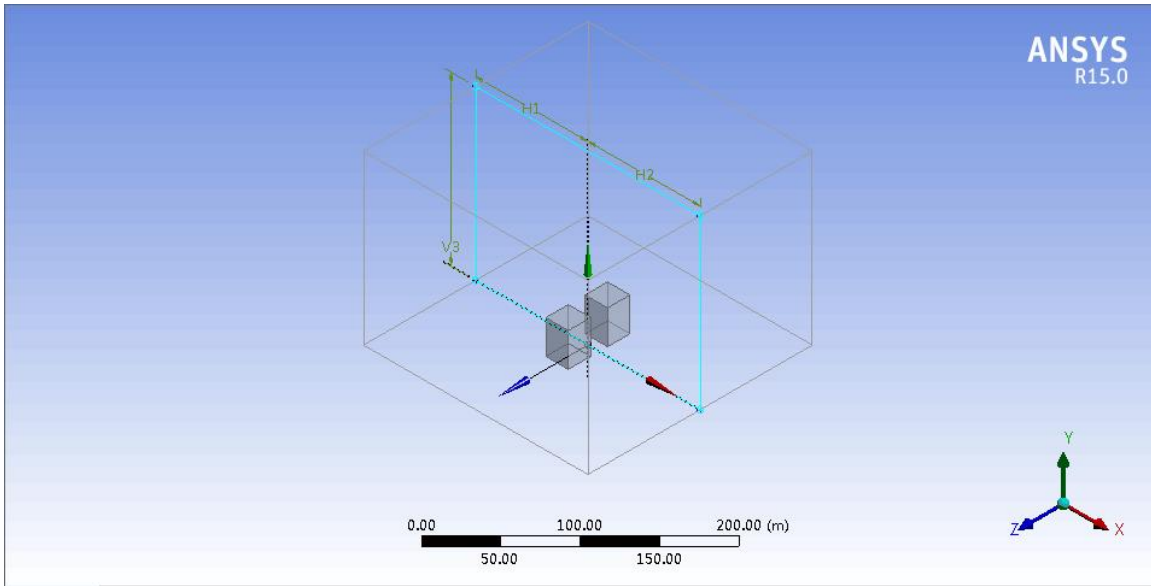


Figure 4.1: Structure of Double Building System, Wind Flow Inlet Is along x-direction (15m)

4.2 Mesh Sensitivity Test

For the turbulence model in large-scale flow, mesh sensitivity should be concerned as a fact of iteration convergence. Before testing and comparing with the different widths of gaps, it is necessary to make sure the result is independent from mesh level. Because of application of wind energy is aimed at the speed acceleration of wind around buildings, the maximum speeds in steady states of the system are compared with different mesh levels. On the other side, turbulence intensity and inlet velocity are controlled to be the same, which are set to be 10% and non-uniform velocity inlet files by User Defined Functions, respectively. Four different mesh levels have been done to investigate the acceleration factors and velocity fields, and the original data of such tests are provided in table 4.1.

Table 4.1: Investigation for Different Mesh Levels (Non-uniform Inlet) with Double Duilding

Simulation Number	Mesh level (Tetrahedral cells)	Turbulence intensity	Impending velocity to roof of building [m/s]	Maximum velocity [m/s]	Acceleration factor α
1	186595	10%	5	6.252235	1.250447
2	383421	10%	5	6.498271	1.2996542
3	663414	10%	5	6.559175	1.311835
4	793551	10%	5	6.519759	1.3039518

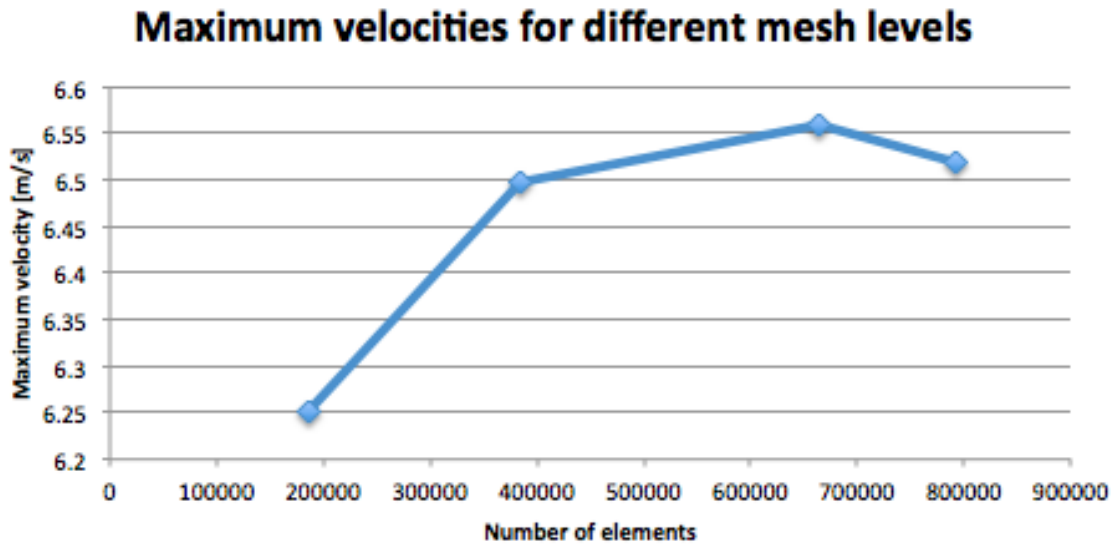
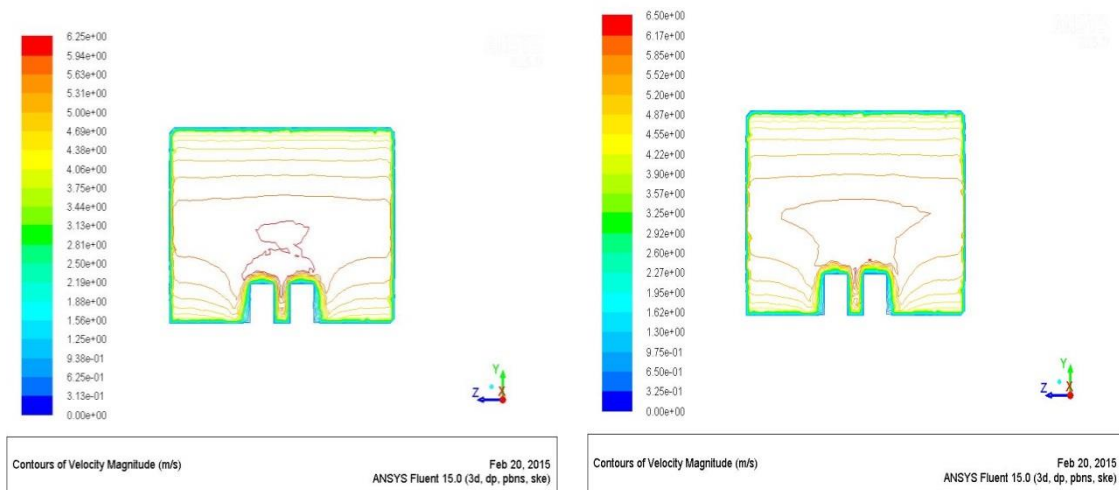


Figure 4.2: Plot for Maximum Velocity with 4 Different Mesh Levels

From the data and plot, it can be observed that, when number of element is more than 40,000, the variance of maximum velocity becomes very small. However, comparing to mesh refinement from single building system, variance from double-building system is 40 times larger than that from single building system with non-uniform inlet velocity. To this situation, there are two major interpretations. Firstly, based on the requirement of simulation, single building system is tested with symmetric part, but whole structure is tested for double-building system for analysis velocity acceleration field between two buildings. That means the simulation structure is twice larger than previous one. In figure 4.2, maximum velocity from the first level has a huge variance with respect to those values when arriving steady state. It indicates that large structure needs more elements for calculation. So, the relative larger variance in steady state is due to the relative coarse mesh intervals. To reduce to variance and arrive at a more steady condition, it needs to additional more computational elements or higher order algorithm for the structure. But it should be concerned that this large scale simulation structure with relative fine mesh level or high order iteration equations will take over much more memory of computer and lead to slow down speed of simulation, which reduces the time commercial efficiency. Secondly, compared with situation of single building, structure of double-building system is more complex. Especially, flow field in the

gap between two buildings is influence by combination of shear flow and turbulence layers from side walls of both buildings. Base on the view along the direction of inlet velocity, which shown for the velocity contours of different mesh level in Figure 4.3, the turbulence layers due to the gap are thick, comparing to the distance of two buildings. The first mesh level shows a completely different structure of velocity field between and over the gap from those from other three higher mesh levels. And the velocity flow of three higher mesh level simulation are relative similar. It can also be observed that even in the highest mesh level in this simulation (4th mesh level), an antisymmetric velocity part is shown on the roof of right building. For this complete symmetry structure and inlet velocity condition, the flow field over the each building is expected same as the other one. This antisymmetric velocity field shows computational error still exists in system. Due to main concern of this simulation is comparing acceleration factors (rates) from different setup and structures, as well as considering the memory limitation of laboratory computer and simulation times, it is roughly regarded that this double-building system arrives steady state when the total number of element is over 65,000.



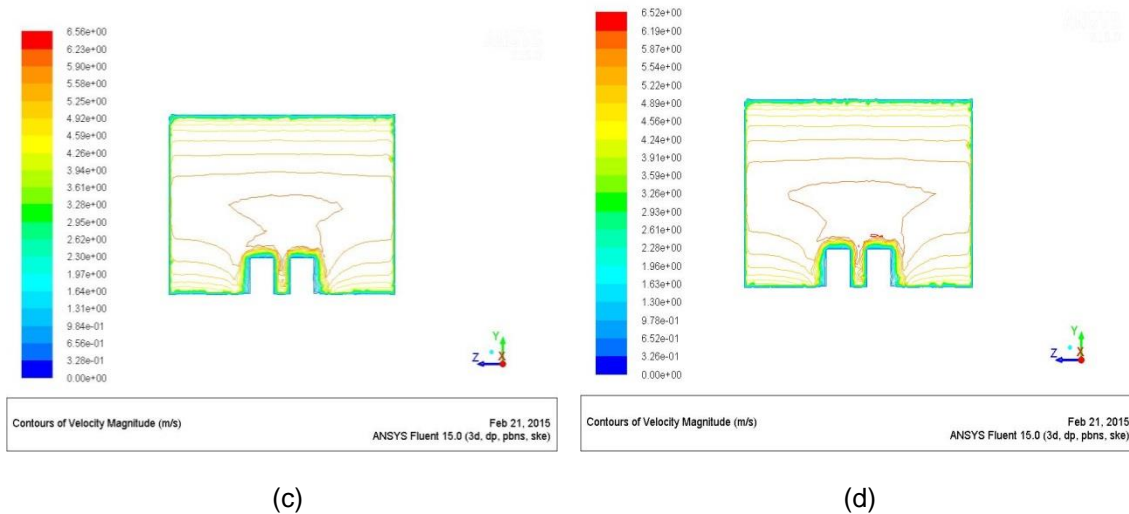


Figure 4.3: (a) Front View of Velocity Field of Double Building System with 1st Mesh Level. (b) Front View of Velocity Field of Double Building System with 2nd Mesh Level. (c) Front View of Velocity Field of Double Building System with 3rd Mesh Level. (d) Front View of Velocity Field of Double Building System with 4th Mesh Level

4.3 Distance Comparison

Based on mesh sensitivity testing on double-building system, which indicates computational iteration convergence will be achieved when the number of elements is over 65,000, it is feasible to test how the distance between two buildings influences on acceleration rate and velocity field. Since length and width of building are both 20 meters and height of building is 30 meter, if a wind turbine is mounted in the gap for renewable energy utilization, it is unrealistic to structure the width of gap over 20 meters. As result, total four different widths of gaps are investigated, which contain width of 5 meters, 10 meter, 15 meters and 20 meters. Same as mesh sensitivity testing, 10% turbulence intensity and non-uniform inlet velocity UDFs file are implemented. In detail, for each simulation, after completing the first round with relative coarse mesh levels, based on velocity gradient, mesh adaption are operated so that number of elements is kept over 65,000 for relative good convergence and compatibility with each other. Figure 4.4 shows the comparison of maximum velocity for single building system and double building system, which contains 4 different distances of gaps. The locations of maximum velocities for all four different widths in double building test are all in the upper part of gap. This phenomenon indicates that the gap can change the maximum location of wind flow from roof of

building to the gap. And Figure 4.4 shows all tested double building have a better acceleration factors than single building, which means efficiency of renewable energy utilization is increased by double-building system. From comparison among four distances, it can also be found that the best acceleration factor is achieved by 10 meters gap and the lowest acceleration factor comes from 5 meters gap. For the other 2 gap-width conditions, which are 15 and 20 meters gaps, are smaller than acceleration factor from 10 meters gap. It shows that if the gap width is larger than 10 meters, the acceleration factor will decrease and lead to a lower efficiency for utilization of wind flow as renewable energy application.

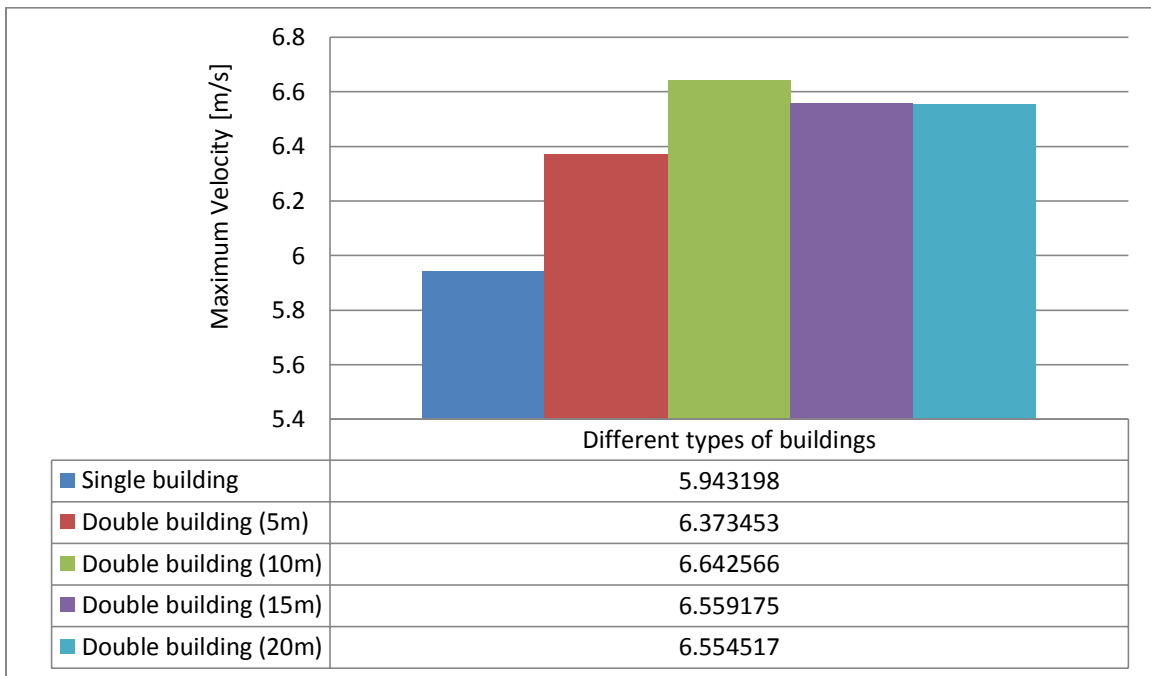
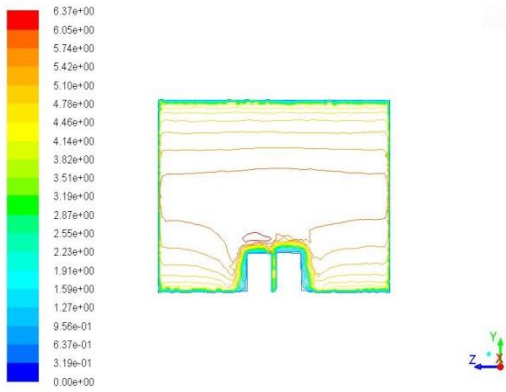
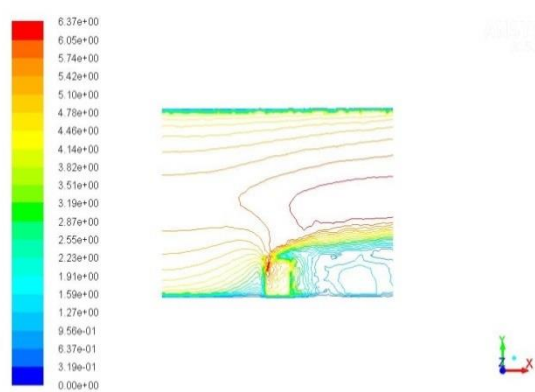


Figure 4.4: Comparison of Maximum Velocities for Single Building System and Double Building System with Four Different Gap Widths



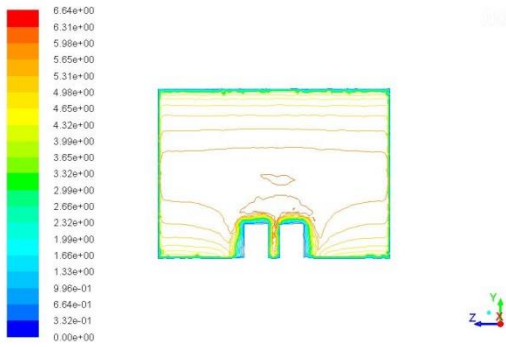
Contours of Velocity Magnitude (m/s) Feb 24, 2015
ANSYS Fluent 15.0 (3d, dp, pbns, ske)

(a)



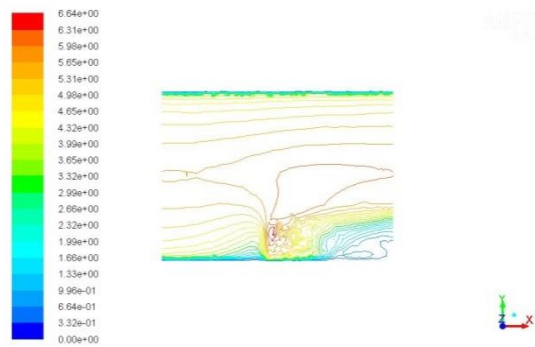
Contours of Velocity Magnitude (m/s) Feb 24, 2015
ANSYS Fluent 15.0 (3d, dp, pbns, ske)

(b)



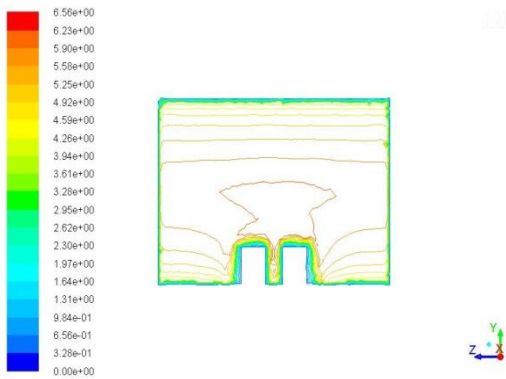
Contours of Velocity Magnitude (m/s) Feb 23, 2015
ANSYS Fluent 15.0 (3d, dp, pbns, ske)

(c)



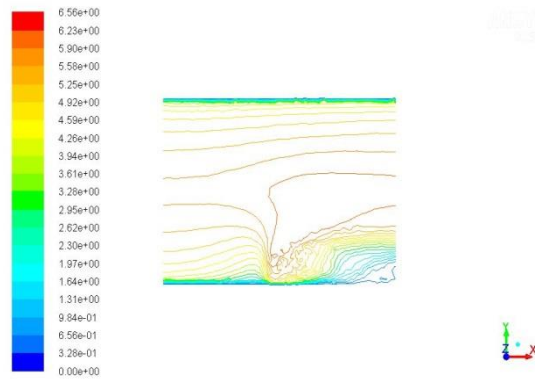
Contours of Velocity Magnitude (m/s) Feb 23, 2015
ANSYS Fluent 15.0 (3d, dp, pbns, ske)

(d)



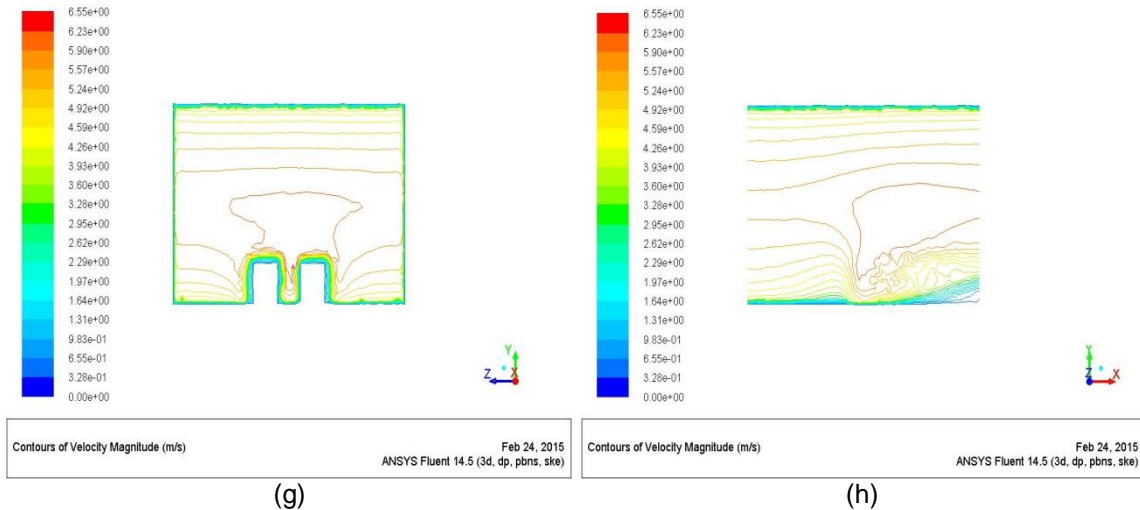
Contours of Velocity Magnitude (m/s) Feb 21, 2015
ANSYS Fluent 15.0 (3d, dp, pbns, ske)

(e)



Contours of Velocity Magnitude (m/s) Feb 21, 2015
ANSYS Fluent 15.0 (3d, dp, pbns, ske)

(f)



(g) (h)
 Figure 4.5: (a) Velocity Contour of Double-building System with 5 Meters Width Gap from Parallel View of Inlet Velocity. (b) Velocity Contour of Double-building System with 5 Meters Width Gap from Vertical View of Inlet Velocity. : (c) Velocity Contour of Double-building System with 10 Meters Width Gap from Parallel View of Inlet Velocity. (d) Velocity Contour of Double-building System with 10 Meters Width Gap from Vertical View of Inlet Velocity. (e) Velocity Contour of Double-building System with 15 Meters Width Gap from Parallel View of Inlet Velocity. (f) Velocity Contour of Double-building System with 15 Meters Width Gap from Vertical View of Inlet Velocity. (g) Velocity Contour of Double-building System with 20 Meters Width Gap from Parallel View of Inlet Velocity. (h) Velocity Contour of Double-building System with 20 Meters Width Gap from Vertical View of Inlet Velocity

From observation of parallel view and vertical view of velocity contour with different gap widths of double-building systems in Figure 4.5, reasons for the different maximum velocities with different gap widths can be interpreted as follows. First, compared with other gap widths, velocity field in the gap of 5 meters is completely different from others, which is more similar with velocity field from single building system. Small gap width leads to cancel out almost the acceleration field between two buildings because turbulence layers over sidewalls of buildings leave no space for acceleration, which would happen over turbulence layers. On the contrary, due to relative thick turbulence layers, comparing to narrow gap, velocity field becomes much more disorder in the gap and decrease the efficiency of wind flow acceleration. Secondly, for the system with gap width of 15 meters and 20 meters, the velocity fields between gaps are much stable than situations in 5 meters and 10 meters systems. That's because relative wide gaps, comparing to boundary turbulence layers over sidewalls of the buildings, reduce influence of interaction of acceleration fields over boundary turbulence layers from sidewalls. So, the maximum velocities are smaller than that from 10 meters width gap and velocity acceleration field is not as

concentrated as that in 10 meters gap system. For realistic application, more concentrated high velocity field will lead even higher utilization efficiency, because equipment for energy storage, conversion and generation can be mounted at best location conveniently. As a result, for condition that wind turbine mounted between the gaps to utilize renewable energy, 10 meters gap in double-building systems should be the best choice, with the basic conditions (like dimension of building and inlet velocity boundary already set in this simulation). For the possibility that wind turbines would be mounted between gaps, as well as over roof of building like single building system to achieve higher energy generation, another concern is how about the acceleration velocity field over roof of building in double-building system. From the result in ANSYS FLUENT, Figure 4.6 figure out the maximum velocity over buildings with different gap widths to compare with maximum velocity over building in single building system. It can be found the maximum velocities for double building system on the roof are all less than that from the single building system, and that variance is increasing with the growth of gap width between two buildings, which means the acceleration efficiency on roof of building becomes lower than original single building system because of existence of gap. In conclusion, the double building system shows significant improvement for the maximum velocity in whole system with respect to original single building design and 10 meters gap achieves the best condition for wind power utilization in this system. The maximum location moves from roof of building to the gap between two buildings, so equipment for renewable energy utilization should be mounted in the gap to achieve highest efficiency. Wind turbines mounted on the Bahrain World Trade Center is an example that wind turbines take advantages of velocity acceleration field in the gap. However, the acceleration factor (rate) is decreased over roof of building in double building system, so, it need to be considered if renewable energy equipment, like wind turbines, should be mounted on roof of building or not.

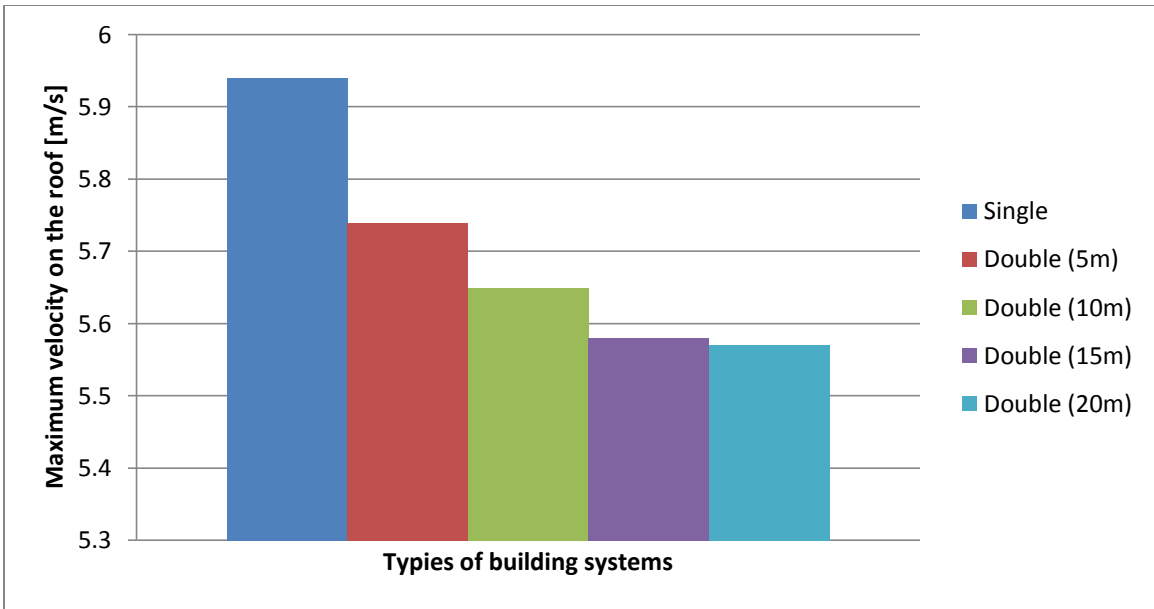


Figure 4.6: Comparison Maximum Velocity over Roof of Building with Single Building Condition and Double Building Condition, Which Contains 4 Different Gap Widths (5 Meters, 10 Meters, 15 Meters and 20 Meters)

CHAPTER 5

INTRODUCTION TO MODIFIED DOUBLE BUILDING SYSTEM, MESH SENSITIVITY STUDY AND IMPROVEMENT STUDY

5.1 Introduction to Modified Double Building System

From the study of single building and double building systems, the maximum velocity is significantly improved by double building systems. In order to take advantage of higher energy utilization efficiency, many works have been done based on the acceleration field in the gap of double buildings. For example, based on Dany Perwita Sari (2014) mentioned, a gap with circular shape round that rooftop was generated to acceleration wind speed for higher usage rate of wind energy. Through numerical and wind tunnel study, the result indicated this structure can achieved additional 55.24% wind speed. In this case, one design of double building structure will be investigated to be compared pristine double building system. Based on pristine double building system, this modified system is designed to improve wind flow between the gaps, which is different form the destination of mentioned paper that aimed at wind flow over rooftop of building. Instead of rectangular shape of both buildings, the walls, which opposite to each other, are added smooth cycle round. As shown in Figure 5.1, the basic structure is completely the same as pristine double building systems. A 20 meters diameter cylinder, which direction is parallel to the building, is generator and its center is 10 meters away from the symmetry plane. And the distance of two rectangular parts is set to be 20 meters. This structure is expected to increase the acceleration rate and concentrate the velocity field between two buildings, which can get the better improvement than the small circular gap mounted on the top of building. In order to keep consistency, setup in ANSYS FLUENT is same as double building system. Inlet turbulence intensity is set to be 10% and outlet is the pressure-based outlet boundary condition. On the other hand, the inlet velocity is non-uniform inlet velocity mentioned before, and the input file for the UDFs is the same C program as used in previous tests. All other walls are set to be stationary, no-slip wall. The outside boundaries are also set to be no-slip wall; which dimension is 200 meters in length, 200 meters in width and 150 meters height, same as pristine test. Before

comparing with previous tests, it is necessary to deal with mesh to make sure the result is independent from mesh level.

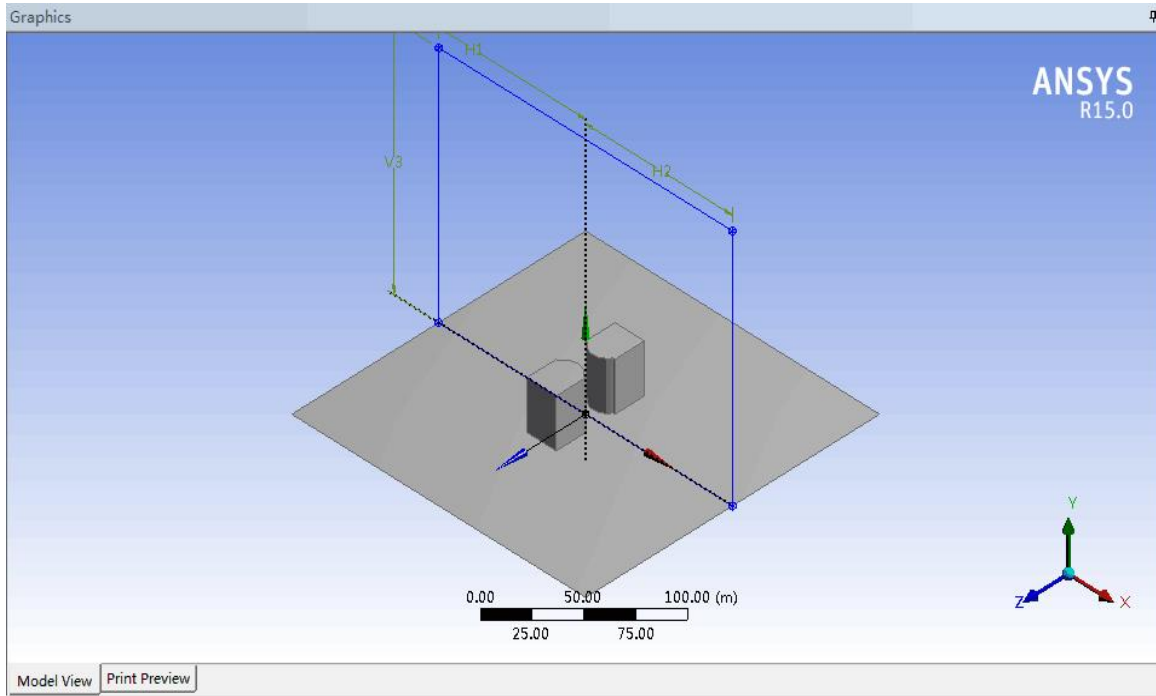


Figure 5.1: Structure of Modified Double Building System with Circular Shape Gap

5.2 Mesh Sensitivity Study

For the turbulence model in large-scale flow, mesh sensitivity should be concerned as a fact of iteration convergence. For the reliability of result from any simulation, it is necessary to make sure the result is independent from mesh level. Because of application of wind energy is aimed at the speed acceleration of wind around buildings, the maximum speeds in steady states of the system are compared with different mesh levels. If the result shows variance small enough to be ignored, it can be regarded that the wind flow system is converged. On the other side, turbulence intensity and inlet velocity are controlled to be the same, which are set to be 10% and non-uniform velocity inlet files by User Defined Functions, respectively. Five different mesh levels have been done to investigate the acceleration factors and velocity fields with the distance between the 10 meters gap between buildings, and the original data of such tests are provided in table 5.1.

Table 5.1: Mesh Study for Modified Double Building System

Simulation number	Number of elements (Tetrahedral cells)	Maximum velocity (m/s)	Impending velocity to roof of building [m/s]	Acceleration factor
1	191828	7.868616	5	1.5737232
2	461160	8.460300	5	1.69206
3	612514	8.780697	5	1.7561394
4	711361	8.748717	5	1.7497434
5	780262	8.741863	5	1.7483726

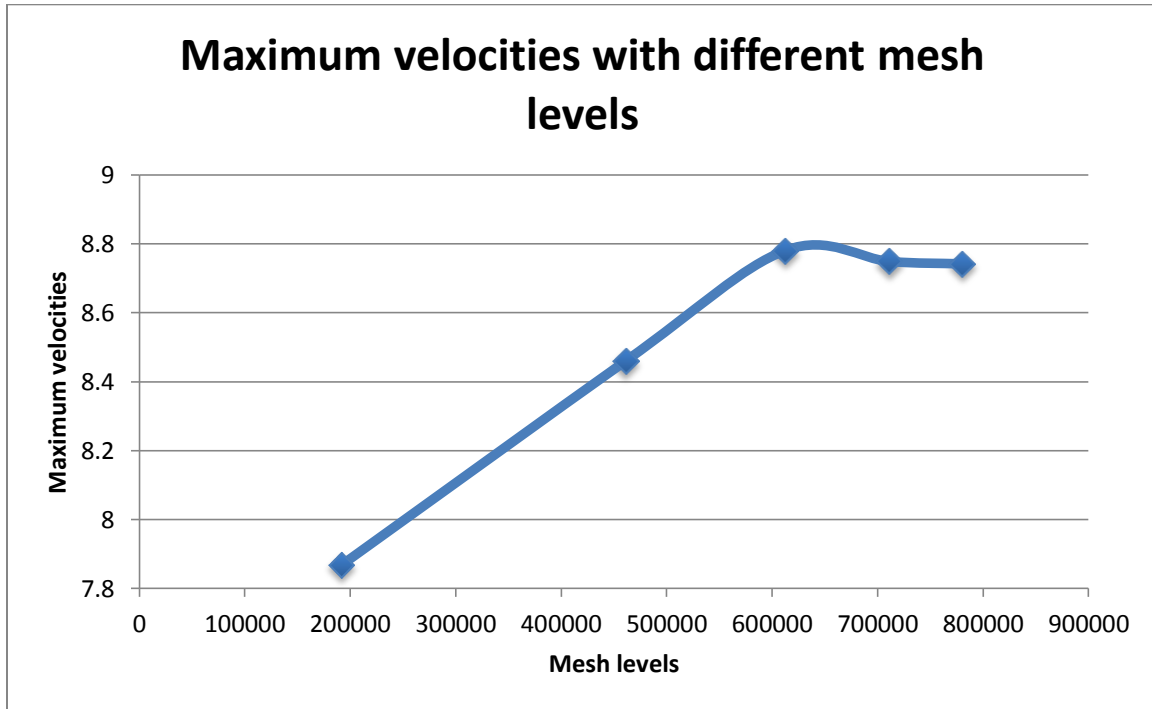
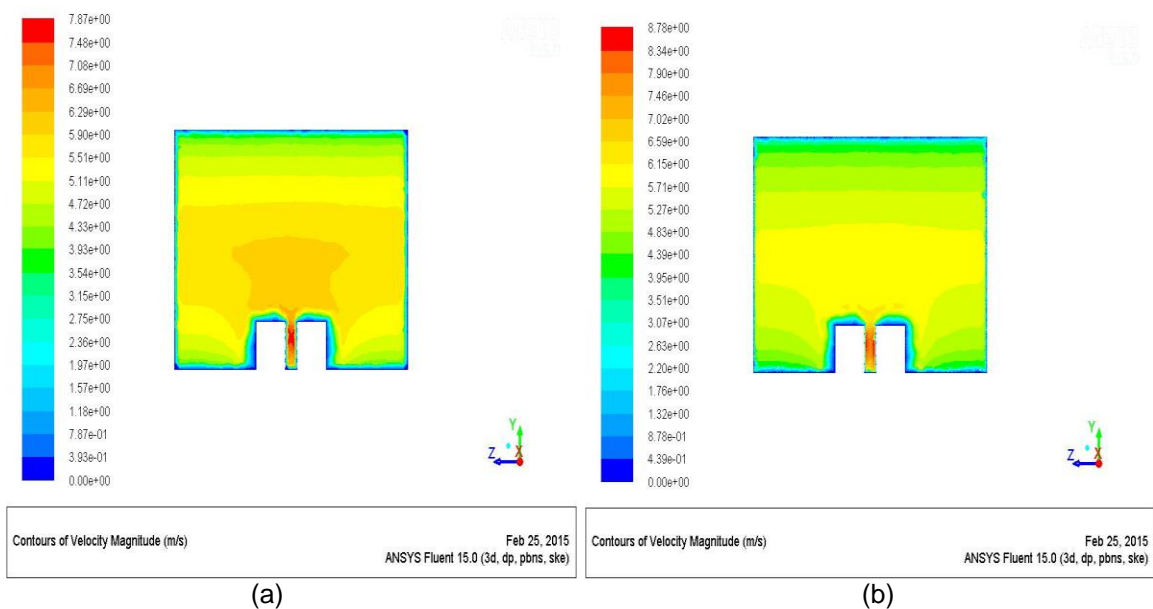


Figure 5.2: Plot for Maximum Velocity with 4 Different Mesh Levels

From the data and plot in Figure 5.2, it can be observed that, when number of element is more than 60,000, the variance of maximum velocity becomes very small. However, comparing to mesh refinement from single building system, variance from modified double-building system is 5 times smaller than that from original double building system with non-uniform inlet velocity. To this situation, there are two major interpretations. Firstly, compared with the convergence condition from original double system, modified double system has a better convergence condition. It indicates wind flow in modified double building system is steadier than the original one, which means circular shape can make more steady flow condition for renewable energy

utilization. At this point of view, it also can be found such dissimilarity from velocity field in contour of front view of wind flow in Figure 5.3. In original double building system, the front view of this contour shows large difference with respect to different mesh level. However, for modified double building system, the velocity field contour shows high consistency with relative high mesh level, and the rooftop anti-symmetric part is also very small. Secondly, from the observation of convergence history, huge error appears when the mesh level is under 40,000. Base on the contour of Figure 5.3, there is a obvious flow field with around 6 m/s over two buildings, this region cannot be found at the same location in higher mesh levels. On the other hand, the relative high velocity region in the gap has larger vertical length comparing to same location in higher mesh levels. This situation share the same reason with convergence interpretation of pristine double building system, which is due to complete structure is also implemented in modified double building system. It needs a higher mesh level or high order algorithm to get accuracy result comparing to single building system with symmetric structure for simulation. With concern of calculation time and computer memory limitation, the total element is controlled to be less than 80,000. For the reasons mentioned above and a relative high quality converged condition, this modified double building system is regarded that reliable result is achieved with the highest mesh level.



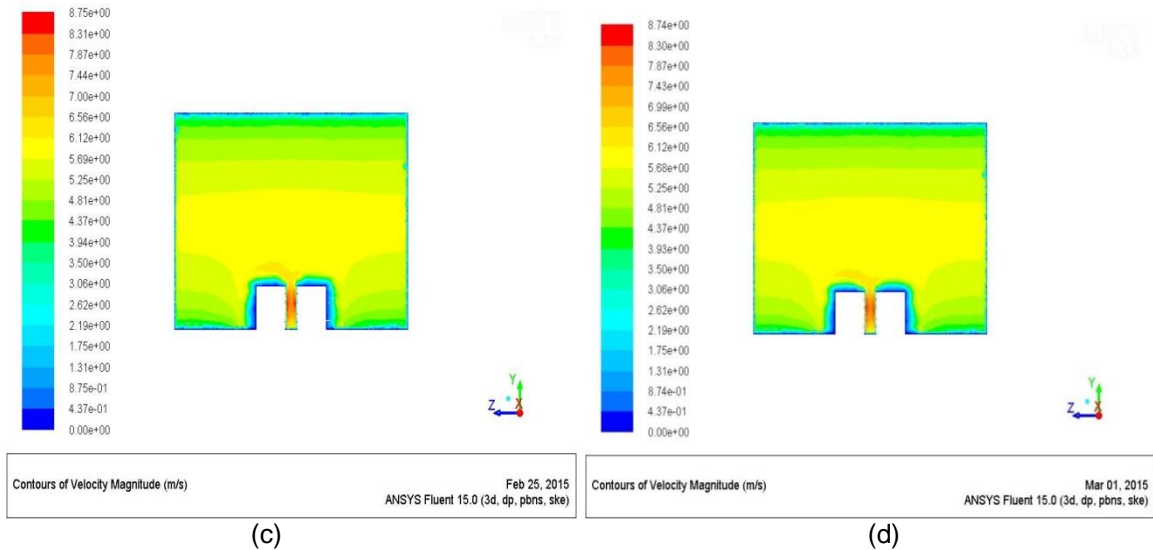


Figure 5.3: (a), (b), (c), (d) Are Front View of Velocity Field of Modified Circular Double Building System with 1st Mesh Level, 3rd Mesh Level, 4th Mesh Level, 5th Mesh Level, Respectively

5.3 Improvement Study

To certify the improvement of modified double building system, three different comparisons will be investigated, which contain maximum velocity field comparison, rooftop maximum velocity comparison and maximum velocity field comparison. In addition, the feasibility of circular shape double building wind flow system will be analyzed, which shows the realistic meanings for this improvement.

Comparing with velocity field in pristine double building system, the maximum velocity is located at the gap between two buildings as well. Three specific conditions, which are single building system, double building system with 10 meters gap and modified building system with 10 meters gaps, are compared with the maximum velocities and the comparison plot is shown in the figure 5.4. If maximum velocity in single building system is regarded as reference for comparison, pristine double building improves maximum velocity in the gap with 0.699368 m/s (from 5.943198 m/s to 6.642566 m/s). However, the modified double building system can increase maximum wind speed in the gap with 2.798665 m/s (from 5.943198 m/s to 8.741863 m/s). Modified double building shows extremely great improvement for maximum velocity than pristine one, which is roughly 4 times larger. It can be interpreted the circular shape improves velocity field and make

the wind flow between two buildings increase much easier. On the other hand, comparing with the result Dany provided in circular rooftop building system, circular gap double building system provides additional 74.83% wind flow velocity and the circular rooftop building system provides additional 50.00% wind flow velocity. But in the setup for circular rooftop building system, the height of building is 64 meters and normalized velocity is 10 meters per second, which is much larger than parameters in this modified double building systems. Even compared to a higher buildings and larger inlet velocity, the values of acceleration ratio for modified double building system are higher. So, it can be concluded that the modified circular double building system has a significant improvement for wind flow speed acceleration in the gap between two buildings.

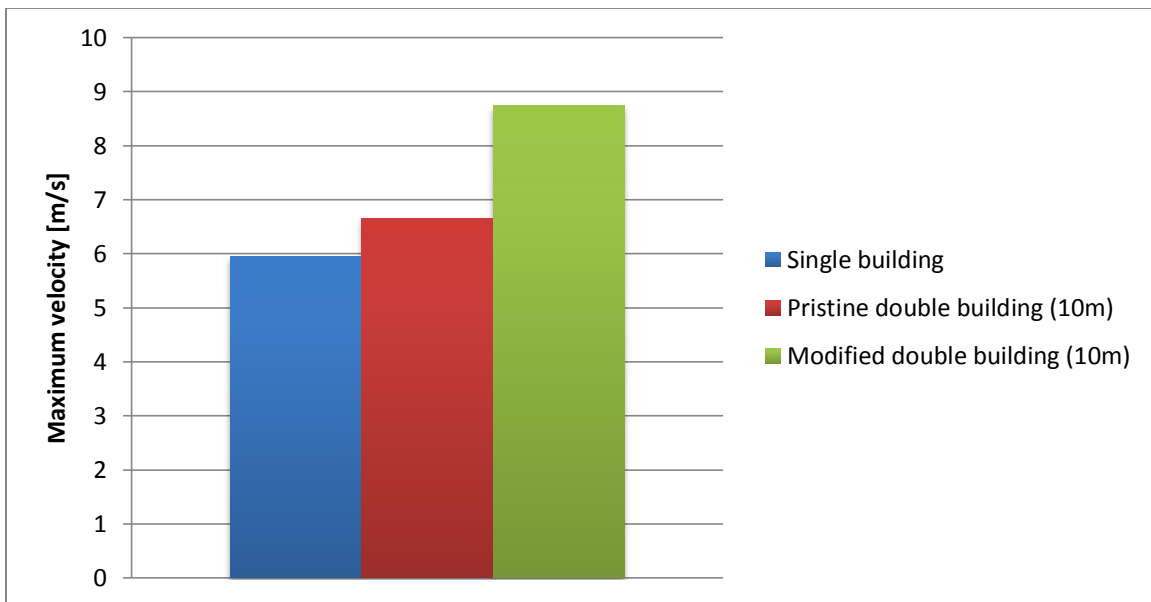


Figure 5.4: Maximum Velocity in Modified Circular Double Building System Compared with Maximum Velocities from Single and Pristine Double Building Systems

On the other hand, it shows different velocity fields between pristine double building system and modified circular double building system. Figure 5.5 provides contours for front and side view from both pristine and circular double building systems. It can be observed that lines in the Figure 5.5 (b), which is from circular double building system, is much more smooth and the area of maximum velocities are more concentrated than that from pristine double building system. It indicates that circular shape gap not only can improve efficiency for wind speed accelerating but also can limit the maximum velocity field into a relative small region. There is another

observation from front view of contours that the location of maximum velocity in circular double building system is near the walls of building. This is because circular shape reduces turbulence boundary layers so that costs for installation and maintenance can be decreased significantly. The condition, which contains relative higher maximum velocity and smaller velocity field, can make the utilization for wind energy by wind turbine much easier.

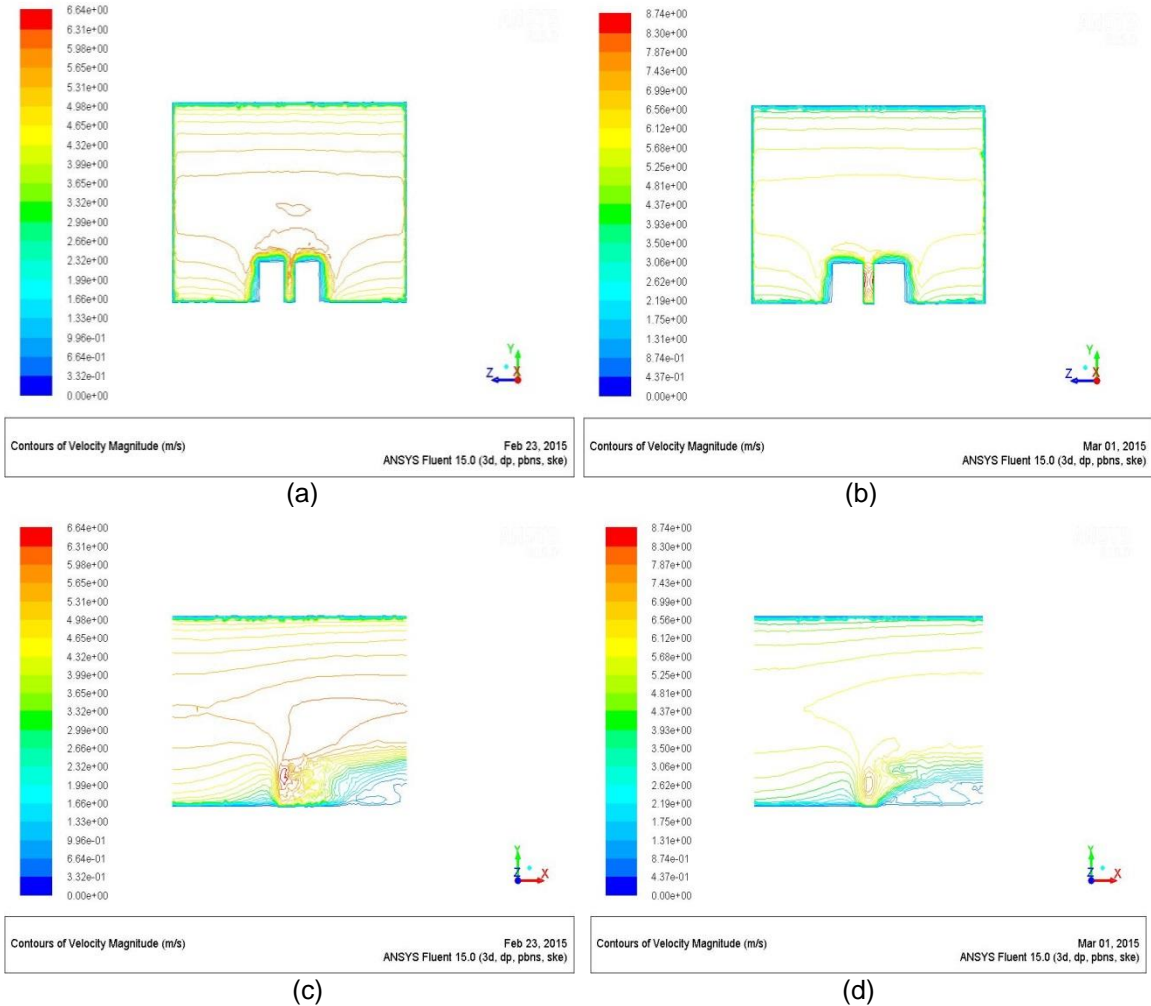


Figure 5.5: (a), (b) Are Front Views of Pristine Double Building Systems and Circular Gap Double Building Systems, Respectively. (c), (d) Are Side Views, Parallel to the Inlet Velocity, of Pristine Double Building Systems and Circular Gap Double Building Systems, Respectively

In addition, as concern in pristine double building system, the acceleration factors should be concerned in modified circular double building systems. In some situation, the wind turbine would be mounted on rooftop of building only or work as a combination of wind turbines mounted on rooftop and gap. So, the acceleration factor at rooftop of buildings is also regarded as another

important factor in wind energy utilization. Figure 5.6 shows the comparison for maximum velocity on the rooftop in three different conditions, which contains single building system, double building system and circular gap double building system. It can be found that rooftop maximum velocity from circular gap building system is not decreased like original building system, however, compared with single building system, maximum velocity on the roof is increased approximately 3.00%, which is from 5.94 m/s to 6.12 m/s. Compared with totally 47.09% increasing of gap maximum velocity, although the ratio of improvement is not as much as that at gap between buildings, wind flow over rooftop is also in a better condition for circular shape gap double building system. In conclusion, the circular gap double building system improves wind flow over buildings in three sides; maximum velocity, rooftop velocity and velocity field concentration. It cancels the drawback of pristine double building system, which has a relative weak improvement for wind flow over rooftop. So, this design for double building system with circular shape can be regarded as one of the feasible improvement methods for wind energy utilization.

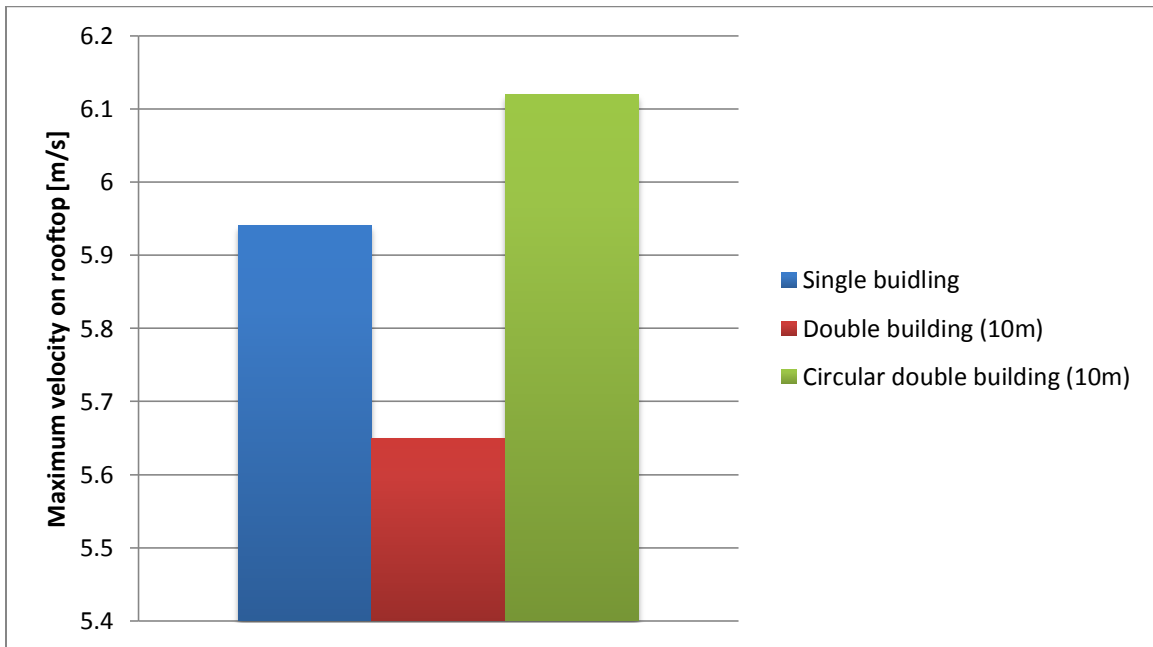


Figure 5.6: Maximum Velocity on Rooftop in Modified Circular Double Building System Compared with Maximum Velocities from Single and Pristine Double Building Systems

CHAPTER 6

SUMMARY

In this thesis, totally three main structures of environment flow over buildings are tested, which include single building, double building and modified circular gap double building systems. For each structure, mesh sensitivity is tested and analyzed. Because of large dimensional environment and limitation of computer memory, solution does not converged completely. Reliable result is obtained when convergence curve is steady as well as the maximum velocity and shapes of velocity field are independent from different mesh levels. Different variables and situations are compared with each other in single building system. After comparison, standard $k - \varepsilon$ turbulence model is determined for further simulation. It is found that inlet turbulent intensity does not influence on result in this simulation from the range 5% to 20%, and 10% inlet turbulent intensity is chosen for consistency in other simulations. On the other hand, for a more realistic inlet wind speed boundary condition, uniform inlet velocity and non-uniform inlet velocity with Power Law Theory are compared. Based on realistic urban condition, non-uniform inlet velocity is selected as velocity boundary condition in further simulations. In double building system, each building has exactly the same dimension as that in single building system. After mesh sensitive study for the first structure, which has a 15 meters gap between two buildings, the result shows double building can achieved a higher velocity than single building system. But the maximum velocity location moves from top of building to gap between buildings. At the same time, velocity field over rooftop of double building system shows a small decreasing compared with that in single building system. Then, 4 different distances, which contains 5, 10, 15 and 20 meters, are tested and compared to each other. The maximum velocity is achieved in double building system with 10 meters gap between two buildings. In order to improve acceleration factor in double building system, new design is generated, which is named as circular gap double building system. The basic structure of this modified building system is the same as pristine double building systems, but a circular shape structure is added in the gap as shown before. For consistency, the distance of gap is controlled to be 10 meters. Result shows that, not only in the gap but also over rooftop, circular gap double building system achieve a higher improvement for

wind speed and high velocity location than both single building system and pristine double building system. So, circular gap double building system can be used as a higher efficient structure for wind energy application.

However, it should be noticed that the value of velocity is not the only concern when choosing the location of wind turbine. Two more major variables should be considered as well. Firstly, based on the theory of the first law of thermodynamics, the work, which done by wind flow, depends on difference between the inlet pressure and outlet pressure. Higher pressure difference leads to relatively more work being done. Because for some locations that wind velocity is fast, the pressure difference may be relatively low, wind turbine should not be set at such locations. On the other hand, for wind energy application in urban condition, different area has different temperature, and the temperature of air affects the velocity as well. In this simulation, the thermal effect is turned off and the temperature is regarded keeping the same. For a more realistic wind flow simulation, the temperature should be considered due to its influence on the velocity. For example, in an open boundary condition area, comparing to lower temperature, a relatively higher temperature will reduce the velocity at the same location. In addition, temperature in real world has different values at different heights, so this condition leads to more complex velocity distribution when the wind flow in front of buildings, especially relatively high buildings.

For further study, cause the gap in this system is refined at right side and left side where the wind turbine will locate, if two beams with circular shape joint two building at the location above and below the location of wind turbine, the acceleration factor would be higher. And effects of temperature and pressure are also recommended to be considered for further testing. For these tests, the simulation will has a more realistic meaning for wind energy application

REFERENCES

- Tribal Energy and Environmental Information CLEARINGHOUSE: Ways to Reduce Carbon Dioxide Emissions, <http://teeic.indianaffairs.gov/er/carbon/carboninfo/reduce/index.htm>.
- Lin Lu, Ka Yan Ip, 2007: Investigation on the feasibility and enhancement methods of wind power utilization in high-rise buildings of Hong Kong, *Renewable and Sustainable Energy Reviews*, **13**, 450-461.
- Sinisa Stankovic, *et al.* 2009, "Urban Wind Energy", Earthscan, UK and USA.
- Dany Perwita Sari, Kang-Pyo Cho, 2014: CFD and Wind Tunnel Analysis for Mounted-wind Turbine in a Tall Building for Power Generation, *Mechatronics, Electrical Power, and Vehicular Technology*, **05**, 45-50.
- Bahrain World Trade Center: http://en.wikipedia.org/wiki/Bahrain_World_Trade_Center.
- ASU Wind Turbine Generate Electricity and Interest: <https://sustainability.asu.edu/news/archive/wind-turbines>.
- Pacific Rim Construction, 2012: Atkins- 10 years, 10 cities.
- Dursun Ayhan, Safak Saglam, 2012: A technical review of building-mounted wind power systems and a sample simulation model, *Renewable and Sustainable Energy Reviews*, **16**, 1040-1049.
- Islam Abohela, *et al.* 2013: Effect of roof shape, wind direction, building height and urban configuration on the energy yield and position of rood mounted wind turbines, *Renewable Energy*, **50**, 1106-1118.
- Shuzo Murakami, 1993: Comparison of various turbulence models applied to a bluff body, *Wind Engineering and Industrial Aerodynamics*, **46 & 47**, 21-36.
- Robert Meroney, *et al.* 1999: Wind-tunnel and numerical modeling of flow and dispersion about several building shapes, *Wind Engineering and Industrial Aerodynamics*, **81**, 333-345.
- Bernd Leitl, *et al.* 1997: Concentration and flow distributions in the vicinity of U-shaped buildings: Wind-tunnel and computational data, *wind Engineering and Industrial Aerodynamics*, **67 & 68**, 745-755.
- ANSYS, Inc. (2012): FLUENT Tutorial Guide, Chapter 3: Introduction to using ANSYS FLUENT: Fluid Flow and Heat Transfer in a Mixing Elbow.
- Ernest Peterson and Joseph. Hennessey Jr. (1978), "On the use of power laws for estimates of wind power potential," *Journal of Applied Meteorology*, vol. **17**, no. 3, 390-394.
- CFD Online, Turbulence intensity, http://www.cfd-online.com/Wiki/Turbulence_intensity.

APPENDIX A

NON-UNIFORM INLET VELOCITIES WITH MATLAB CODES

```
% One-seven Power Law
clear
z=0:0.1:75;
% z is the domain of the height
u=zeros(0,length(z));
% u is the inlet velocity
z1=30;
% z1 is the height of inlet domain
u1=5;
% u1 is velocity at the upper boundary
p=1/7;
% one-seven power law coefficient
for i=1:length(z)
    u(i)=u1*(z(i)/z1)^p;
end
plot(u,z)
xlabel('Inlet velocity [m/s]')
ylabel('location [m]')
```

APPENDIX B

NON-UNIFORM INLET VELOCITIES WITH C PROGRAM CODES

In this part, the non-uniform inlet velocity file used in UDFs of ANSYS FLUENT is shown below. This file is compiled with c programming. Firstly, the function is defined in vertical domain with the equations derived in the previous chapters. Then, the velocity at the middle point of vertical domain is defined. Through implementation of power law equation, velocity of the lower half domain can be found, and velocity of upper half domain is defined as the symmetry part of lower part.

C program code is shown below

```
//
// udf.c
// thesis
//
// Created by IIGuoyi on 14-10-20.
// Copyright (c) 2014 年 IIGuoyi. All rights reserved.
//

#include "udf.h"
#define ymin 0.0
#define ymax 150.0
#define b 1./7.
#define DELOVRH 0.5

DEFINE_PROFILE(x_velocity,t,i)
{
    real y;
    real del;
    real h;
    real x[ND_ND];
    real ufree;
    face_t f;

    h=ymax-ymin;
    del=DELOVRH*h;
    ufree=5.69926;

    begin_f_loop(f,t)
    {
        F_CENTROID(x,f,t);
        y=x[1];
        if (y<=del)
            F_PROFILE(f,t,i)=ufree*pow(y/del,b);
        else
            F_PROFILE(f,t,i)=ufree*pow((h-y)/del,b);
    }
    end_f_loop(f,t)
}
```