American Scientific Research Journal for Engineering, Technology, and Sciences (ASRJETS)
ISSN (Print) 2313-4410, ISSN (Online) 2313-4402

http://asrjetsjournal.org/

Study on Wind Flow Behaviours of High Rise Buildings with CFD Simulation

Thet Mon Soe[a*], San Yu Khaing[b], Kyi Kyi Thein[c]

[a]*Associate Professor, Civil Engineering Department, Mandalay Technological University, Mandalay, Myanmar*
[b]*Professor, Civil Engineering Department, Mandalay Technological University, Mandalay, Myanmar*
[c]*Lecturer, Civil Engineering Department, Mandalay Technological University, Mandalay, Myanmar*
[a]*Email: thetmonsoe.658@gmail.com*
[b]*Email: sykpku@gmail.com*
[c]*Email: kyikyithein@mtu.mm*

Abstract

Wind distribution flows are very important for designer to develop new urban planning and new design buildings. The aim of this paper research was to investigate wind flow behaviours around the buildings cluster. Present work used a three dimensional scale down model of buildings where transient flow analysis was done. CFD (Computational Fluid Dynamics) simulation is the main research method to investigate the wind environment around building complexes. Numerical investigation of air flow pattern around a cluster of existing high rise structures located in Mandalay city was carried out. It has been implemented through ANSYS Fluent 17.0 using SIMPLE algorithm as solver. Standard k- ε model was used for turbulence modelling. The inflow Basic Wind Speed in Mandalay is 80 mph. The inflow wind velocity profile with height is computed by power law equation and imposed by UDF (C+ program). This simulation was carried out to study the effect of wind directions on velocity distribution around the structures and wind pressure coefficient on the face of the L shape building. Analysis was performed at eight wind directions such as N, NE, E, SE, S, SW, W and NW. In these simulation results, high stream line velocities are especially entering into the east side of the Condo A building as obstacles on the wind pathway from the interval between Tower building and L-shape building. When wind strikes building, causing a positive pressure zone to be formed on the windward face and a negative pressure zone is created at the sides and leeward face of the building.

Keywords: CFD simulation; turbulence modeling; buildings cluster; velocity distribution; wind directions.

----

* Corresponding author.

1. Introduction

With the development in technology, high rise structures are being designed and constructed to care of the local need and desire. Such structures have a significant effect on the surrounding wind patterns. Wind is caused by differences in pressure. When a difference in pressure exists, the air is accelerated from higher to lower pressure. High wind acting on the buildings has become one of the main challenges facing structural designers, and so, structural designers of the high rise buildings must take into consideration the wind loads as well. From the structural design point of view, a high rise building is more affected by lateral loads created by wind or earthquake actions compared to other building types. The response of a building to wind excitation depends on characteristics of the approach wind, building size and shape, distribution of mass and the stiffness, ability of the structural system to dissipate vibration.

On the other hand, the surrounding topography also influences such aerodynamic loads on high rise buildings, and the neighboring buildings may increase or decrease wind loads, depending on the relative location to the measured building. The construction of a new building alters the microclimate in its vicinity; hence wind comfort and safety for pedestrians become important requirements in urban planning and design. High rise buildings in urban areas should be designed to ensure comfort of their inhabitants and users. The construction of a building inevitably changes the outdoor environment around the building. These changes include wind speed, wind direction, air pollution [1].

The generalized estimation of wind loading is carried out by defining pressure coefficients. Pressure coefficients are non- dimensional parameter which is used to assess magnitude. Pressure coefficients are influenced by various parameters like shape, structural geometry, incident wind profile, turbulence in the wind, terrain roughness, location of a particular structure etc [2].

Flow structure and wind pressure distribution caused by obstacles are usually the focuses in Computational Wind Engineer researches. CWE as a branch of Computational Fluid Dynamics (CFD) has been developed rapidly to evaluate the interaction Computational Fluid Dynamics. (CFD) simulations are being widely used by engineers for various wind engineering studies such as determine wind loads on buildings, evaluating wind flow patterns in built areas, evaluate pedestrian level wind comforts, etc. The first CFD techniques were introduced in the early 1950s, made possible by the advent of the digital computer. CFD is a computer-based mathematical modeling tool capable of dealing with fluid flow problems and predicting physical fluid flows and heat transfer [3]. CFD techniques have been adopted for the estimation of wind flow behaviours around building.

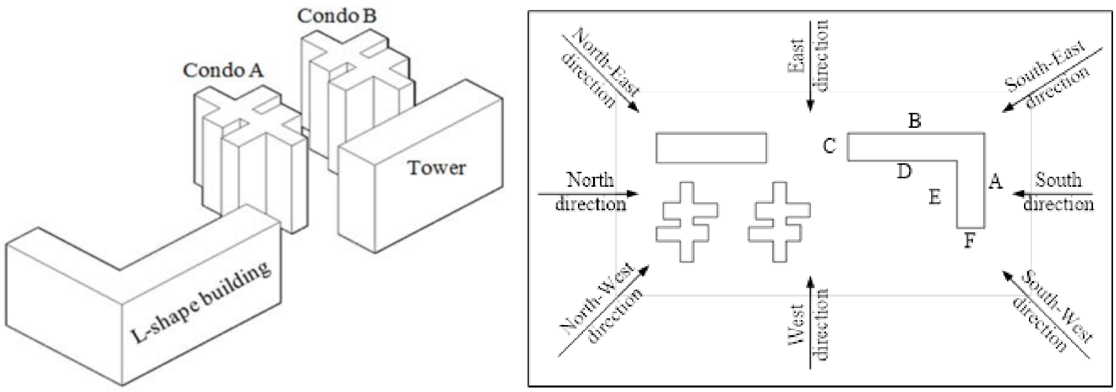
In Mandalay City, there are increasing numbers of high construction and complex forms which can involve problems of significant wind discomfort around buildings. So, wind flow patterns and wind pressure coefficient around buildings with various wind direction have been investigated in this paper.

2. Numerical Simulation

When performing a simulation, the turbulence model, the level of detail in the geometrical of the case study buildings, the size of the computational domain, the type of the computational grid, the boundary conditions, the discretization schemes, and the iterative convergence criteria are chosen.

2.1. Descriptions of a Case Study

In this study, the existing cluster of high rise structures located the corner of 73st street and Mingalar street 2 in Mandalay. It is consist of L shape building, Condo A building, Condo B building and Tower building. The Location of the case study, name of the different faces of the L shape building such as Face A, B, C, D, E, F and position of eight wind directions are described in Figure. 1.



(a) (b)

Figure 1: (a) Name of the buildings (b) Name of the different faces of the L shape building and wind directions

2.2. Computational Domain

The size of the entire computational domain depends on the targeted area and the boundary conditions. For the size of the computational domain, the best practice guidelines recommended by Franke and his colleagues (2007) and Tominaga and his colleagues (2008) were employed. The blocking ratio should not exceed 3% [4]. The blocking ratio is the ratio between the buildings vertical surfaces exposed to the wind and the surface formed by the height and width of the simulated field which is generally the air inlet surface in the simulation. A blocking ratio less than 3% is recommended, even for large groups of buildings. The shape of the section of the simulation volume should preferably follow that of the buildings vertical surface exposed to the wind [5].

For the lateral boundaries, 10H is required between the buildings sidewalls and the edge of the computational domain. The upstream distance from the front side of the L shape building to the inlet boundary was selected as 5H. Outflow boundary is defined 25H downstream of the building. In this study, the height of the computational domain is selected 5H with a maximum blockage ratio of 2.4%, where H is the height of the L shape building. The dimensions of the computational domain size and arbitrary domain size are shown in Table 1 and Figure 2. B and L are the dimensions of L shape building, respectively.

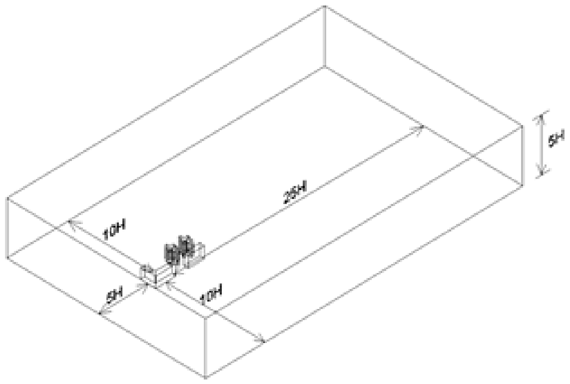


Figure 2: Arbitrary domain size

Table 1: Dimensions of the computation domain size for case study

Computational Domain Size	10H+B+10H	5H+L+25H	5H
Full dimension (m)	982.45	982.45	228.5

2.3. Meshing

Meshing is another important stage of a CFD simulation, which corresponds to accurate simulation of the atmosphere boundary layer as well as fluid motion near the ground. Therefore, some of the meshing parameters such as first node height, number of nodes within boundary layer should be kept within certain limits in order to perform successful simulations [6]. In this study, this structured mesh has over 2930000 hexahedral cells with MultiZone Meshing Method. The maximum cell size for all three domains was 0.02 m and minimum cell size was 0.001 m.

2.4. Boundary Conditions

Boundary conditions represent the influence of surroundings that have been cut off by the computational domain. The boundary conditions for inlet, outlet and outer walls should be provided. These boundary conditions may be of the values for velocity, pressure or mass flow rates etc [7]. In these simulations, the inlet boundary conditions were specified by using a user defined function for the power law ABL profile. The Basic Wind Speed is 35.76 m/s at standard height in Mandalay [8]. The power law equation and input parameters are shown in Table 2.

2.5. Numerical Setup

The commercial CFD code Fluent was used to perform the simulations. The 3D steady RANS equations were solved in combination with Standard k-ε model. Standard k-ε model is used for turbulence modelling. SIMPLE algorithm is used for pressure-velocity coupling. As pressure based solver is used, second-order upwind discretization schemes are used for the convection terms and the viscous terms of the governing equations, i.e.

conservation of mass and momentum equations.

Table 2: Power Law equation and input paramrters

S.N	parameter	Value
1	Vertical Velocity Profile	35.76 x $(\frac{z}{10})^{1/7}$
2	Basic Wind Speed (m/s)	35.76
3	Power Law Coefficient, α	1/7
4	Density of air (kg/m^3)	1.225
5	Viscosity of air (kg/m-s)	$1.7594*10^{-5}$
6	Solver	Pressure –based steady state
7	Models	standard k-ε model
8	Solution Method	SIMPLE pressure velocity coupling

2.6. Governing Equations for fluid flow

Wind engineers study more about the lower part of the atmosphere though entire earth atmosphere extends few kilometres above the earth surface. This lower parts of the atmosphere is called as atmospheric boundary layer. This boundary layer depth can be varying from several hundred meters to more than a kilometre aloft. Thus most of manmade structures are well within the atmospheric boundary layer, governing flow equations can apply in this layer easily. Most of governing equations in fluid dynamics can be applied to the atmospheric flows. The main Governing equations are about conservation of mass and momentum.

$$\frac{\partial \rho}{\partial t}+\frac{\partial\left(\rho \mu_i\right)}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial \rho \mu_i}{\partial t}+\frac{\partial\left(\rho \mu_i \mu_j\right)}{\partial x_j}= -\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

The widely used of turbulence model is standard k-ε model developed by Hanjalic and Launder [9]. It has been verified and validated for a wide variety of flows and relatively low computational costs. This two equation model characteristics small scale turbulence by using two numbers kinetic energy (k), and energy dissipation rate(ε).This helps to calculate the turbulence transport as well as an empirical length scale. The kinetic energy per unit mass (k) is given by

$$k=\frac{1}{2}\left(\overline{u'^2}+\overline{v'^2}+\overline{w'^2}\right) \tag{3}$$

Where, u, v, and w are wind speed components along, lateral and vertical directions.

Epsilon (ε) is the dissipation of kinetic energy as heat by the action of viscosity. Both k and ε can be used to defined length and velocity scales as

$$\text{Velocity scale } \vartheta = k^{\frac{1}{2}} \tag{4}$$

$$\text{Length scale } l = \frac{k^{\frac{2}{3}}}{\varepsilon} \tag{5}$$

Which leads to the eddy velocity (μ_t) being defined as follows:

$$\mu_t = C_\mu . l . \vartheta = C_\mu \rho \frac{k^2}{\varepsilon} \tag{6}$$

Where k and ε are subject of the transport equations

$$\rho\frac{\partial k}{\partial t}+\rho\overline{\mu_i}\frac{\partial k}{\partial x_j}=$$

$$\tau_{ij}\frac{\partial \overline{\mu_i}}{\partial x_j}-\rho\varepsilon+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_1}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \tag{7}$$

$$\rho\frac{\partial \varepsilon}{\partial t}+\rho\overline{\mu_i}\frac{\partial \varepsilon}{\partial x_j}=$$

$$C_{\varepsilon 1}\frac{\varepsilon}{k}\tau_{ij}\frac{\partial \overline{\mu_i}}{\partial x_j}-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k}+\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_1}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] \tag{8}$$

$$C_\mu = 0.09, \quad \sigma_k = 1, \quad \sigma_\varepsilon = 1.3, \quad C_{\varepsilon 1} = 1.44, \quad C_{\varepsilon 2} = 1.9$$

3. Numerical simulation results for various wind direction

The present study is carried out to understand the behaviours of wind distributions and wind pressure coefficients on the existing buildings cluster with varying wind directions for pedestrian level. Pedestrian-level of wind conditions is one of the first microclimatic issues to be considered in modern city planning and building design. When wind strikes a building surface and gets deflected towards the ground, it generates high wind speeds and as well as pressure on the windward side as well as near the corners of the buildings at pedestrian level. Simulation results of stream lines pattern of the velocity and pressure coefficients are discussed below.

3.1. Stream Line pattern of the Velocity

In these simulation results, Figure 4 and Figure 5 are demonstrated stream line patterns of the velocity on the vertical plane for south direction and horizontal plane around buildings with the effect of various wind directions, respectively.

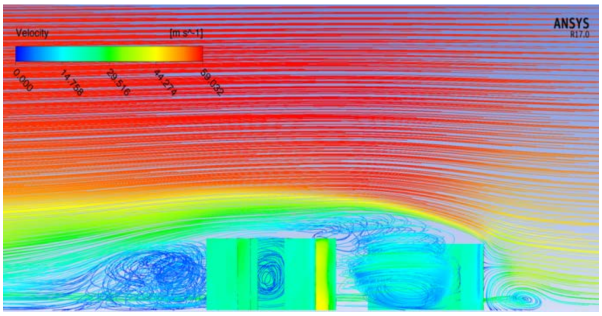
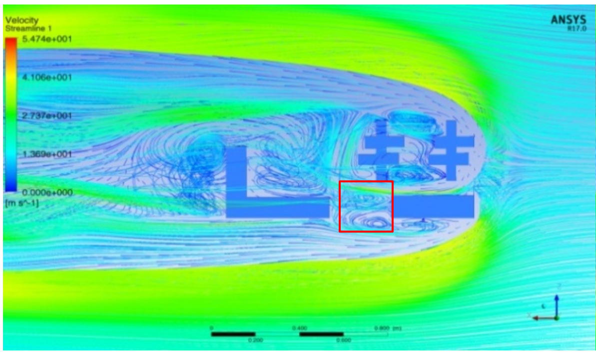
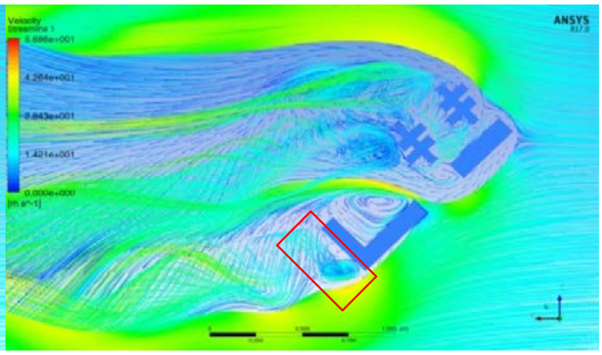


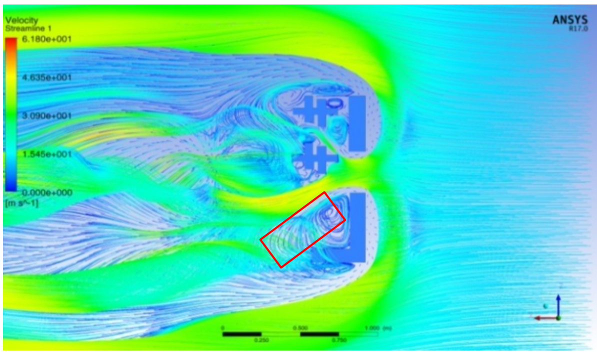
Figure 4: Close up View of Stream Line Pattern of the Velocity on the Vertical Plane at the Centre of the L Shape Building for South Direction



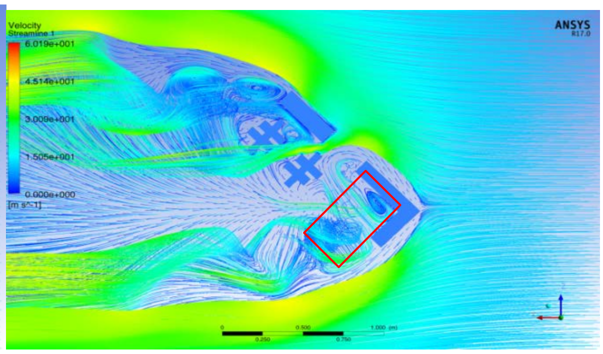
(a) North Direction



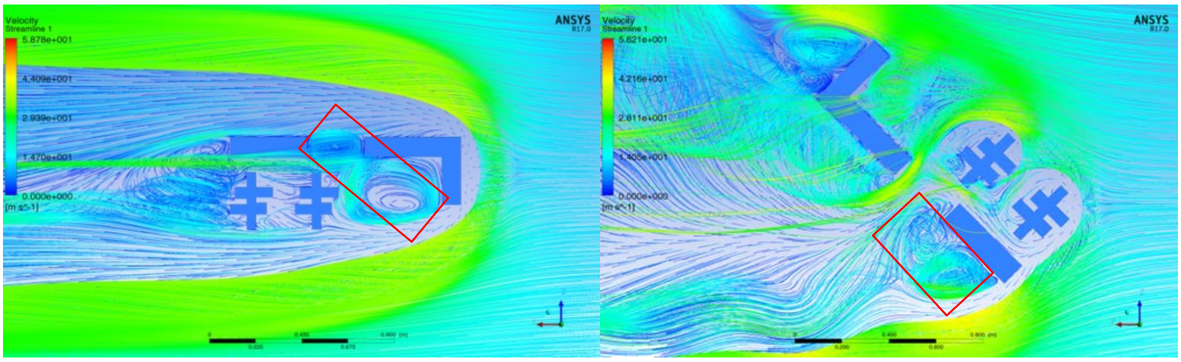
(b)North-East Direction



(c) East Direction

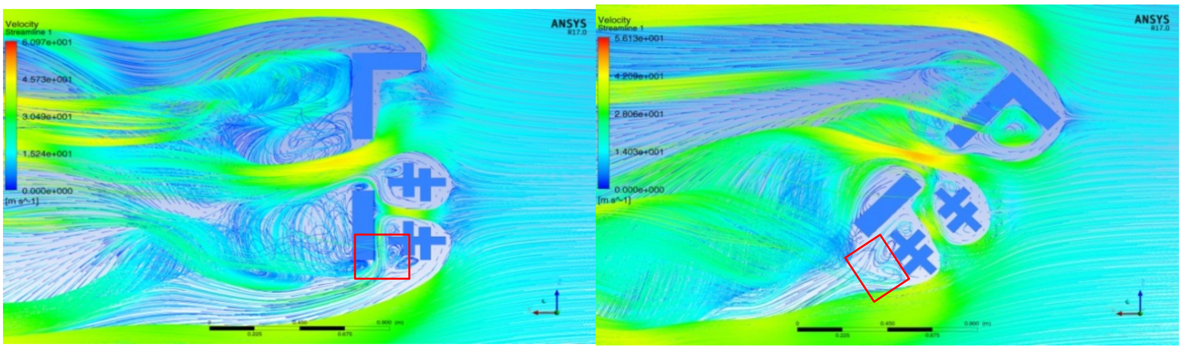


(d) South-East Direction



(e) South Directio

(f) South-West Direction



(g) West Direction

(h) North-West Direction

Figure 4: Stream line patterns of the velocity on the horizontal plane around the structures for eight wind directions

In these simulation results, wind flow patterns around buildings in build-up areas can be quite complex. According to visualization of stream line pattern of the velocity on the vertical plane at the centre of the L shape building, there is a small wind speed up area in upstream from the building due to the down wash of the building. The flow comes to rest to form the front stagnation point at the centre of the windward face of the building. From this point, the flow deviates into four main streams. Above this point, the first stream goes up and over the top of the building. Below this point, the second flow goes down into the recirculation at the base of the windward face of the building due to downwash effect. The third and fourth streams flow rapidly from there around the windward corners of the building.

It is clearly seen that wind flow deviation at the windward corners of the building and wake structure at leeward side of the building as the flow streams reattach. These flows can be dangerous and cause pedestrian discomfort due to wind velocity. Horseshoe vortices are occurred, one is counter clockwise and other is the clockwise as shown in Figure 3 with the red box for eight wind directions. When wind strikes from North direction, high wind speed with are occurred in the windward corner of the Condo B and Tower building. Horseshoe vortices are occurred behind the Tower building and wake structure are found the passage between Condo A and Condo B and near the windward and leeward of the L shape building.

In North East direction, wind first blows the corner of the tower building, and then the reattachment flow streams with high wind velocity are occurred near the Face C of the L shape building and wake structure is also occurred near the leeward side of the Tower building.

In East direction, wind first strikes the tower and L shape building and then the flow deviates into the windward corner of the buildings with high velocity. These high wind speeds between two buildings continue to flow into the Condo A building. So, Condo A building is suffered by high wind load and should be designed carefully to ensure safety structure. In South East direction, wind first flows to the corner of the L shape building and separate at each other sides. High wind velocities are fallen to the corner of Tower building.

In South direction, large wake structures are occurred at leeward of the L shape building and at the front of the tower building. In South West direction, wind start blows to the corner of the Condo B building. Large recirculation flows with high velocity become near the L shape building and horseshoe vortices are found leeward of the Tower building. Small recirculation flows are occurred leeward corner of the Condo A and B building.

In West direction, wind strikes directly the Condo A and B. The streams flow rapidly from there around the windward corners of the building. And then these flows are directly fallen into the Tower building with high wind velocity because it is situated behind Condo A and B buildings. High wind velocity are occurred the windward corner of the L shape building.

In North-West direction, wind first impacts the corner of Face A and F of the L shape building. High wind velocities are found the passage between tower building and L shape building. Wake vortices are occurred in the central part and leeward of the L shape building.

From the above flow visualization of the stream lines pattern, high wind velocity and high speed of recirculation flow are occurred the passage between the L shape building and Tower building at east side of the Condo A building. It can be found that buildings are suffered more high wind velocity than in its front buildings due to wake effect. Building form and detailing can greatly affect wind-flow patterns and speeds. With an appreciation of how winds flow around buildings, designers can avoid creating high wind speeds at ground level. This is an especially important consideration for buildings proposed for exposed sites, and near significant sites.

3.2. Comparison of Pressure Coefficients in L Shape Building

According to simulation results, comparison of maximum wind pressure coefficients on the six faces of the L shape building for eight wind directions are listed in Table 3.

In Face A, maximum positive pressure coefficient is occurred in S direction with 0.8 and maximum negative pressure coefficient is found in W direction with - 0.9. In Face B, maximum pressure coefficients are 0.7 by SE direction and - 0.7 by NW direction. Maximum pressure coefficients are 0.5 and - 0.9 in NE and N directions at Face C. Maximum pressure coefficients of Face D are the same magnitude but not the wind directions. In Face E and F, maximum positive pressure coefficient is occurred in W direction with 0.6 and 0.8 and maximum negative pressure coefficient is found in NE direction with - 0.8 and - 0.5, respectively.

The value of maximum positive pressure coefficient of the L shape building in windward face is equal to the value of 0.8 stipulated in the Myanmar National Building Coad and maximum negative pressure coefficient of this building is occurred in leeward face with - 0.9 which are higher than the value of the MNBC Code (i.e. -0.5). In general, wind flow into a building creates a positive pressure zone on the upstream side and negative pressure zones on all other sides.

Table 3: Surface pressure coefficient for the L shape building

Wind Directions	**Pressure Coefficient, C_p (face maximum value) of L shape building**					
	Face A	**Face B**	**Face C**	**Face D**	**Face E**	**Face F**
N	- 0.1	- 0.1	0.1	- 0.1	- 0.1	- 0.1
NE	- 0.5	0.6	0.5	- 0.7	- 0.8	- 0.5
E	- 0.6	0.7	- 0.9	- 0.5	- 0.5	- 0.5
SE	0.7	0.7	- 0.5	- 0.5	- 0.3	- 0.3
S	0.8	- 0.6	- 0.2	- 0.3	- 0.2	- 0.4
SW	- 0.4	- 0.6	- 0.8	0.5	0.5	0.5
W	- 0.9	- 0.4	- 0.8	0.7	0.6	0.8
NW	0.7	- 0.7	- 0.7	0.6	0.4	0.7

4. Conclusions

In this paper, numerical simulations based on Computational Fluid Dynamics (CFD) are performed to investigate wind flow behaviours of the building clusters in Mandalay. The purpose of this research is safe engineering design solutions related to environmental concerns such as the effect of the surrounding buildings with various wind directions. Information of wind pressure and air flow pattern can be provided to the designers at the early stage of design. When wind first impacts the building surface, it gets deflected towards the ground and generates high wind speeds and pressure on the windward side as well as near the corners of the buildings at pedestrian level. High speed of recirculation flow especially occurred around the Condo A due to backwash effect. One building placed to leeward of another can act as wake effect, protecting this building. When wind strikes building, causing a positive pressure zone to be formed on the windward face and a negative pressure zone is created at the sides and leeward face of the building. According to simulation results, the maximum positive pressure coefficient is 0.8 and maximum negative pressure coefficient is - 0.9. In this study, simulation results would be helpful for the planners to take many critical decisions such as influence of high rise buildings and building clusters on surrounding settlements to make their project better and effective from environmental view point.

5. Recommendations

The recommendations for further investigations to numerical analysis of applicability of CFD simulation in ANSYS FLUENT software are wind characteristics and wind flow patterns due to the effect of different geometric plan configurations and due to the effect of various spacing between two buildings should be investigated. Before the initial stages of the building design, the layout of the buildings over the neighbourhood is an important factor to control the wind field for urban planner.

Acknowledgment

The authors would like to Dr. Nilar Aye, Professor, Head of Department of Civil Engineering, Mandalay Technological University for her enthusiastic instruction, invaluable help, and indispensable guidance in the preparation of this paper.

References

[1] Blocken, Bert, and Jan Carmeliet. “Pedestrian wind environment around buildings: Literature review and practical examples.” Journal of Thermal Envelope and Building Science 28.2 (2004): pp.107-159.

[2] S., Bungale, Taranath, Wind and Earthquake Resistant Buildings, 2005.

[3] Daeung Kim, (2013, December 11), “The Application of CFD to Building Analysis and Design: A Combined Approach of an Immersive Case Study and Wind Tunnel Testing’’ Architecture and Design Research, Blacksburg, Virginia.

[4] F., Baetke, H., Warner, (1990), “Numerical Simulation of Turbulent Flow over Surface Mounted Obstacle with Sharp Edges and Corners”, Journal of Wind Engineering and Industrial Aerodynamics, Vol.35, pp.129-147, 1990.

[5] J., Franke, “Recommendations of the cost action c14 on the use of CFD in predicting pedestrian wind environment”, In: the 4th International Symposium on Computational Wind Engineering, Japan Association for Wind Engineering,Yokohama, pp.529-532, 2006.

[6] H, K., Veersteeg, W., Malalasekara, W., An Introduction to Computational Fluid Dynamic, Prentice Hall,1995.

[7] Franke, J: Introduction to the prediction of wind loads on buildings by computational wind engineering (CWE). In T. Stathopoulos and C. C. Baniotopoulos, editors, Wind Effects on Buildings and Design of Wind-sensitive Structures, chapter 3, pages 67-103. Springer Wien, New York, 2007.

[8] MNBC: Myanmar National Building Code, 2016.

[9] K., Hanjakic, B.E., Launder, A Reynolds Stress Model or Turbulence and its Application to thin Shear Flow, Journal of Fluid Mechanics, 1972.