NUMERICAL STUDY ON WIND PRESSURE CHARACTERISTICS OF CLUSTER OF BUILDINGS IN ANSYS FLUENT – A CASE STUDY

[1]THET MON SOE, [2]SUN YU KHAING

[1,2]Civil Engineering Department, Mandalay Technological University
E-mail: [1]thetmonsoe.658@gmail.com, [2]sykpku@gmail.com

Abstract- Wind is air movement relative to the earth, driven by several different forces, especially pressure differences in the atmosphere. Therefore, the effects of different wind flow distribution to the cluster structures in an aspect of structural safety should be investigated. In modern cities, there are increasing numbers of high constructions and complex forms which can involve problems of significant wind discomfort around these buildings. In recent years, the numerical investigation has emerged as powerful and sufficient tool for building optimization and a better township planning. Present work proposes the methodology to carry out exterior numerical on a cluster of existing structures. Present work used a three dimensional scale down model of buildings where steady flow analysis has been done. It has been implemented through ANSYS Fluent 17.0 using SIMPLE algorithm as solver. The turbulence models used as the RANS based model: the standard k-epsilon model, RNG k-epsilon model and Realizable k- ε model. In this paper, the simulation results have predicted the ring vortex, roof separation bubble, side separation and wake recirculation, but the shapes and locations are different from each other. Wind pressure coefficients compared the three different turbulence models in Fluent. The maximum wind pressure coefficients was found to be at the middle of windward surface with height of about 0.7H-0.8H and the minimum wind pressure coefficient generates at the corner of the flow separation.

Keywords- Computational Fluid Dynamics (CFD), RANS Equation, Cluster of Existing Structures, Wind Pressure Coefficients.

I. INTRODUCTION

With the development in technology taller and taller 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. This causes many environmental problems in nearby areas such as accelerated wind flow at the ground level impacting the comfort, and sometimes safety, of the users of the building and the pedestrians in the surrounding street canyons. For a safe structural design of any particular structure, it is necessary to consider wind effects. Although one cannot see the wind, it is a common observation that its flow is quite complex and turbulent in nature and has a tendency to exert differential velocity and pressure field around any obstacle likely to obstruct its flow path. 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, terrain roughness, turbulence in the wind, location of a particular structure etc [1].

Modeling the wind atmosphere associated with proposed or existing buildings is of great importance for the wind engineering, Civil Construction sectors as well as Structural Engineering Sectors. Computational Wind Engineering (CWE) as a branch of Computational Fluid Dynamics (CFD) has been developed rapidly over the last three decades 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, predicting pollutants depression patterns in urban areas, evaluate pedestrian level wind comforts, etc. Numerical simulations are more flexible and robust in terms of simulating wind flow conditions together with detailed surrounding features such as buildings, mountains, tree, etc. CFD techniques have been adopted for the estimation of wind flow around building. Progress in high-speed processing by personal computer and rapid propagation of software for numerical analysis of fluid dynamics in recent years have enabled prediction of the pedestrian wind environment around high-rise buildings based on CFD.

In Myanmar, wind engineering is not as much developed as compared with the other branches of civil engineering. Nowadays, the concern of wind engineering is increasing among Myanmar engineers due to increase of damages due to frequent occurrence of high wind events and construction of many high-rise buildings in city centres, which are more susceptible to wind load. Wind around high rise buildings in cities may create serious problems even at the pedestrian level. The review of literature indicates insufficient work in the area of external wind simulation around building structures which are useful for architects, city planner and building designers [2]. The objective of the present work is to illustrate the capability and potential of modern CFD tool through a detailed case study of a cluster of high rise building located in Mandalay city. Investigation would be helpful for the planners to take many critical decisions such as influence of high rise buildings and building clustering on surrounding

settlements to make their project better and effective from environmental view point. For the city planners and building designers, prior knowledge of the air flow around building play a significant role in finalizing their design.

II. DESCRIPTION OF A CASE STUDY

2.1. Existing Cluster of building

Almost in all major cities in the developing countries, as well as in the developed ones, residential buildings are characterized with high-rise and high density. In Mandalay city, the numbers of high-rise residential buildings are increasing in number, making the city center denser, also leading to urban spread as well. Wind around high rise buildings in cities may create serious problems even at the pedestrian level. In this study, Present work is to carry out numerical investigation of air flow pattern around a cluster of existing high rise structures located in Mandalay city as shown in Fig.1. It consists of four structures of different height and shape.



Fig. 1. Existing Cluster of Buildings in Mandalay City

III. METHODOLOGY

The basic wind speed of Mandalay region is about 38.16 meters per second from Myanmar National Building Code [3]. The movement of air around the building can be modeled as Newtonian fluid with turbulent, viscous, incompressible and steady flow in nature. The flow behaviour is governed by mass and momentum conservation with appropriate turbulence model where the body force can been neglected.

3.1. 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 part of the atmosphere is called as atmospheric boundary layer [4]. 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 (Eq1) and momentum (Eq2).

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

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

Where, u_i, u_j are velocity components, ρis air density, P is air pressure, μ is dynamic viscosity, and t is time.

3.2. k-ε Turbulence Model

The K-epsilon model is one of the most common turbulence models [5].The widely used of turbulence model is standard k-ε model developed by Hanjalic and Launder [6]. It has been verified and validated for a wide variety of flows and relatively low computational costs. It is a two equation model, which means, it includes two extra transport equations to represent the turbulent properties of the flow. This two equation model characteristics small scale turbulence by using two numbers kinetic energy (k), and energy dissipation rate (ε).The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the turbulent dissipation, ε. The kinetic energy per unit mass (k) is given by

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

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

$$\text{Velocity scale } \vartheta = k^{\frac{1}{2}} \qquad \text{Eq(4)}$$

$$\text{Length scale } l = \frac{k^{\frac{2}{3}}}{\varepsilon} \qquad \text{Eq(5)}$$

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

$$\mu_t = C_\mu . l . \vartheta = C_\mu \rho \frac{k^2}{\varepsilon} \qquad \text{Eq(6)}$$

Where k and ε are subject of the transport equations

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

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

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

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

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

The RNG k-ε model is an alternative to the standard k-ε model and was proposed by Yakhot and Orzag (1986). It is based on renormalization group analysis of the Navier- Stokes equations. The analytical derivation results in a model with constants different from those in the standard k-ε model, and additional terms and functions in the transport equations for k and ε. Realizable k- ε model consisted of a new model dissipation rate equation and a new realizable eddy viscosity formulation. The reliability effect was achieved by the constant C_μ of the turbulent eddy viscosity, which was no longer a constant but a function of the turbulent fields, mean strain and rotation rates.

3.3. Pressure Coefficient

The pressure coefficient, Cp, is defined as

$$Cp = \frac{p - p_o}{\frac{1}{2}\rho U_{ref}^2} \qquad \text{Eq(9)}$$

where p and p_0 are static pressure and reference pressure, respectively. ρ is air density and U_{ref} is wind speed at reference height.

3.4.Turbulent Model for Study

Turbulence modeling is the computational procedure to solve and analyse the fluid flow introducing some approximations in the governing differential equations so that required solution is obtained approximately consuming feasible computational memory and time. Turbulence modeling is based on the assumption that the real flow field may be substituted by an imaginary field of mathematically defined continuous functions. ANSYS FLUENT has one-equation models and two-equation models which are RANS based models and other two are Detached Eddy Simulation and Large Eddy Simulation. One-equation model is Spalart-Allmaras model. Two-equation model includes standard k-ε model, RNG k-ε model, Realizable k-ε model, standard k-ω model, SST k-ω model and Reynolds stress model. In this study, models have been selected for numerical simulation for wind load prediction using three different turbulence models which are standard k-ε model, RNG k-ε model and Realizable k- ε model.

3.5. Computational Domain

The size of computational domain in vertical, lateral and flow direction depends on the region that shall be represented by the simulation [7]. Domain should be large enough to avoid reflecting of fluid streams, which may cause abnormal pressure field around the model. The blocking ratio should not exceed 3% (Baetke & al 1990) and Werner is also a consideration for select size of the computational domain [8]. 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 (Franke & al 2004) [7]. The minimum length of the simulation field is 5 * min (L, 2H) upstream of the building and 8 * min (L, 2H) downstream of it, taking L equal to the length of the building and H equal to its height (Blocken & al 2003). The minimum dimensions of the simulation field are: 5H upstream of the building, 5H on each building side, 5H above it and 15 H downstream of the building, the height H is the height of the target building [9]. It should be noted that there is a possibility of unrealistic results, if the computational region is expanded without proper representation of surrounding of the building. However, it is also true that unnecessary large fluid domain may demand higher computational cost.

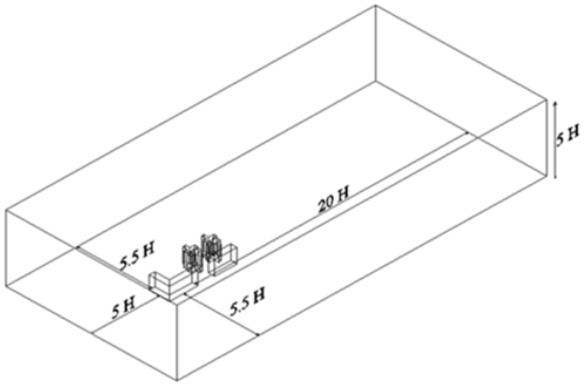


Fig. 2. Arbitrary Domain Size

In this study, the arbitrary domain size is shown in Fig.2 to simulate the building within the atmospheric boundary layer. The length of the simulation volume extends upstream of buildings over a distance of 5H, downstream over a distance of 20H, summaries side of the buildings over a distance of 5.5 H and height of the domain size over a distance of 5H, H being the height of the target building. The full scale dimensions of the computational domain are 571.15m × 228.5m×1236.3 in lateral, vertical and flow directions respectively. In this study, the simulated grid was made up over 2670000 hexahedral cells with MultiZone Meshing Method. The maximum cell size for all three domains was 0.02m and minimum cell size was 0.0001m.

3.6. 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. The velocity profile at inflow is imposed using UDF (user defined function) in FLUENT. The power law was used to simulate the inflow u- velocity profile of the

atmospheric boundary layer. Sides of domain are symmetry. At top and bottom walls of domain are no slip walls. The summary of the data required for computation through ANSYS FLUENT is shown in Table .1.

Table .1: Input Parameter

S.N	Parameters	Values
1	Velocity Profile	$38.16 \times (\frac{Z}{10})^{1/7}$
2	Basic Wind Speed (m/s)	38.16
3	Power Law Coefficient, α	1/7
4	Turbulent Viscosity Ratio	10
5	Density of air (kg/m^3)	1.225
6	Viscosity of air (kg/m-s)	$1.7594*10^{-5}$
7	Solver	Pressure –based steady state
8	Models	standard k-ε model, RNG k-ε model and Realizable k-ε model with standard wall functions
9	Solution Method	SIMPLE pressure velocity coupling

3.7. Numerical Setup

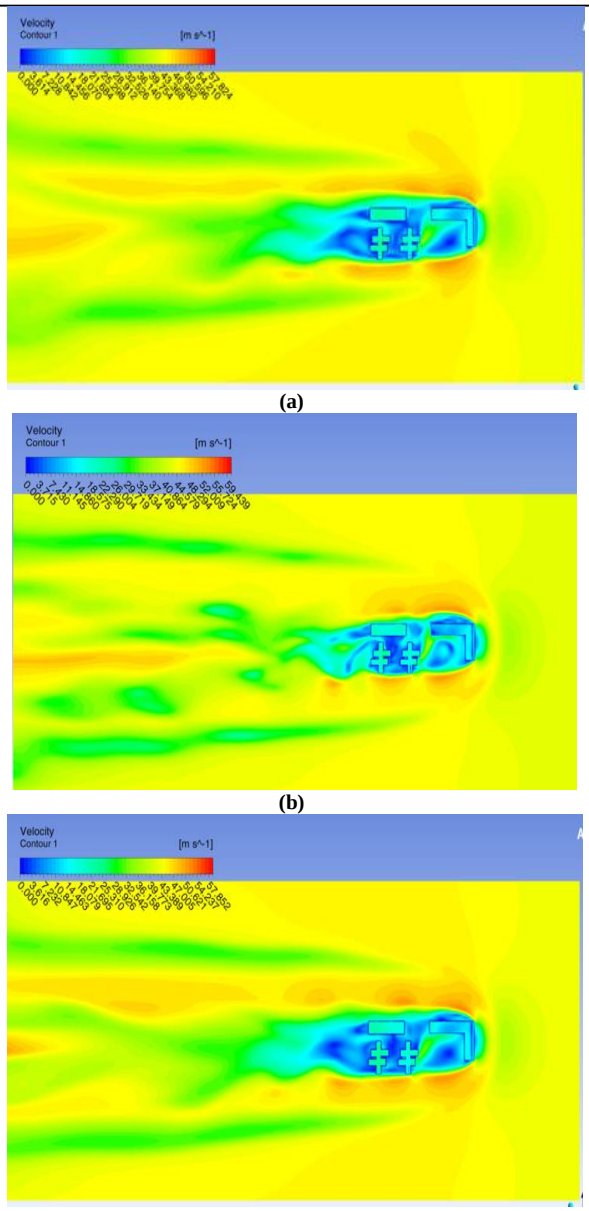
CFD code Fluent v17 is used to solve the 3D Reynolds-Average Navier-Stokes (RANS) equations and continuity equation, using the finite volume method. Standard k-ε model, RNG k-ε model and Realizable k- ε model with standard wall functions are used for turbulence modelling. SIMPLE algorithm is used for 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.

IV. SIMULATION RESULTS FOR THREE DIFFERENT K-Ɛ MODELS

4.1 Results Of Contour Map Of The Velocity

In this study, the contours line of the velocity flow on the horizontal plane at the middle height of the L-shape building (a) standard k-ε model, (b) RNG k-ε model and (c) Realizable k-ε model are shown in Fig. 3.

It is clearly seen that the simulation results performed the ring vortex, roof separation bubble, side separation and wake recirculation, but the shapes and locations are different from each other. Contour line deviation at the front corners of the building and wake structure are found at leeward side of the building. High wind speed area is extending from front corners of the building to sideways and is found over the building height.



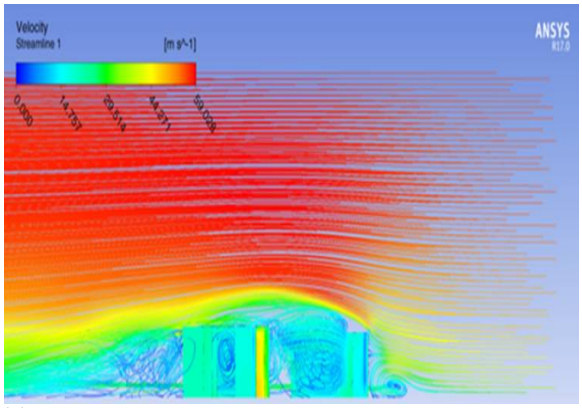
(a)

(b)

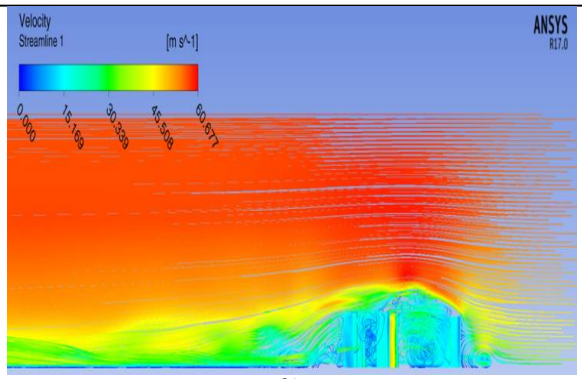
(c)

Fig. 3. Contours Line of the Velocity Flow on the Horizontal Plane at the Middle Height of the L-shape Building (a) Standard k-ε Model, (b) RNG k-ε Model and (c) Realizable k-ε Model

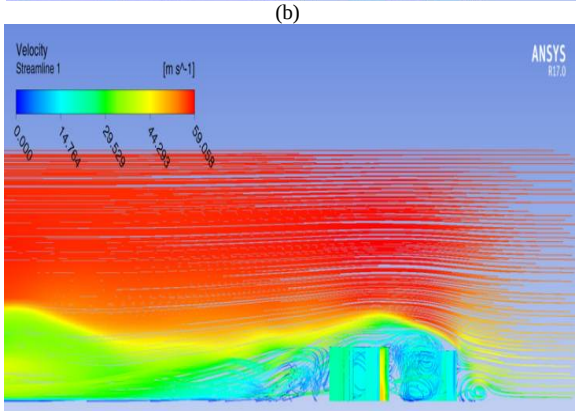
4.2. Results Of Streamline Patten Of The Velocity



(a)



(b)



(c)

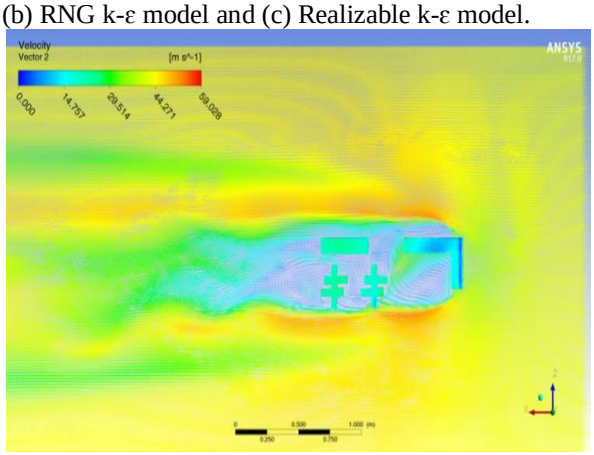
Fig. 4. Stream Lines of the Velocity Flow on the Vertical Plane at the Center of the L-shape Building (a) Standard k-ε Model, (b) RNG k-ε Model and (c) Realizable k-ε Model

Stream lines of the velocity flow on the vertical plane at the center of the L-shape building (a) standard k-ε model, (b) RNG k-ε model and (c) Realizable k-ε model are shown in Fig. 4.

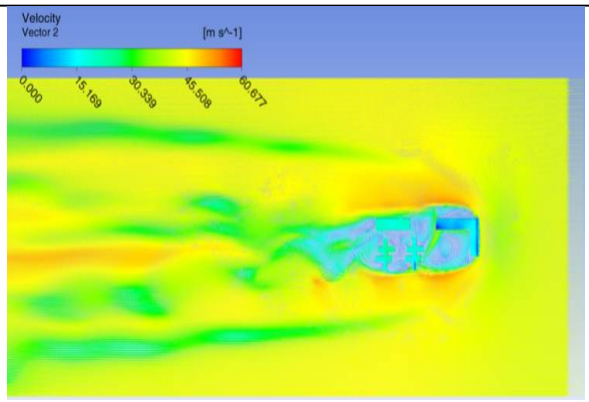
On the windward edge of the building, the flow is separated and a separation bubble appear. On the windward face of the building, at about two-thirds of the height, the flow comes to rest at the stagnation point, where the pressure is the maximum. Below that point, the flow goes down and recirculation vortices to develop near the ground. Above that point, the flow goes up to the top of the building.

4.3. Results Of Vector Flow Of The Velocity

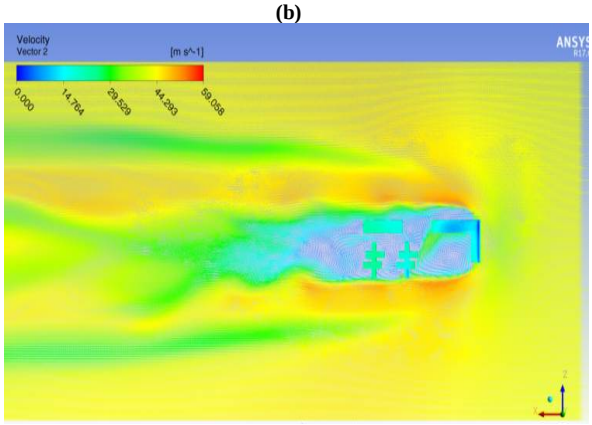
To examine the airflow process, Fig. 5 shows stream line pattern on the horizontal plane at the middle height of the L-shape building (a) standard k-ε model, (b) RNG k-ε model and (c) Realizable k-ε model.



(a)



(b)



(c)

Fig.5. Vector Flow of the Velocity Flow on the Horizontal Plane at the Middle Height of the L-shape Building (a) Standard k-ε Model, (b) RNG k-ε Model and (c) Realizable k-ε Model

In this study, the analysis results shown that in front of the building there is a small wind speed up area in upstream from the building. Vortices occur in both windward and leeward sides of the L-shape building. In the center of these vortices, the suction is large, possibly leading to structural severe damage. Resulting from the adjacent airflow around the buildings, a wake region was evidently observed with the recirculating flow immediately behind the solid structures in the leeward side. A wake region and stagnant zone develops behind the building. This is not good for pleasant ambience. The RNG k-ε model and Realizable k-ε model was found larger fluctuating of the flow field than standard k-ε model. The Standard k-ε model greatly over predicts the size of the wake recirculation while Realizable k-ε model appears shorter wake recirculation zone. The RNG k-ε model appears to produce exactly prediction of the recirculation zone and a strong wake flow among buildings. The town planners need to be careful in minimizing such wind effects.

V. SIMULATION RESULTS FOR PRESSURE COEFFICIENTS

The comparison of pressure coefficients for the windward and leeward wall of the L-shape building with different turbulent model are shown in Fig. 6 and Fig. 7.

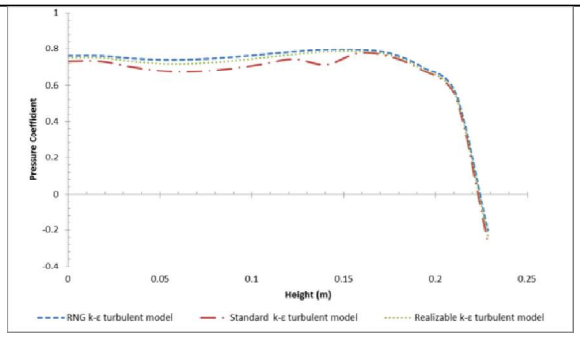


Fig. 6. Comparison of pressure coefficients for the windward wall of the L-shape building with different turbulent model

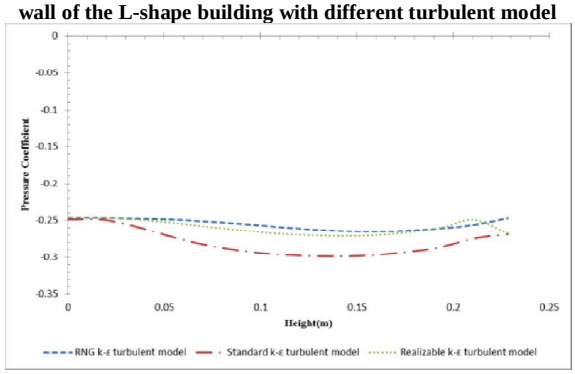


Fig.7 Comparison of pressure coefficients for the leeward wall of the L-shape building with different turbulent model

It can be easily found that in three turbulence models, RNG k-ε model is the highest pressure coefficient other than turbulence models. The maximum wind pressure coefficients of the windward wall of the L-shape building with different turbulent model are 0.79, 0.77, 0.78, respectively, which are close to the value of 0.8 stipulated in the Myanmar National Building Coad. The minimum wind pressure coefficients of the leeward wall of the L-shape building with different turbulent model are -0.27, -0.3,-0.27, respectively, which are higher than the value of the MNB Code (i.e. -0.5).

VI. DISCUSSIONS AND CONCLUSION

In this paper, the three-dimensional numerical model was developed to analyze the flow pattern around buildings. Investigation would be helpful for the planners to take many critical decisions such as influence of high rise buildings and building clustering on surrounding settlements to make their project better and effective from environmental view point. Computational Fluid Dynamic (CFD) simulations are getting popular with the improvement of performance of computer resources. In this paper, the Standard k-ε model and Realizable k-ε model predict similarly the magnitude of the maximum velocity while the RNG k-ε model is the most value of the maximum velocity other than two models. The maximum wind pressure coefficients are found to be at the middle of windward surface with height of about 0.7H-0.8H and the minimum wind pressure coefficient generates at the corner of the flow separation. The numerical obtained in this study are expected to provide practicing engineers a better understanding of wind field around building.

ACKNOWLEDGMENT

The author wishes to express her deepest gratitude to her Excellency, Union Minister Dr. Myo Thein Gyi, Ministry of Education for the celebration of this international conference on science and engineering at Yangon, Myanmar. The author is deeply gratitude to Dr. Myo Nyunt, Rector, Mandalay Technological University, for his kindness, help, permission, guidance and advice of this paper. The author would like to thank to Dr. Nilar Aye, Professor, Head of Department of Civil Engineering, Mandalay Technological University, for her valuable suggestion and giving useful comments. We would like to thank our parents, our teachers and all of friends for their support and encouragement through the whole period of studying for this research

REFERENCES

[1] S., Bungale, Taranath, Wind and Earthquake Resistant Buildings, 2005.
[2] H., Robert Scanlan, Wind Effects on Structures, 1985.
[3] Myanmar National Building Code, 2016.
[4] H, K., Veersteeg, W., Malalasekara, W., An Introduction to Computational Fluid Dynamic, Prentice Hall,1995
[5] J. R., Garatt, G. D., Hess, Physick, W>L, P., Bougeault, The Atmospheric Boundary Layer- Advances in Knowledge and Applications, Boundary layer Meteorology, 1996.
[6] K., Hanjakic, B.E., Launder, A Reynolds Stress Model or Turbulence and its Application to thin Shear Flow, Journal of Fluid Mechanics, 1972.
[7] 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, 2006.
[8] 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, 1990.
[9] B., Blocken, J., Carmeliet, T., Stathopoulos, CFD Evaluation of the Wind Speed Conditions in passages between Buildings- Effect of Wall-function Roughness Modifications on the Atmosphere Boundary Layer Flow. Journal of Wind Engineering and Industrial Aerodynamics, 20007.

★★★