SECM/15/168

ICSECM 2015 - Wind Design of Slender Tall Buildings: CFD Approach

D. Mohotti[1*], K. Danushka[1] and P. Mendis[2]

[1]School of Civil Engineering, The University of Sydney, Sydney, Australia
[2]Department of Infrastructure Engineering, University of Melbourne, Melbourne, Australia
*E-Mail: damith.mohotti @ sydney.edu.au,

Abstract: Urbanization has led to the uprising of such buildings in densely populated areas where land availability and prices are a concern in such areas. Where such concerns exist the land must be fully exploited and thus constructions of tall buildings are always found as a solution in such areas. Wind behaviour is a key designing parameter for such building and need to be assessed accurately in the preliminary and secondary design stages. As most of the existing design codes have their own limitations in providing necessary guidelines for the wind designing, such as height limits of the buildings, the existing practice is to conduct wind tunnel tests to determine the wind induced loads on the buildings. However, the cost of the wind tunnel test is comparatively high and conducting wind tunnel tests at preliminary design stage is uneconomical. The rapid growth of Computation Fluid Dynamic (CFD) technique over the last few decades enables Engineers to simulate the wind behaviour around moving objects such as aeroplanes and automobiles. Therefore use of such methodology to predict wind loads on the buildings, especially at the preliminary design stages could be beneficial. This paper discusses a preliminary investigation that carried out on a non-typical 350m tall slender building using CFD approach.

Keywords: Wind loads, Tall buildings, Computational fluid dynamics.

1. Introduction

In the past recent, the construction of tall buildings has become a common featurette around the world. The word skyscraper has been often used for high rise buildings that are generally 150m or more tall. However the terms such as “Tall”, “Super-tall” and “Mega-tall” have been classified by the Council on Tall Buildings and Urban Habitat (CTBUH) for buildings which are 200m, 300m and 600m respectively [1]. These terms have originated because of a massive upsurge of buildings with heights within this range. In the past fifteen years many advances have been made in regions such as Asia and the Middle East (mainly the U.A.E) where super and mega tall structures have been erected. [2].

Urbanization has led to the uprising of such buildings in densely populated areas where land availability and prices are a concern in such areas. Where such concerns exist the land must be fully exploited and thus constructions of these tall buildings are always found in such areas. Furthermore countries such as U.A.E and China which currently claim to have the first and second tallest buildings, , use such structures as tourist attractions, thus bolstering their reputation and economy.

The major concerns in designing of tall buildings are not limited to the resistance of the structural systems to the lateral loads, but also extended to the comfort of the building’s occupants in relation to the wind-induced motion of the buildings.

The current practice is the use wind tunnel tests and relevant standards wind induced pressure and building motion parameters. Australian standard AS/NZS-1170.2 (Standards Australia, 2011), China Code (GB50009:2012), American Code (ASCE7-10:2010), EURO Code (EN1991-1-4:2005) and Japanese Code (AIJ 2004) can be considered as the frequently used standards for such analysis all over the world. AS1170.2, GB50009:2012 and EN1991-1-4:2005 have specified the upper height limits for the building where the relevant codes can be adopted. In addition, for free standing tall buildings the first-mode fundamental frequency shall be larger than 0.2 Hz. Therefore, tall slender buildings outside these limits, the AS-/NZS-1170.2 have recommended using wind tunnel tests as a supplementary technique in order to estimate the wind loads. However, performing wind tunnel tests require considerable resources and it is a time consuming and expensive effort. Therefore, it is worthwhile to look into alternative solutions to replace such experimental procedures.

Application of Computational Fluid Dynamics (CFD) in wind engineering can be considered as an alternative option to the well-known wind tunnel tests procedure. Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase (gas or liquid, but not both) fluid flow. With the advancement of computer capabilities, it is now possible to simulate considerable complex numerical simulations within a feasible time period. One of the main areas of application of CFD in wind engineering is in the environmental aspect. In the past few decades, research on the application of CFD has been conducted extensively in areas such as pedestrian wind comfort and safety, exterior building surface heat transfer, pollutant dispersion around buildings, and natural ventilation of buildings [3]. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. CFD has been used effectively in modelling aerodynamics effect on automotive. Therefore, it has shown a considerable accuracy in simulating atmospheric boundary layer effect. This highlights the possibility of using a similar approach to simulate the wind behaviour around the buildings [4, 5]

There is a considerable research gap exist in this area of interest as there is no significant work has been conducted to validate the results obtain through CFD simulations. Accurate representation of flow separation, turbulent formation are vital for achieving better representation of actual scenario of virtual wind models. In this study advanced Finite element code, ANSYS FLUENT 16 [6] has been used as the solver. Six different turbulence models are available in FLUENT which are incorporated with equations to solve the transported variable, turbulent viscosity, turbulent production and turbulent destruction terms. LES (large Eddy simulation) and K- ε models are the most commonly used turbulent models to represent the wind flow around building domains. There are considerable advantages and disadvantages of these models in terms of using in wind simulations. This paper only presents the results obtained using K- ε model even though the LES and SAS models [7], [8]; [9]; [10] were used in the current research study.

This paper presents the preliminary work done on an ongoing research project to simulate the wind pressure acting on tall slender building. Section 2 of the paper briefly discusses the key design parameters and general effects of wind on buildings. Section 3 describes the modelling process and parameters used in this study. Section 4 presents some results that were obtained and section 5 draws some final conclusions.

2. Wind Flow around Buildings

A tall building concentrates wind at its base (Figure 1 (a). This causes significantly turbulent region in front of the buildings. Similarly when a building is significantly taller than its surroundings it can experience high wind loads and concentrate pedestrian-level winds 1(c). It is also important to understand the flow due to the interaction of adjacent buildings. One of the better alternative to the wind tunnel tests to understand the flow characteristics if virtual wind tunnels using the CFD approach.

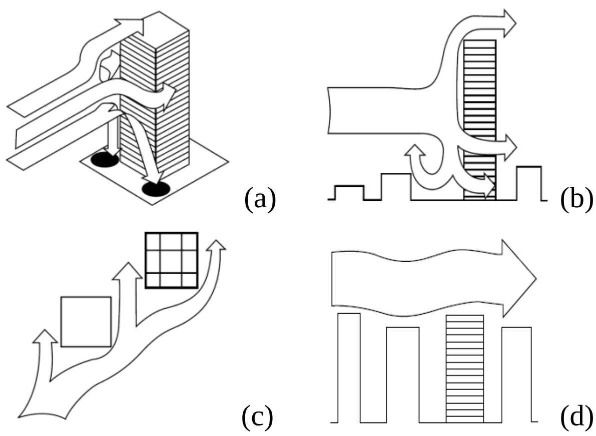

Figure 1: Flow past bluff body

Vortex Shedding

Vortex shedding is an oscillating flow that takes place when fluid such as air flowing past a bluff body such as buildings. The characteristics of the vortex shedding depend on the velocity of the flow, size and shape of the body. The frequency of this vibration is called shedding frequency (f_s) as defined as follows.

$$f_s = \frac{S_r U_\infty}{L} \qquad (1)$$

where L is a characteristic length (equal to the diameter D in case of a circular cylinder or tube in cross flow) and U_∞ the freestream velocity.

$$S_r = 0.198\left(1 - \frac{19.7}{R_e}\right) \qquad (2)$$

Vortex shedding is significantly important in buildings with cylindrical or similar cross sections. The building consider in this study mainly consists with circular cross sections where vortex shedding have become a critical design parameter.

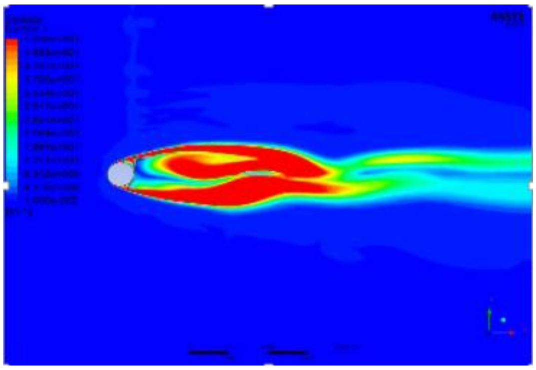

Figure 2: Vortex shedding around cylindrical object

3. Modelling

Figure 3: 350m tall building

The main purpose of this study is to conduct CFD tests for a 350m tall building which is irregular of shape with a base of 38m diameter. Commercial software ANSYS FLUENT v16 used in developing the numerical models.

The method by which a CFD application works can be broken down into three stages namely;

- Pre-Processor
- Solver
- Post-Processor

Domain Size: Choosing an appropriate domain size in important and needs to be large enough to ensure the effects of the boundary walls does not interfere with the final result. For this study a recommended guide is used for the domain size such that the inlet face, the lateral faces and the height of the domain is five times the height of the building (5H). The outlet must is to be located at least 15H away from the building to allow for fully developed flow, as shown in Fig. 4a&b [11].

Quality of the mesh: The generation of mesh for the model is important as the quality of the generated mesh has a direct impact on the credibility of the results. For the domain a hexa mesh is to be used whilst for the layer of the building a tetrahedral mesh is opted due to the incapability of generating hexa meshes on a complex geometry surface. To ensure that the mesh quality is within an acceptable level, the maximum skewness of any element is not to be exceeded a value of 0.85, which is deemed to be good. The number of elements within the mesh is also an important number that needs to be considered. The higher the number of elements the better the accuracy of the solution but longer time it takes for computation. Thus a balance needs to be struck such that a large number of small elements are not employed to gain accuracy which in turn increases time taken to compute but also an adequate number of elements are included such that it would not compromise the accuracy of the results. This was achieved by defining a minimum and maximum element size for the domain and the building surface. The building surface is forced to have a smaller maximum element size to that of the domain and thus this serves as the focal point where elements start to grow. Further to this other methods such as providing a refinement box around the building and in the wake region of the domain to capture features such as vortex formation could also be employed, but for this initial study this was not used.

The following information explains some of the meshing criteria that are imposed on to this model to ensure a high quality mesh generation;

- Building minimum mesh size 0.2m and maximum 0.8m with a growth rate of 20%
- Building mesh growth function defined as a soft function
- Surface mesh function soft with minimum mesh size of 0.2m and maximum mesh size of 30m at a growth rate of 20%
- Rest of the domain with a minimum mesh size of 0.2m and maximum mesh size of 40m at a growth rate of 20%

The domain and the ground surface contain hexa meshes (Fig. 5), however the surface boundary that represents the building, consists of tetrahedral meshing as its complex geometry makes it hard to force hexa meshes to be included on its surface without altering its shape considerably (Fig.6). An inflation layer is defined for the building surface as it is important to capture the nonlinear variation of wind flow when it approaches the building surface ("no slip" wall condition). As mentioned earlier the goal was to obtain a mesh with skewness of less than 0.85. The element with highest skewness was noted to be 0.78 which is very good for the purpose of this analysis.

Turbulent Models: At the solver stage the meshed model is imported and this is where the turbulence model is defined along with any other initial conditions such as wind speed. The manner in which fluids flow is important to understand. Most engineering related problems where the flow of a fluid is concerned the flow is turbulent as opposed to laminar. Laminar generally occur at low velocities such that the flow of fluid can be conceptualized as a streamline and thus each streamline flows parallel to each other in layers. However turbulent flows are completely random and the flow of liquid can be termed as being 'chaotic'. In wind engineering the flow of air is always turbulent therefore the resulting eddies from such a flow are unsteady and occurs in a three dimensional spatial manner. For computer simulations there are various models that can be chosen from and few important ones are listed in table 1. However it must be noted that these are computational turbulent models cannot predict all kinds of turbulent flows and thus each model have its strengths and weaknesses [12]

Table 1: Description of turbulence models

Turbulence Model	Description
Direct Numerical Solutions (DNS)	Numerically solving the Navier-Stokes equations and hence theoretically all turbulent flows can be simulated. Only applicable to simple flow problems where the Reynolds number is relatively low that is, laminar flow [12].
Reynolds-Averaged Navier-Stokes (RANS)	Solves Navier-Stokes equations by either time averaging or ensemble averaging. All turbulence is modelled but large eddies are not resolved. RANS incorporates many types' models where the most prominent modelling tool is the two equation $k - \varepsilon$ model.
Large Eddy Simulations (LES)	Navier-Stokes equation is solved spatially within the domain. Large eddies that are formed are directly solved whilst eddies smaller than the mesh are modelled

At the initial stage of experimentation a steady state RANS model will be used however at a later stage LES simulations and RANS transient analysis can be undertaken to induce effects such as crosswind excitation as it is an important factor in the design of tall buildings [13]. For the solver a steady state RANS k-ε realizable turbulence model was chosen as it requires less computational power and produce accurate results for the needs of this study. Furthermore modified RANS models have been shown to produce good results that are within reasonable deviation limits of those that were recorded in wind tunnel testing [14-16].

Boundary layer profile : For this particular study the use of the wind velocity profile has been derived from the Australian Commentary for the wind code AS 1170.2:2002 known as the Deaves and Harris Model [17]. The wind profile is input into the ANSYS via a Matlab script. The wind profile is as plotted and shown in Fig. 7 and the output from the model depicts this wind profile and as is shown in Fig. 8 at intervals of 10 seconds.

Convergence and Solution: The solution for the CFD simulation is brought about by a step process and hence number of iterations/steps needs to be provided. This is entirely up to the user and the rate of convergence of the solution. For this study we are looking at a convergence for all solutions in the order of 10^{-3} as this is an acceptable value for engineering applications. This whole process is iterative, if the solution does not converge then the user needs to fix the domain size, improve the mesh quality or increase the number of steps and rerun the program until such a convergence is obtained. The Post-processor stage is where the results are observed. In ANSYS visually attractive and meaningful results can be obtained by employing the CFD post processing results tool. Streamlines of air flow, pressures along the building, flow of air as contours on different planes are some of the useful visual outputs that can be obtained. CFD results must always be compared to those that are obtained by wind tunnel experimentation. For this particular building current wind tunnel data is unavailable however there might be a possibility that wind tunnel experimentation for a scaled model to be done in the near future where then results can be compared with. The number of iterations that was used for the solution to converge was kept at 200 whilst first order solver was used. It was required for the solution to converge to 10^{-3} and this was achieved in a mere 36 steps. Such a quick convergence to the solution was due to the size of the domain and the quality of the overall mesh in the domain which provided good continuity of flow throughout the domain.

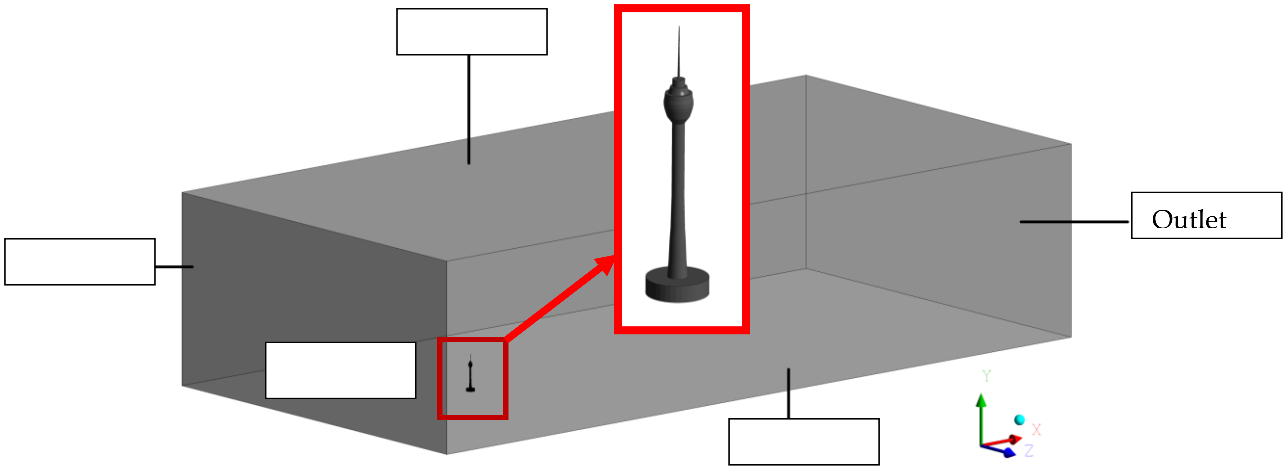

Figure 4a: Virtual wind tunnel

Figure 4b: Dimensions of domain (H: is the height of the building)

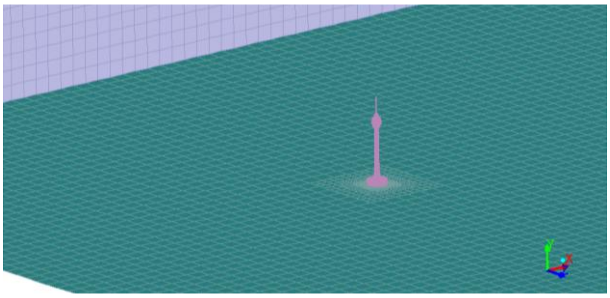

Figure 5: Hexa mesh on ground

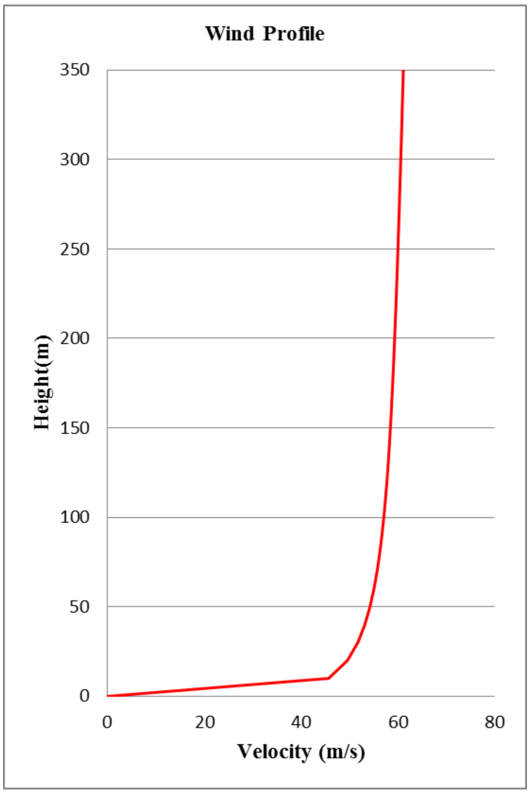

Figure 7: Wind profile along height of domain

Figure 6: Tetrahedral mesh on building surface

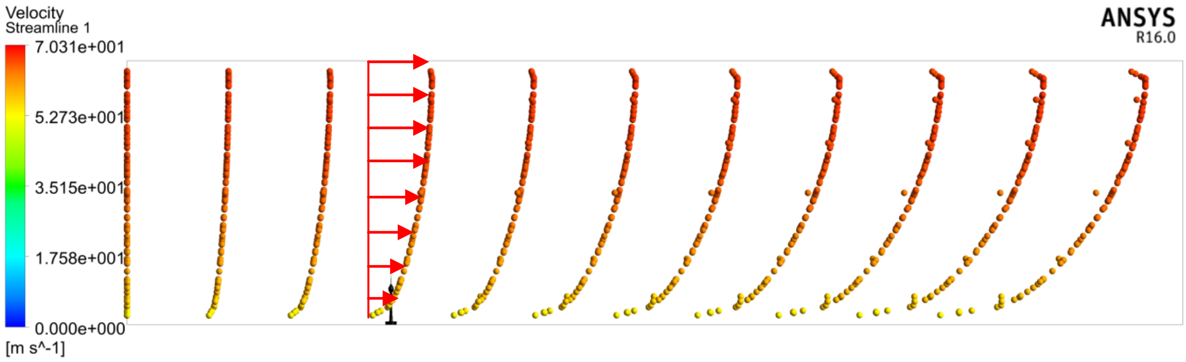

Figure 8: Wind profile in CFD simulation captured at intervals of 10 seconds within the domain

4. Results

The flow of wind around the building in its symmetrical plane is as shown in Fig. 9. The wind approaches the building from the left side and flows around the buildings and leaves on its downstream to the right. It can be seen that the wind significantly retards to a minimum at the face of the building surface and at the wake, it forms a vortex. Furthermore the velocity of the wind approaching the buildings depending on height and location is between the range of 40m/s to 70m/s (according to the wind profile use –terrain category 2: AS1170.2) however at the edges, the speed significantly increases to around 100m/s. This can be depicted from Fig. 10 where the velocity of wind is measured on a horizontal plane at Z=240m above the ground which is, at the bulb of the building. The velocity contour map at a reference height of Z=100m (Fig. 11) above ground show that the acceleration of wind around the building at this point is far less than that which was experienced at Z=240m. This difference was anticipated as the incident velocity at Z= 240m is greater than that at Z=100m also the geometry of the bulb is such that its diameter is larger than that of the stalk of the building thus making the velocity around the building at this level to travel faster.

This result was also observed in previous research conducted by authors Damith et al. [18], where a 600m building of rectangular shape was used to perform the analysis. The behaviour of wind depicted by velocity contour map as shown in Fig. 12 in this study replicates the pattern of the wind flow to that which was obtained for the rectangular building. Furthermore the wind velocity at different levels also show similar patterns of flow development to that observed in Fig. 10 and Fig. 11.

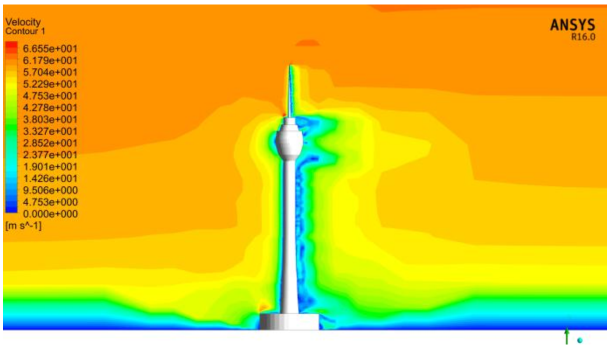

Figure 9: Wind flow around building

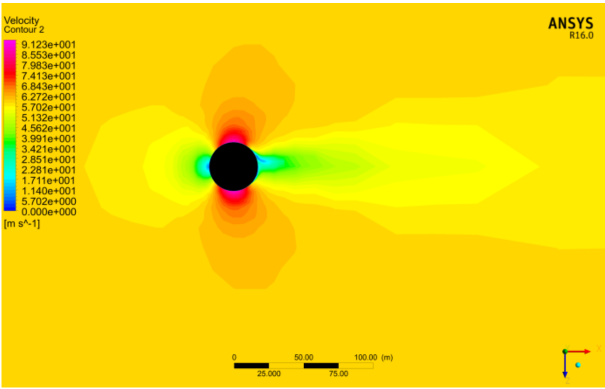

Figure 10: Velocity contour map at Z=240m

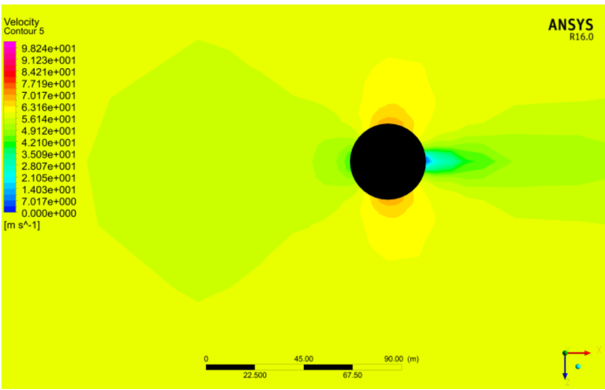

Figure 11: Velocity contour map at Z=100m

Predicting surface pressures accurately is important in the design of tall buildings as it will influence

the lateral strength and thus design of the building. Furthermore it also affects the design of facades on the building surface.
The pressure distribution on the face of the building (windward face) where the wind profile directly approaches it is as shown in Fig. 12 and the pressure distribution of leeward face is as shown in Fig. 13. From the pressure contours it can be observed that the on the windward face a positive pressure distribution is observed because of the incident pressure on the face. It must be noted that the pressure increases with the height of the building as it correlates directly to the velocity profile that is incident on the building, which also increases with the height of the building.
On the leeward face the pressures drop and cause suction as shown in Fig. 13 and also on the side of the building, where the flow of wind is greatest, it causes large suction and the pressure significantly drops as shown in Fig. 14.

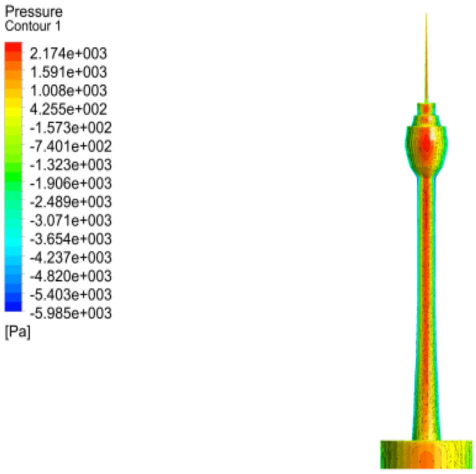

Figure 12: Pressure distribution on windward face

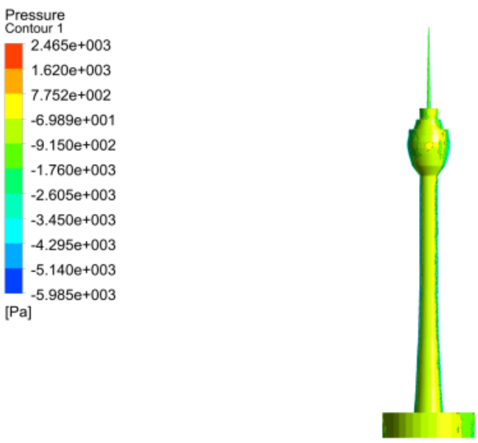

Figure 13: Pressure distribution on leeward face

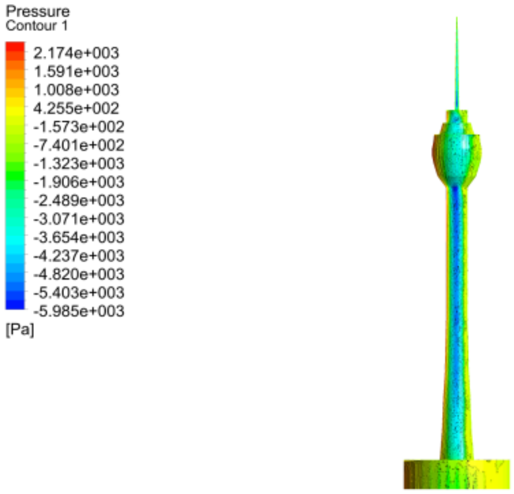

Figure 14: Pressure distribution on side of the building

These results that are obtained in this study are in good agreement with previous studies conducted by different authors where similar results were obtained for tall buildings.

Mean Cp values obtained for the cross section of the tower at 165m is presented in Fig. 14 (Series 6). The results were compared with the predictions from the previous researchers and the Euro code prediction. As it can be seen from Fig. 14, CFD prediction is shown a very good agreement with the experimental and code predicted data for Cp.

5. Conclusion

In the recent past, the application of CFD to analyse the flow of wind and its effects of buildings has greatly been investigated by many authors. Most of the research has been conducted on medium rise buildings whilst only a few has conducted research on buildings that are greater than 200m tall. The importance of the work presented in this paper is unlike previous work done by other authors, the shape of this building is not regular, it is complex in comparison to the conventional rectangular model where various researchers have performed tests.
Currently this paper shows the preliminary work conducted on this complex building to understand the flow of wind around this structure. The results show good correlation to previous work conducted and shows that the application of CFD for this model can be greatly beneficial and is likely to produce credible results that can be used as proof in the future for the design of any tall buildings of unorthodox shapes.

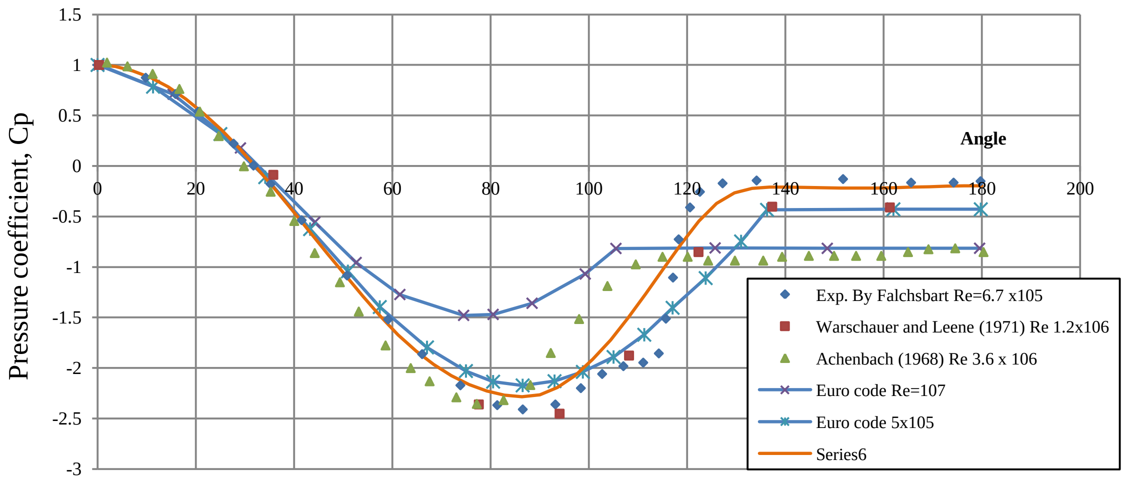

Figure 14: Pressure coefficient, C_p at 165m height

Acknowledgement
Authors would like to acknowledge the PCU structural consulting team of University of Moratuwa, especially Prof Priyan Dias and Mr. Shiromal Fernando for providing data about the Lotus tower.

References

[1]. Hollister, N. and Wood, A., Market Trends: The 20 Tallest in 2020: Entering the Era of the Megatall. Elevator World, 2012. 60(3): p. 38.

[2]. Safarik, D., Wood, A., and Gerometta, M., CTBUH Year in Review : Tall Trends an All-Time Record 97 Buildings of 200 Meters or Higher Completed in 2014. 2015.

[3]. Blocken, B., Stathopoulos, T., Carmeliet, J., and Hensen, J.L., Application of Computational Fluid Dynamics in Building Performance Simulation for the Outdoor Environment: An Overview. Journal of Building Performance Simulation, 2011. 4(2): p. 157-184.

[4]. Blocken, B., Janssen, W., and van Hooff, T., CFD Simulation for Pedestrian Wind Comfort and Wind Safety in Urban Areas: General Decision Framework and Case Study for the Eindhoven University Campus. Environmental Modelling & Software, 2012. 30: p. 15-34.

[5]. Hang, J., Li, Y., Buccolieri, R., Sandberg, M., and Di Sabatino, S., On the Contribution of Mean Flow and Turbulence to City Breathability: The Case of Long Streets with Tall Buildings. Science of the Total Environment, 2012. 416: p. 362-373.

[6]. Ansys V16. 2016, SAS IP, Inc.

[7]. Zhang, C.X., Numerical Predictions of Turbulent Recirculating Flows with a K-ε Model. Journal of Wind Engineering and Industrial Aerodynamics, 1994. 51(1994): p. 177-201.

[8]. Blocken, B., Carmeliet, J., and Stathopoulos, T., CFD Evaluation of Wind Speed Conditions in Passages between Parallel Buildings—Effect of Wall-Function Roughness Modifications for the Atmospheric Boundary Layer Flow. Journal of Wind Engineering & Industrial Aerodynamics, 2007. 95(9-11): p. 941-962.

[9]. Parente, A., Gorlé, C., van Beeck, J., and Benocci, C., Improved K–ε Model and Wall Function Formulation for the Rans Simulation of Abl Flows. Journal of Wind Engineering & Industrial Aerodynamics, 2011. 99(4): p. 267-278.

[10]. Lou, W., Huang, M., Zhang, M., and Lin, N., Experimental and Zonal Modeling for Wind Pressures on Double-Skin Facades of a Tall Building. Energy and Buildings, 2012. 54: p. 179-191.

[11]. Franke, J., Hirsch, C., Jensen, A.G., Krüs, H.W., Schatzmann, M., Westbury, P.S., Miles, S.D., Wisse, J.A., and Wright, N.G. Recommendations on the Use of CFD in Wind Engineering, Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics, 2004. von Karman Institute, Sint-Genesius-Rode, Belgium.

[12]. Blazek, J., Computational Fluid Dynamics: Principles and Applications:(Book with Accompanying CD) 2005: Elsevier.

[13]. Chok, K., Ando, Y., and Vytla, K.S.V. Computational Fluid Dynamics in the Design of High-Rise Structures, CTBUH 9th World Congr. , 2012. Shanghai

[14]. Li, B., Liu, J., Luo, F., and Man, X., Evaluation of CFD Simulation Using Various Turbulence Models for Wind Pressure on Buildings Based on Wind Tunnel Experiments. Procedia Engineering, 2015. 121: p. 2209-2216.

[15]. Liu, X., Niu, J., and Kwok, K.C. Evaluation of Rans Turbulence Models for Simulating Wind-Induced Mean Pressures and Dispersions around a Complex-Shaped High-Rise Building, Building Simulation, 2013. Springer.

[16]. Dagnew, A.K., Bitsuamalk, G.T., and Merrick, R. Computational Evaluation of Wind Pressures on Tall Buildings, 11th American conference on Wind Engineering. San Juan, Puerto Rico, 2009.

[17]. Standards Australia, AS/NZS-1170.2, in Structural Design Action-Part 2: Wind Actions. 2011: Sydney.

[18]. Mohotti, D., Mendis, P., and Ngo, T. Apllication of Computational Fluid Dynamics (CFD) in Wind Analysis of Tall Buildings, 23rd Australasian Conference on the Mechanics of Structures and Materials (ACMSM23), 2014. Byron Bay, Australia.