Nineteenth Australasian Wind Engineering Society Workshop, April 4-6, 2018, Torquay, Victoria

Predicting Wind-Induced Pressure on Slender Tall Structures Using Steady and Unsteady RANS CFD Analysis

Kasun Wijesooriya[1], Damith Mohotti[1]*,
[1]School of Civil Engineering, Faculty of Engineering and IT, university of Sydney, Darlington, NSW, 2006, Australia
*Correspondence e-mail: damith.mohotti@sydney.edu.au

ABSTRACT

This paper presents numerical and experimental study performed on a 406 m tall slender structure with a circular cross-section. One of the objectives of this study is to provide an alternative engineering approach for predicting wind induced pressures on tall structures. Experimental tests were carried out in a wind tunnel where a scaled down model was subjected to turbulent flow. Computational Fluid Dynamics (CFD) simulation including a polyhedral mesh with SRANS and hybrid RANS-LES turbulence models was used in this study. It is shown that SRANS SSTkω model can predict the pressure variations on the windward and crosswind faces and show a good agreement with the experimental data, whilst over predicting leeward pressures. The hybrid RANS-LES, DDES turbulence model showed improved predicament of pressures all three sides including the leeward pressure, which shows that a full LES turbulence model is not required. Furthermore, the importance to employing stringent first cell height of $y^+ \leq 1$ is highlighted.
Keywords: Super Tall Structures, Slender Structures, Wind Tunnel, CFD, RANS, Hybrid-LES

1. Introduction

Super tall buildings are a common featurette in many modern city skylines. With the advancement of construction materials and techniques, the ability to build such structures have become possible. Super tall buildings, which are usually of heights greater than 300 m Hollister and Wood (2012), experience large lateral loads due to wind. Furthermore occupancy comfort criteria that arise due to sway motion under wind excitation can also be a governing factor in most cases.

The design of super tall buildings fall out of the scope of most modern design standards. In the Australian standard Standards Australia/New Zealand (2011), a limitation of 200 m is imposed in terms of height and further limitations apply which depends on the structures natural frequency. Wind tunnel testing is the industrial standard for super tall structures where the assessment of wind is concerned. Three different types of methodologies in the assessment of wind in the wind tunnel, which are; high frequency base balance (HFBB), high frequency pressure integration (HFPI) and aero-elastic modelling.

Computational fluid dynamics (CFD) is a computer based numerical analysis that utilizes a combination of finite element modelling and mathematical computations, in the form of Navier Stokes equations, to simulate fluid flow in a given domain. It is a popular tool and is vastly employed in the aerospace industry where in design companies such as Boeing, certain features of their commercial planes are entirely designed using CFD whilst other components incorporate CFD Spalart and Venkatakrishnan (2016). CFD however is an evolving tool in the computational wind engineering (CWE) domain. Currently is has been used in various aspects such as estimating pressure loads on buildings, pedestrian wind comfort, pollution dispersion and wind energy harvesting Montazeri and Blocken (2013, Ozmen et al. (2016, Perén et al. (2015). The main advantage of CFD is the ability to model at the real scale instead of model scale unlike wind tunnel experimentations. For tall structures with geometries that include sharp edges that advocate separation of flow, the possible influence of reduced Reynolds number effect found in the model scale may not be a concern. However, circular or ellipsoid geometries are sensitive to Reynolds number which influence position of flow separation and resulting aerodynamic forces such as drag and lift. A few examples on the effects of Reynolds number on circular cylinders have been studied by Roshko (1961), Achenbach (1968) and more recently Miau et al. (2011). Furthermore the ability to visualize flow at any given instant, reduced operational costs and the ability to perform simulations within a reasonable time are some reasons why CFD has recently gained an appreciation in CWE.

In this study a super-tall slender structure of 406 m which consists of circular cross-sections of varying diameters is used for the estimation of wind loads. Tests conducted in the University of Sydney boundary layer wind tunnel (BLWT) is compared to those results obtained from a CFD simulation. The CFD tool used in this study is ANSYS FLUENT version 18 ANSYS® (2017). A comparison of wind pressures and appropriate modelling techniques required for CFD is proposed for such a structure. A graphical representation of the structure displayed in Figure 1.

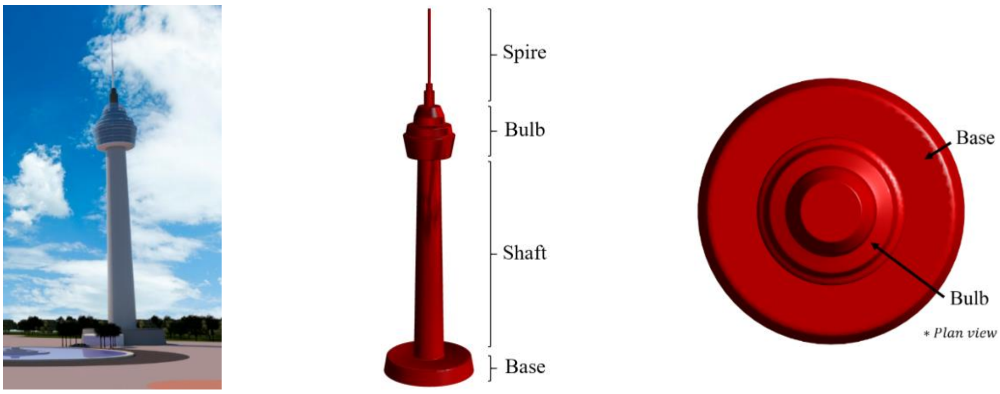


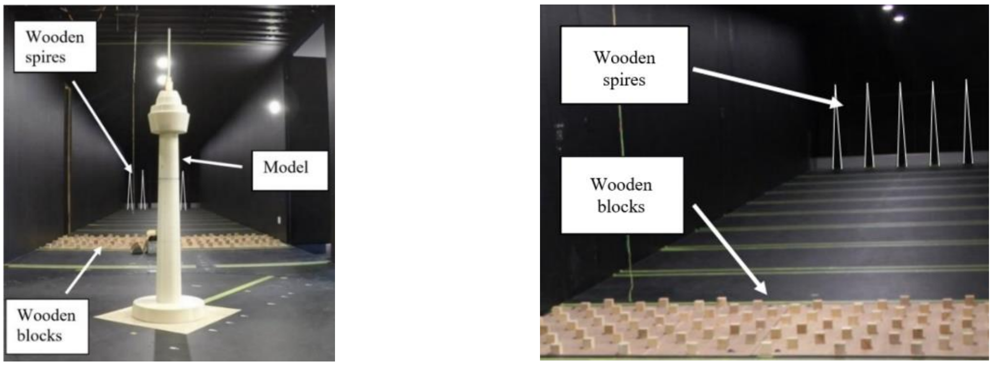
Figure 1: Graphical representation of the tower used in this study

2. Experimental modelling

The wind tunnel used for this study was the BLWT in the School of Civil Engineering at The University of Sydney. The tunnel is a closed loop system with a testing section measuring 19 m long, 2.5 m wide and 2 m high. It is capable of handling a top speed of 27 m/s at full capacity. The building model is scaled down by 1:300. It was subjected to a turbulent boundary layer flow with the use of roughness elements as shown in Figure 2a & b. The normalised velocity and turbulence intensity profile subjected in the wind tunnel is as shown in Figure 3 and closely depicts that of TC-2 in the Standards Australia/New Zealand (2011).

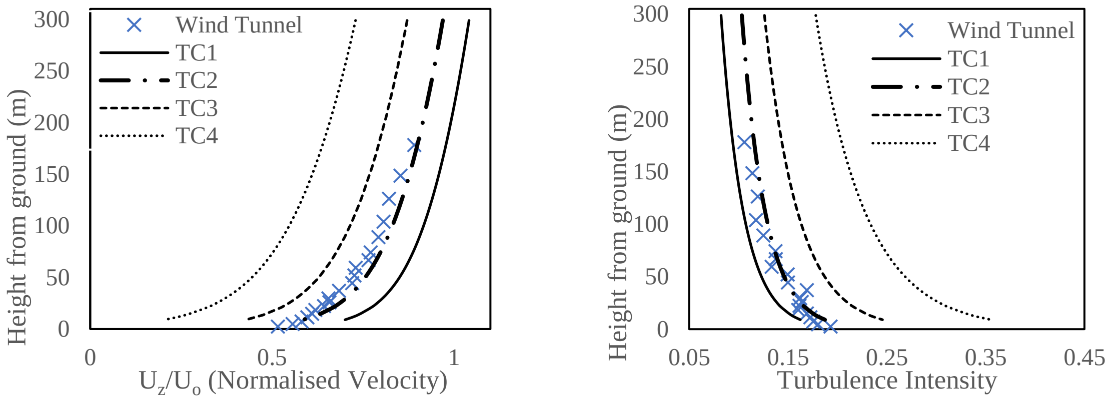
3. Numerical modelling

Turbulence modelling is the heart of CFD where the fluid flow is modelled. There are many turbulence models available in the FLUENT package that range from Reynolds averaging Navier-Stokes (RANS) to scale resolved models. However the application of an appropriate model with correct boundary conditions and mesh parameters remain a challenge for most CFD users. In this study RANS turbulence model is used in the initial phase of the study. These tests were performed in a steady state where time is not a varying function. The scaled building is nested in a domain much like a wind tunnel as shown in Figure 4a. Initially three mesh schemes were proposed of which an optimum was selected as shown in Figure 4b. with polyhedral cells and a final count of 1.5 million elements. A second phase



(a) (b)

Figure 2: Wind tunnel setup: (a) the building, (b) roughness elements used in the wind tunnel



a: Normalised velocity vs height b: Turbulence intensity vs height

Figure 3: Normalised velocity and turbulence intensity recorded in the wind tunnel

of study was conducted where transient tests were undertaken where tests are conducted as a function of time. Both average and instantaneous variations of flow can be observed from such a test much similar to the wind tunnel. The boundary conditions such as velocity profile and turbulence intensity should match in the CFD domain for it to be comparable to that of the experimental. As shown, Figure 5 displays the comparisons made in an empty domain where profiles of velocity and turbulence measured at the building location within the domain.

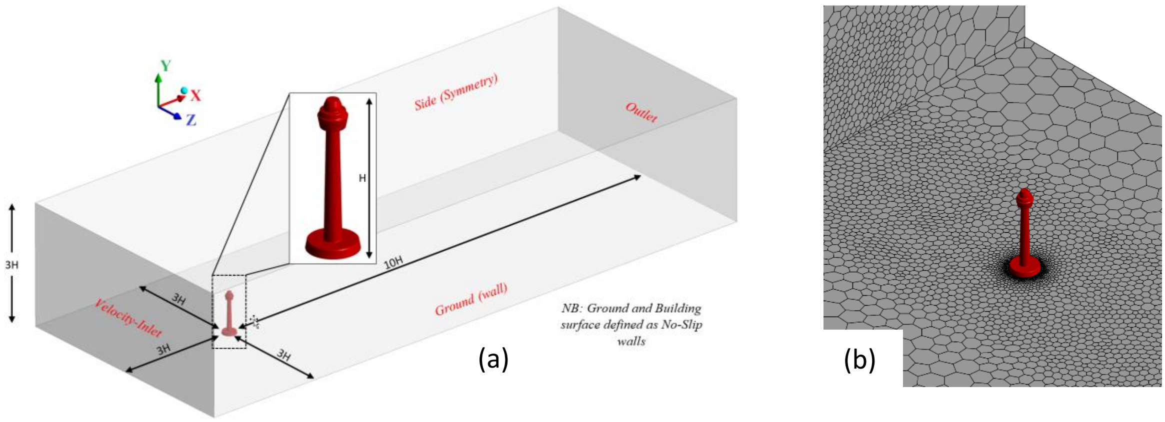
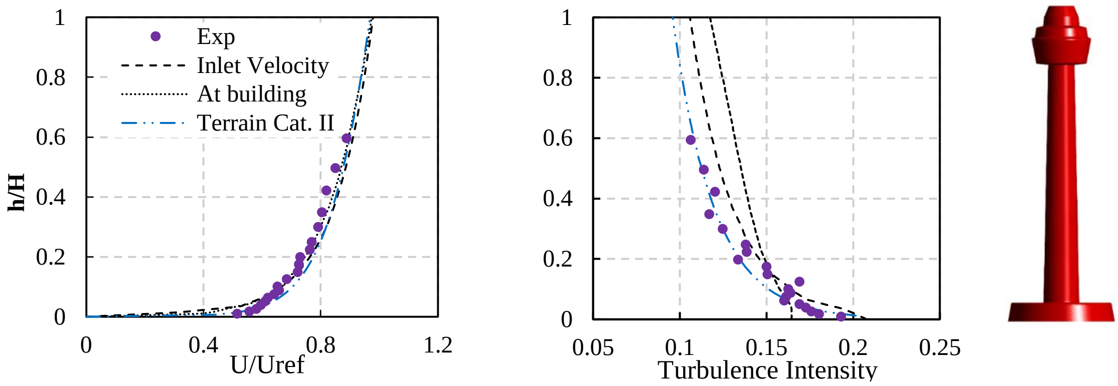


Figure 4: (a) Computational domain size and arrangement of boundaries, (b) Close up mesh arrangement for final setup and polyhedral cell visualization



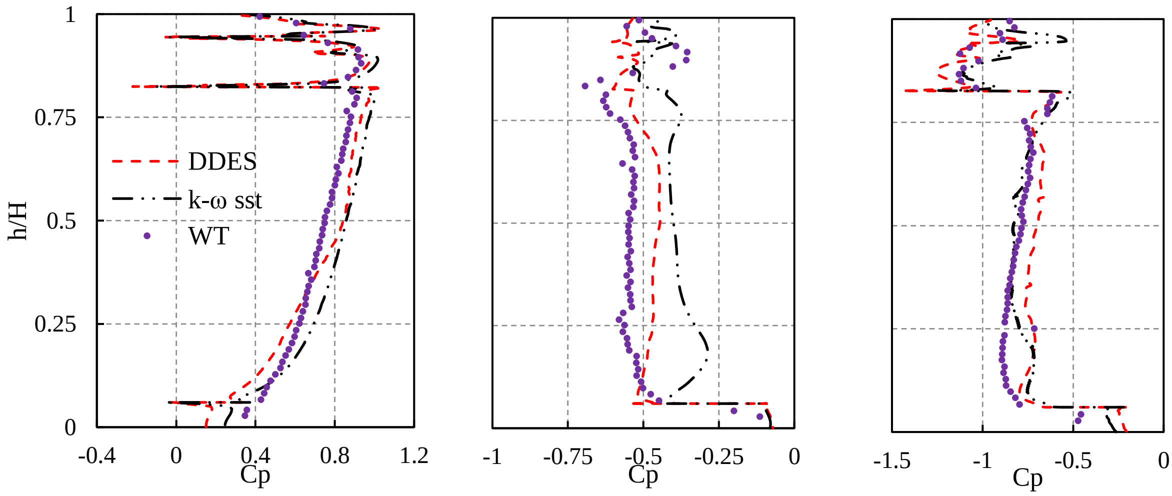
(a) Velocity profile (b) Turbulence intensity profile

Figure 5: A comparison of profiles measured numerically (inlet and building location), in the wind tunnel and AS/NZS 1170.2:2011

4. Results and Discussions

For the steady state simulation $SST\ k-\omega$ (SSTkω), developed by Wilcox (1988) later modified by Menter (1994), turbulence model was used. The SSTkω is best suited in situations where the effects of laminar sub layer flow is essential at predicting separation of flow. Since the structure consists of circular cross-sections, the SSTkω option was opted in conjunction of a $y^+ \leq 1$ mesh at the boundary of the building. For the transient simulation, a hybrid RANS-LES turbulence model was opted. Delayed detached eddy simulation (DDES) was used as it would considerably lower the computational costs in comparison to a large eddy simulation (LES). The DDES employs transient SSTkω model at the boundary layer whilst switching to LES behaviour in regions outside the boundary layer by a blending function ANSYS Inc. (2013). A turbulent inflow at the inlet is generated by employing the spectral synthesiser option in FLUENT.

The results for the mean pressure coefficient (C_{pe}) variation along the height of the building concerning the windward, leeward and crosswind faces are presented in Figure 6. It can be said that the SSTkω model is well suited at predicting the pressures. However SSTkω fails at predicting the leeward face pressure.



(a) Windward (b) Leeward (c) Crosswind

Figure 6: Mean pressure distribution comparisons for different turbulence models

In the case of DDES a slight improvement of windward pressure is observed whilst a substantial improvement is observed in the leeward face with retrospect to values and shape of graph. The crosswind face showed the same variation albeit for a small degree of over prediction in pressures. This improvement for the leeward pressure is attributed towards the scale-resolving component of the turbulence model. Whilst the SSTkω model was adequate to predict pressures from the windward to the crosswind face, it failed to predict leeward pressures because unlike DDES contribution from the large scale fluctuations which are not present in SSTkω. Furthermore, the assumption of smooth inflow that is used in steady RANS, can also inhibit the accuracy of the leeward pressures. This observation was also made by Dagnew and Bitsuamlak (2014) where different inflow generators produced varying leeward pressures. Figure 7 shows pressure coefficient and streamline plots at an instantaneous time flow. High pressures are observed at the bulb of the structure and flow patterns indicate circulation of flow around the structure which correspond to shedding of vortices.

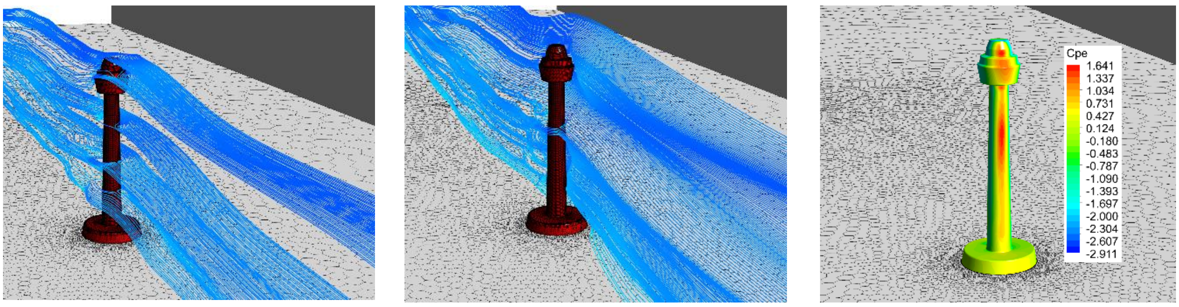


Figure 7: Instantaneous velocity stream and pressure coefficient obtained from analysis

Figure 8 shows the fluctuation of the longitudinal velocities recorded for the transient DDES simulation, wind tunnel experimentation and the theoretical von-Karman spectrum. A strong correlation is observed between the numerical and wind tunnel simulations which further enhances the validity of the experiment.

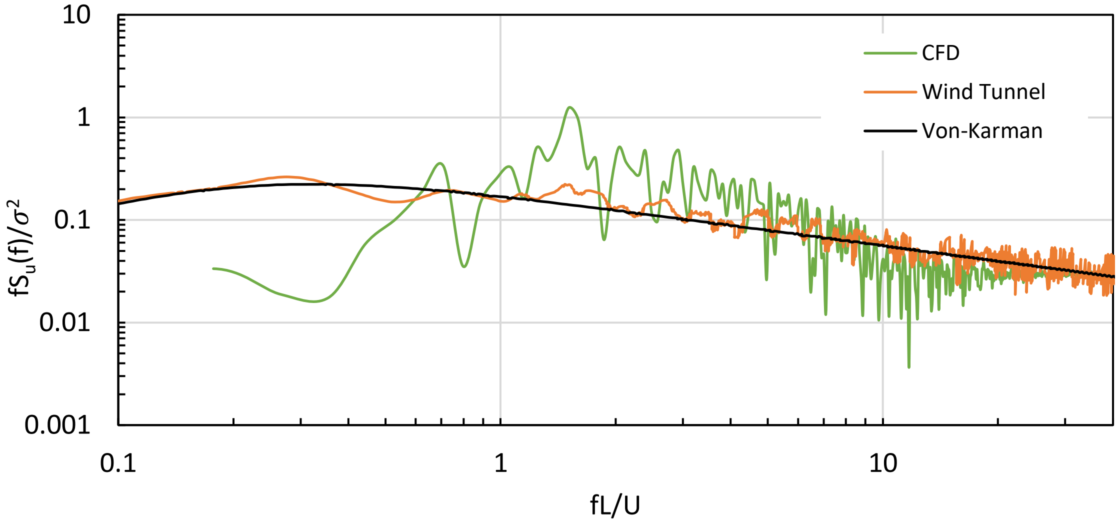


Figure 8: Simulated velocity spectrum for numerical, experimental vs the theoretical von-Karman spectrum

5. Conclusions

This paper presented a comparative study performed on wind pressures on uniquely shaped tall slender structure of circular cross-sections. The main focus of this study was to develop a practical approach in the use of CFD by employing SRANS and hybrid LES transient turbulence models. It was shown that for the prediction of mean pressures the SSTkω RANS model is sufficient (with $y^+ \leq 1$) with a slight over-prediction of leeward pressures. The DDES model showed better accuracy in terms of mean pressures which attributes towards its capability as a model and the use of a turbulent inflow generator. This study is particularly important from a structural engineer's perspective as most CFD studies have been conducted on rectangular sections. This study showed that even with the use of a RANS model, given that an appropriate global mesh, wall mesh and inlet conditions are given, sufficient accuracy in the predicament of pressure can be obtained within 5 % error (leeward face 15 %). For further accuracy the transient hybrid-LES model is more than sufficient as shown in these studies. The RANS simulation only took 4 hours to compute on a 4-core intel I7 machine whilst the DDES simulation took less than 20 hours on a 32-core HPC. Finally, the time taken for numerical simulations is less than conducting a wind tunnel study where at the conception of a project such a test programme would not be feasible.

References

Achenbach, E. (1968). Distribution of local pressure and skin friction around a circular cylinder in cross-flow up to Re = 5 × 10^6. *Journal of Fluid Mechanics,* 34**,** 625-639.

Ansys Inc. (2013). *ANSYS Fluent User's Guide,* ANSYS, Inc. Southpointe 275 Technology Drive Canonsburg, PA.

Ansys® (2017). *Academic Research Fluent, Release 18*.

Dagnew, A. K. and Bitsuamlak, G. T. (2014). Computational evaluation of wind loads on a standard tall building using LES. *Wind and Structures,* 18**,** 567-598.

Hollister, N. and Wood, A. (2012). The Tallest 20 in 2020: Entering the Era of the Megatall. *CTBUH Journal,* 2012.

Menter, F. R. (1994). Two-equation eddy-viscosity turbulence model for engineering applications. *AIAA,* 32**,** 1598-1605.

Miau, J. J., et al. (2011). Experiment on smooth, circular cylinders in cross-flow in the critical Reynolds number regime. *Experiments in Fluids,* 51**,** 949-967.

Montazeri, H. and Blocken, B. (2013). CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: validation and sensitivity analysis. *Building and Environment,* 60**,** 137-149.

Ozmen, Y., et al. (2016). Wind flow over the low-rise building models with gabled roofs having different pitch angles. *Building and Environment,* 95**,** 63-74.

Perén, J. I., et al. (2015). CFD analysis of cross-ventilation of a generic isolated building with asymmetric opening positions: Impact of roof angle and opening location. *Building and Environment,* 85**,** 263-276.

Roshko, A. (1961). Experiments on the flow past a circular cylinder at very high Reynolds number. *Journal of Fluid Mechanics,* 10**,** 345-356.

Spalart, P. R. and Venkatakrishnan, V. (2016). On the role and challenges of CFD in the aerospace industry. *The Aeronautical Journal,* 120**,** 209-232.

Standards Australia/New Zealand 2011. Structural design actions, Part 2: Wind actions, AS/NZS 1170.2:2011(R2016).

Wilcox, D. C. (1988). Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal,* 26**,** 1299-1310.