19th Australasian Fluid Mechanics Conference
Melbourne, Australia
8-11 December 2014

Turbulent Airflow past a Rectangular Cylinder Building

T.E. Downie[1] and B.P. Huynh[2]

[1]Faculity of Engineering and Information Technology
University of Technology, Sydney 2000, Australia

[2]Faculity of Engineering and Information Technology
University of Technology, Sydney 2000, Australia

Abstract

The successful use of Computational Fluid Dynamics (CFD) in structural wind engineering applications would significantly reduce the time and resources required compared to scale model testing in a physical wind tunnel. The current industry standard for calculating allowable façade loads on medium and high rise structures involves building and hot tapping a detailed scale model and testing in a physical wind tunnel; this is a time and labour intensive process which could greatly benefit from the successful application of CFD.

This paper uses a case study of a rectangular cylinder building to compare the mean pressure coefficient results simulated using CFD k- ω Shear Stress Turbulence (SST) model with wind tunnel measurements. It analyses findings at 0°, 30°, 60° and 90°, in turbulent conditions, at 143 separate points for each angle. Previous research with this model for this application has focused on square cylinders, minimal points of measurement and common angles of incidence. The wind tunnel used was a commercial blockage-tolerant boundary layer wind tunnel and the CFD simulation was run using Ansys CFX 14.0.

The wind tunnel results showed mean values falling within a pressure coefficient range of ±1, except for those points located adjacent to the leading edge on the roof. The CFD results showed a good agreement with the mean values obtained in the wind tunnel test for central windward points and locations that were not immediately adjacent to an edge. However a significant over prediction of the pressure coefficient up to 250% was found in areas of separation.

The k-ω SST model claims to perform better in areas of adverse pressure gradient and, consequently, areas of separation than the standard k-ω model, k-ε model or their variants. This paper demonstrates that whilst such claims may be correct, the k-ω SST model is still not at an acceptable standard for commercial wind engineering applications.

Introduction

Murakami [6] suggested there were a number of difficulties for Computational Wind Engineering (CWE) to overcome in the future. These difficulties arise from four main factors: the high Reynolds number, complexity of flow field with impingement, sharp edges of bluff bodies and the inflow and outflow conditions common to most CWE applications.

If these challenges are overcome, CWE could offer a number of benefits over scale model wind tunnel testing. The ability to model the simulation at full scale and dictate the exact velocity profiles and turbulence quantities measured from the field would allow a significantly higher degree of accuracy. Furthermore the reuse and flexibility of 3D models would allow the creation of virtual cities, greatly reducing set up time.

Case Study

To limit the scope of this paper a single case study was selected. A rectangular cylinder building of dimensions 30 x 12.6 x 60 meters was chosen to represent a common medium-rise structure (Figure 1).

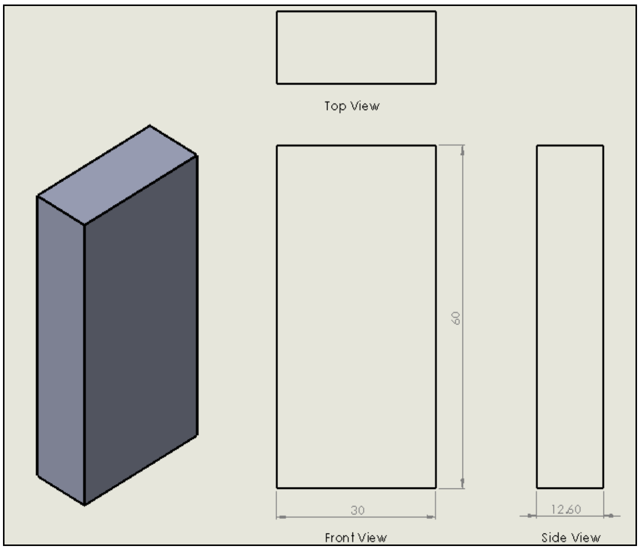


Figure 1. Orthogonal view of rectangular cylinder for case study

The two methods were compared by pressure coefficient results as this is the common practise when presenting façade load test outcomes in commercial consulting. The pressure coefficient is defined as: the pressure caused by wind at a point divided by the dynamic wind pressure in the free stream region [1], equation (1).

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^{2}} \qquad (1)$$

The maximum permissible stress design (in kPa) can be derived by multiplying the pressure coefficient by the equivalent full-scale reference pressure given in AS/NZS 1170.2 Structural design actions [9].

The k-ω Shear Stress Transport model was selected as the turbulence model due to its relatively low computational cost and claimed effectiveness in adverse pressure gradient conditions. There is limited research on the accuracy of this model, particularly regarding varied degrees of incidence, and thus the case study was examined at 0°, 30°, 60° and 90°. The discrepancy in pressure coefficient results between traditional wind tunnel methods and CFD is the focus of this paper.

Wind Tunnel Testing

The aim of the wind tunnel tests was to set up a baseline for comparison using proven methods employed commercially by wind engineering consultants. Pressure coefficient results were collected under turbulent conditions on the windward, sides, leeward and roof of the rectangular cylinder scale model at 143 points for 36 directions.

Testing was performed in a blockage tolerant boundary layer wind tunnel (Figure 2). The upstream conditions were modeled with vertical spires, barriers and roughness blocks as specified in the AS/NZS1170.2 [9] for 1:100 scale, category 2 tests. Category 2 represents open terrain and was selected to limit the test focus.

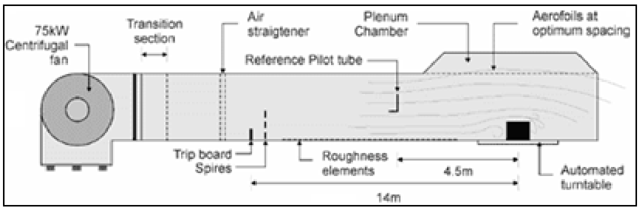


Figure 2. Layout of the wind tunnel used for testing

The wind tunnel was initially run with an empty fetch to obtain zero and dynamic references using both a manometer and pitot tube. Each reference was taken at the building's height after 60 seconds with the dynamic reference captured at 90% fan speed. The wind tunnel for this testing was regularly used for commercial projects and internal calibration results compared to International Standards [4] can be seen in Figure 3.

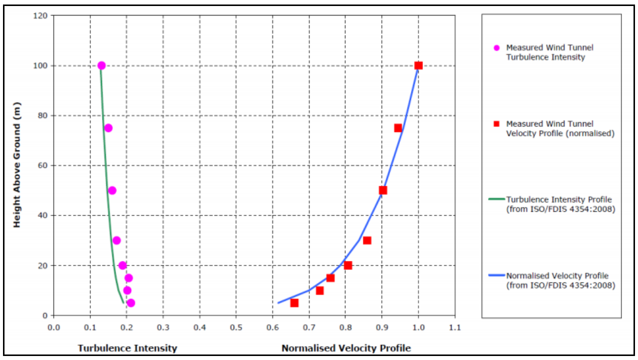


Figure 3. Tunnel vs. ISO4354: Wind actions on structures profiles

The 1:100 scale model was constructed using medium density fiber board and pressure tapped with 1.4mm diameter soft PVC tubing of 1000mm and 1500mm lengths dependant on location. The model was placed in the centre of a blockage tolerant, boundary layer wind tunnel turntable (Figure 4) with the long side perpendicular to the fan (0°) and its base nailed to the floor. The top of the model was tethered to reduce vibration and simulate rigid building conditions. The pressure taps were then puff tested for response, collared and connected to their respective pressure acquisition boxes.

Figure 4. The wind tunnel as prepared for testing

A direction was sampled for 32 seconds before the turntable rotated 10° and a new sample begun. Static pressure results were acquired at each tap through the software LabView 7 and subsequently post processed to convert the raw voltages into pressure data and exported into an Excel spread sheet. A graph for each point showing the maximum, minimum, mean and standard deviation of the pressure coefficient at each direction was created using macros (Figure 5).

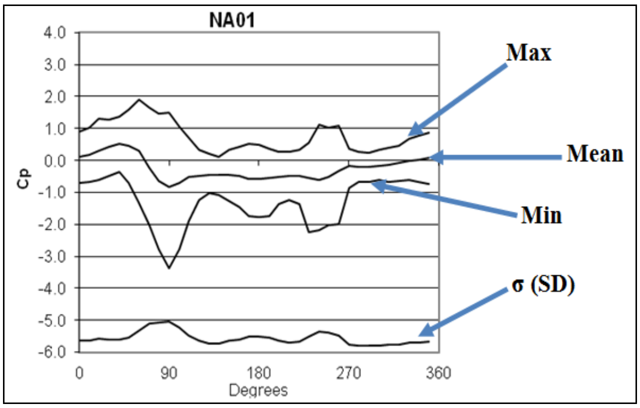


Figure 5. Example results of pressure tap NA01

All pressure coefficient means fell within a range of +/-1. Positive values, representing flow moving toward the face, appeared on the windward face, with all other faces showing negative numbers as air moved away.

Computational Fluid Dynamics Simulations

The aim of CFD simulations was to replicate the results obtained in the wind tunnel tests and in doing so assess the k-ω SST model's applicability to calculating allowable façade loads for a medium rise building. Four simulations were run in turbulent conditions at the incident angles of 0°, 30°, 60° and 90° and pressure coefficient data was assessed at 143 points across the windward, sides, leeward and roof faces of the rectangular cylinder.

The computational domain was designed using Huang's [3] spatial recommendations but modified to include an increased downstream length to avoid backflow errors. An orthogonal view of the computational domain is shown in Figure 6.

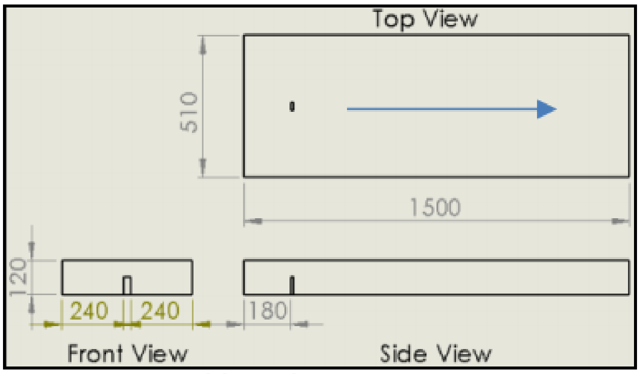


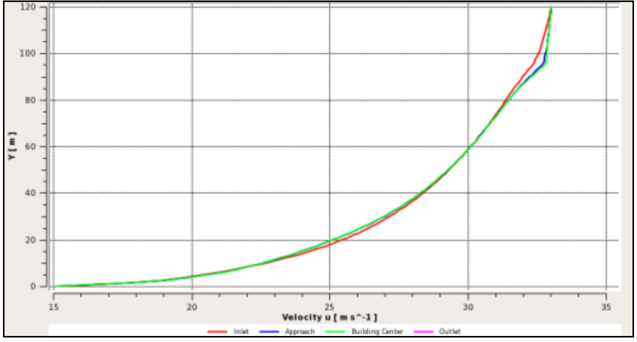
Figure 6. Computational domain dimensions (m)

The velocity profile was generated in excel using the $\frac{1}{7}$ power law and the same meteorology data used in the wind tunnel test. Likewise the turbulence intensity and length scale were obtained from AS/NZS 1170.2 Structural design actions (2011) and generated at the inlet of the flow domain using the read-from-file option.

The ground surface was defined as a rough wall with physical height of 0.6 meters to reproduce the wind tunnels 0.03 meters using equation (2) as defined by Hargreaves and Wright [2]. Effort was made to ensure that the centre of the wall-cell adjacent to the ground surface was greater or equal to the physical roughness.

$$K_s = 20z_0 \tag{2}$$

The shear roof boundary condition was implemented using the Richards and Norris [7] method to eliminate profile decay along the fetch. Initial tests were run with an empty fetch to confirm a horizontally homogenous atmospheric boundary layer (Figure 7).



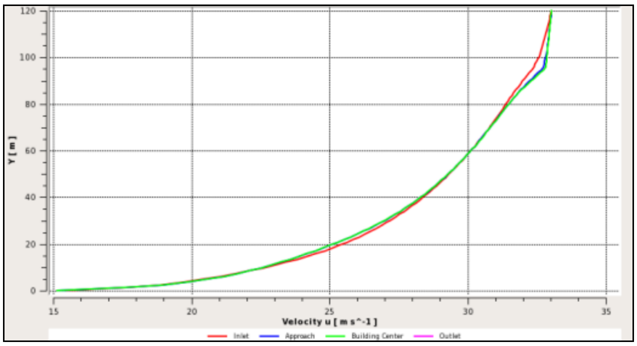


Figure 7. Velocity profile in empty domain

The fluid domain was automatically meshed using CFX 14.0's high resolution unstructured tetrahedral method and manually altered with inflation, face and edge spacing constraints (Figure 8). A 3-grid independence test was run to ensure consistency of results.

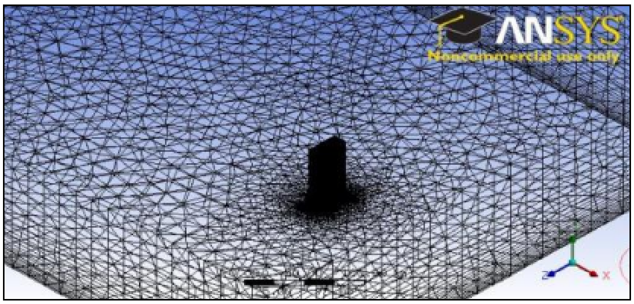


Figure 8. Detail: mesh set up for 0°

The transient simulation was run with the time step set to 2 $x10^{-3}$ seconds and was considered to reach convergence when the relative residual RMS error values reached less than $1x10^{-5}$. The solution was obtained using local parallel computations at double precision. A user defined function was written to calculate the mean pressure coefficient at a point, and each desired point was manually defined using the probe function.

For the change of angle the geometry was modified and the entire process regenerated. The computational domain was rotated around the building so the same location could be used for each probe point. The mean pressure coefficient contour plots and 2D streamline profiles shown in Figure 9 and 10 give a good indication of the flow experienced by the rectangular cylinder building during the simulation. Warm colors represent positive pressure and indicate airflow moving toward the surface; inversely cool colours represent negative pressure indicating airflow moving away from the surface.

For all directions, pressure coefficient magnitudes were highest at the windward face due to the direct, oncoming wind velocity. Negative readings were found on the side, roof and leeward faces indicating that the wind flow moved away from these surfaces.

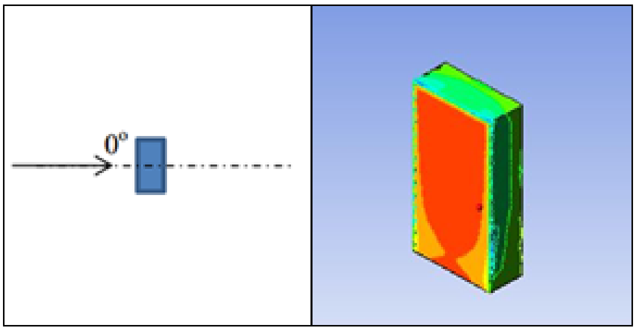


Figure 9. Isometric contour plot for the 0° simulation

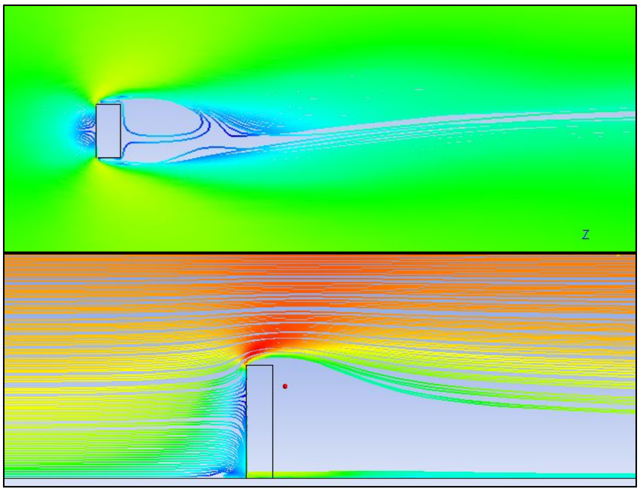


Figure 10. 2D streamline profiles for the 0° simulation

Comparison of Results and Discussion

Figure 11 shows the results of both the wind tunnel and CFD tests for the 0° windward face. Many of the CFD results fell within 5% of the wind tunnel data in central regions but reduced significantly in accuracy as an edge was approached. Overall a good indication of flow direction and magnitude can be discerned, particularly in the central regions.

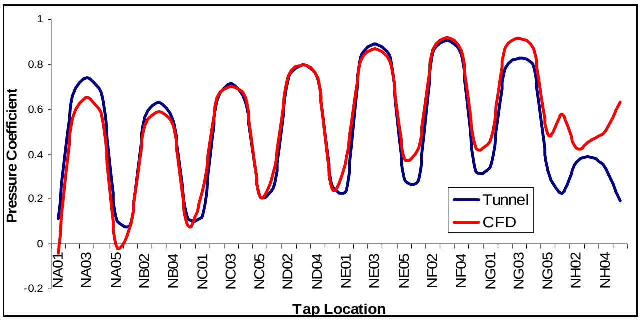


Figure 11. Windward trend graph for 0° tests

Figure 12 shows the results for the roof at 0°. The two rows of points immediately adjacent to the front edge of the roof suffer from a significant over prediction; this is consistent with previous studies [5] with the k-ω SST model's known issues in areas of separation and CFD's inability to accurately handle sharp edges of bluff bodies [6].

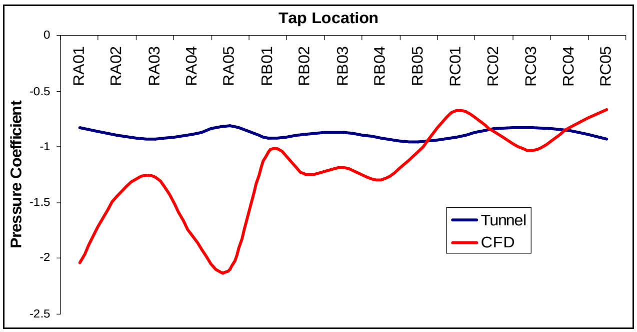


Figure 12. Roof trend graph for 0° tests

Figure 13 shows the pressure coefficient data for the side face at 0°. The majority of CFD results fall within 15% of the respective wind tunnel reading, however a substantial over prediction can be see on those points immediately adjacent to the leading edge. These errors are presented as 'spikes' on the trend graph and render the rest of the data difficult to read due the proportional difference between results at these locations.

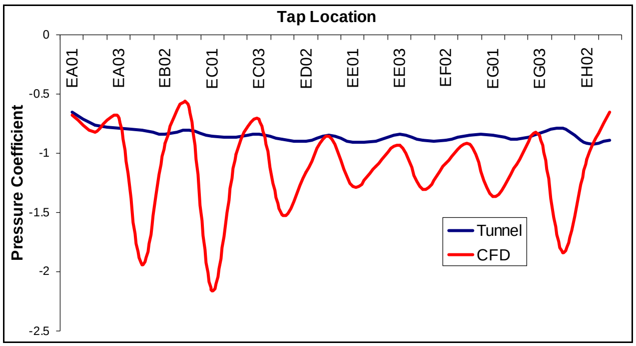


Figure 13. Side trend graph for 0° tests

Finally, Figure 14 shows the leeward face results for the 0° tests. A trend can be discerned, with the mean percentage difference between the methods being 13.11%. This is lower than the windward face, which was 28.85%, but this appears to be due primarily to a reduction in outliers.

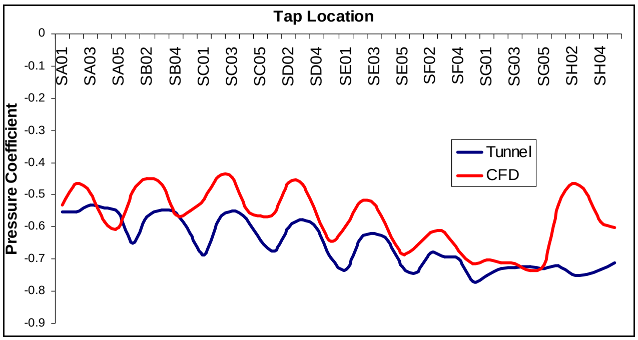


Figure 14. Leeward trend graph for 0° tests

Trend graphs for the 30°, 60° and 90° are not shown here due to the spatial limitations of this paper; however, consistent results were seen across all angles of incidence. The 30° and 60° tests faired slightly worse than the 0° and 90° tests in terms of percentage difference between the wind tunnel and CFD results, but again this is not surprising as additional points fell within areas of separation as the model was rotated and leading and trailing edges were created.

Conclusions

The results presented in this paper clearly show that the k-ω SST model is not yet at a commercial standard for computation wind engineering applications. Excellent agreement was found in central windward locations and the results of the two methods converged as points moved away from sharp edges for all faces, for all angles of incidence. These areas of separation greatly over predicted the pressure coefficient results by as much as 250%, despite the k-ω SST's turbulence production limiting factor.

Previous CWE pressure coefficient studies of this nature have mostly focused on square cylinders, with minimal points of measurement at common angles of incidence (0°, 45°). This paper, by addressing these areas of limited research, has attempted to close the gap in understanding CFDs application to wind engineering. Whilst some of the results are promising, it should be noted that only the mean pressures were analysed and the successfully simulation of maximum and minimum fluctuating pressures needs additional study and development. These pressures, representing gusts, are very important for façade loading studies but as of yet have had little success through CWE.

Until solutions to these problems and those raised by Murakami [6] are found, computational fluid dynamics' application to wind engineering is restricted to research. Strides have been made in recent years, particularly regarding the modeling of rough surfaces and a horizontally homogeneous atmospheric boundary layer, but still cannot offer a reasonable substitution for wind tunnel testing in a commercial setting.

References

[1] Etheridge, D.W. & Sandberg, M., *Building ventilation: Theory and Measurement*, John Wiley & Sons Chichester, USA, 1996.

[2] Hargreaves, D.M. & Wright, N.G., On the use of the k–ε Model in Commercial CFD Software to Model the Neutral Atmospheric Boundary Layer, *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 95, no. 5, pp. 355-69, 2007.

[3] Huang, S., Li, Q.S. & Xu, S., Numerical Evaluation of Wind Effects on a Tall Steel Building by CFD, *Journal of*

Constructional Steel Research, vol. 63, no. 5, pp. 612-27, 2007

[4] International Organization for Standardization, *Wind Actions on Structures*, ISO/FDIS 4354, ISO, Sydney, 2008

[5] Menter, F., Kuntz, M. & Langtry, R., Ten years of industrial experience with the SST turbulence model, *Turbulence, Heat and Mass Transfer*, vol. 4, pp. 625-32, 2003.

[6] Murakami, S., Overview of turbulence models applied in CWE–1997, *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 74, pp. 1-24, 1998.

[7] Richards, P.J. & Norris, S.E., Appropriate Boundary Conditions for Computational Wind Engineering Models Revisited, *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 99, no. 4, pp. 257-66, 2011.

[8] Rofail, A.W. & L.J.Aurelius, A Wind Tunnel Modelling Methodology for the Determination of the Internal and Net Design Pressures for Tall Buildings having Operable Facades, *International Conference on Building Envelope Systems and Technologies*, Sydney, 2004.

[9] Standards Australia, *Structural Design Actions 2011- Part 2: Wind Actions*, AS/NZS 1170.2, Standards Australia, Geneva, 2011.