View metadata, citation and similar papers at core.ac.uk

brought to you by CORE

provided by University of Strathclyde Institutional Repository

Strathprints Institutional Repository

Clannachan, G.H. and Lim, J.B.P. and Bicanic, N. and Taylor, I.J. and Scanlon, T.J. (2009) *Practical application of CFD for wind loading on tall buildings.* In: 7th International Conference on Tall Buildings, 2009-10-29 - 2009-10-30, Hong Kong, China.

Strathprints is designed to allow users to access the research output of the University of Strathclyde. Copyright © and Moral Rights for the papers on this site are retained by the individual authors and/or other copyright owners. You may not engage in further distribution of the material for any profitmaking activities or any commercial gain. You may freely distribute both the url (http://strathprints.strath.ac.uk/) and the content of this paper for research or study, educational, or not-for-profit purposes without prior permission or charge.

Any correspondence concerning this service should be sent to Strathprints administrator: mailto:strathprints@strath.ac.uk

http://strathprints.strath.ac.uk/

PRACTICAL APPLICATION OF CFD FOR WIND LOADING ON TALL BUILDINGS

Gordon H. CLANNACHAN[*], James B. P. LIM[*], Nenad BICANIC[**], Ian TAYLOR[***], Tom J. SCANLON[***]

[*]*Department of Civil Engineering, University of Strathclyde, John Anderson Building, Glasgow, G4 0NG, UK*
gordon.clannachan@strath.ac.uk

[**] *Department of Civil Engineering, University of Glasgow, Rankine Building, Glasgow, G12 8LT, UK*

[***] *Department of Mechanical Engineering, University of Strathclyde, James Weir Building, Glasgow, G1 1XJ, UK*

This paper is concerned with assessing the scope of applicability for computational fluid dynamics (CFD) in the field of structural engineering, with a particular reference to tall buildings. Modern design trends and advances in engineering materials have encouraged the demand for taller and more slender structures. This pattern induces inherent structural flexibility; these cases exceed the limitations of the quasi-static method offered by current codes of practice. Wind tunnel testing is the traditional solution for such dynamically sensitive structures. However, even this scaled modelling approach is clouded by some uncertainties, including scaling the Reynolds number and assuming damping values for the aeroelastic model. While CFD cannot be used as a replacement for wind tunnel testing, there are results within the literature to suggest it has the potential to act as a complimentary tool – provided it is used within its capabilities. The paper outlines the various turbulence models that are available and summarises the extent of their application in a practical structural engineering sense. It also details the user-defined criteria that must be satisfied and discusses the potential for simplified models in tall building CFD analyses, with a view to promoting more efficient and practical solutions.

Keywords: wind loading, CFD, turbulence modelling, surface pressure, tall buildings.

1. Introduction

Wind loading rivals seismic events as the dominant lateral loading for structures. Davenport (1998) classifies the aerodynamic wind forces acting on a typical structure into three categories. The first of these is the extraneously-induced loading caused by buffeting from the naturally turbulent oncoming wind. This can be enhanced from the wakes of upstream obstructions. Secondly, forces are induced from unstable flow phenomena such as separations, reattachments, and vortex shedding. The third source is caused by the movement-induced excitation of the body; the deflection of the structure creates fluid forces. This final unsteady phenomenon is only pertinent for highly flexible structures, such as long-span bridges. However, modern design trends and advances in engineering materials have encouraged the demand for taller and more slender structures.

This pattern induces inherent structural flexibility and heightens concerns regarding the aeroelastic fluid-structure interaction between the wind and the tall building.

Codes of practice have been formulated with a view to providing an acceptable balance between the overly complex reality and an oversimplified (conservative or unreliable) approach. In the UK, the current British Standard covering wind loading on buildings is BS 6399-2:1997. It is highlighted by Allsop (2009) that this is widely regarded as the most flexible and inclusive code for normal buildings. However, the introduction of the Eurocode means it is to be withdrawn by 31st March 2010 and replaced with BS EN 1991-1-4:2005. The quasi-static methods offered by both these codes are only applicable for buildings with structural properties such that they are not susceptible to dynamic excitation (Mehta, 1998). Thus, tall buildings, particularly those with high slenderness ratios and/or asymmetric plans, exceed these limitations and are advised to be tested in a wind tunnel.

Scaled-model wind tunnel testing is an established tool among industry design practices. Boundary layer wind tunnels are capable of quantifying time-dependent surface pressures, including the complex acrosswind and torsional loadings. They can also be used to determine the best orientation of the proposed building, as was performed for the Burj Dubai (Irwin and Baker, 2006). Nevertheless, it has its own limitations. These include the difficulty to maintain proportionality between the scaled turbulence characteristics and the scaled building model, especially if the topography is significant (Taranath 1998). Furthermore, it is important to ensure Reynolds number effects on the pressures are kept to a minimum. It is noted by Sun *et al*. (2009) that a computational approach has the capability of being more flexible than traditional wind tunnel experiments. For example, a fully coupled solution between computational fluid dynamics (CFD) and finite element modelling (FEM) can be developed to model the fluid-structure interaction (FSI). A wind tunnel test relies on the simplified assumption that the scaled aeroelastic model can satisfactorily replicate the dynamic properties of the full-scale design. It neglects the influence of higher modes.

The application of CFD for practical wind engineering problems has received a lot of research attention over the last three decades and has made major progress due to the advancement of computer technology. Thus far, the leading applications for the built environment have concentrated on mean wind speeds for areas including: natural ventilation; pollution dispersion; and human comfort at street and balcony level (Stathopoulos, 1997). It has proven very difficult for CFD to acceptably model the complex flow interference phenomena induced from buildings. Typical features of this unsteady flow regime include turbulent length scales and separation regions larger than the body size of the structure. This is the reason less work has been performed on predicting time-dependent surface pressures on these man-made bluff bodies. CFD has not developed enough to suggest it could replace wind tunnel testing in this respect. It does, however, offer encouraging potential to act as a complimentary tool.

In this paper, the various turbulence models will be discussed with respect to their ability to predict surface pressures and resulting wind loads for a tall building. This includes a detailed review of previous validation studies performed within the literature. It proceeds to highlight the user-defined criteria that must also be satisfied. Finally, the scope for future research on simplifying CFD analyses for tall buildings is discussed, with a view to producing a more efficient and practical solution.

2. CFD Turbulence Models

Basic fluid flow is solved in CFD by subdividing the domain into a continuous mesh of control volumes before applying the equations of mass and momentum conservation. These equations can be supplemented by a variety of different turbulence models, one of which must be used for all FSI problems (Sun *et al*., 2009). At present, no single turbulence model exists that is capable of producing accurate results for all types of problem (Castro, 2003). The selection of the turbulence model has a significant bearing on both the computational cost and accuracy. Therefore, the choice of an appropriate model is normally governed by criterion such as: the accessible computational power; the level of accuracy required; and the amount of time available for simulations.

The most complete form of CFD is Direct Numerical Simulation (DNS). This technique calculates the direct solution of the Navier-Stokes equations for each control volume. Wind turbulence is an inherently random phenomenon and the fluctuations develop in a variety of sizes and frequencies, from large sustained gusts to small eddies. The DNS mesh must be smaller than the smallest turbulent eddy within the flow in order to accurately capture all turbulent effects. Therefore, the computational cost of DNS is extremely inefficient and Knapp (2007) concedes that this particular method should be limited to small scale simulations with low Reynolds numbers.

Reynolds-averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) are two of the leading turbulence models which create their own additional terms in the instantaneous governing equations to achieve 'closure'. Thus, the small scales of turbulence do not require direct modelling and the computational requirement is reduced. RANS equations control the transport of the time (or ensemble) averaged flow quantities, modelling the full spectra of the scales of turbulence. The most popular RANS methods are the two-equation k-ε and k-ω models. This is because they are relatively simple to use, are robust and have a low computational cost (ANSYS Inc, 2005). These empirical models solve two additional transport equations to obtain the turbulent viscosity; the kinetic energy, k, and either the turbulent dissipation rate, ε, or the specific dissipation rate, ω.

LES is a transient solution that adopts a spatial filtering approach: eddies exceeding the mesh size are resolved directly in a time-dependent method whilst those smaller must be resolved by a subgrid-scale (SGS) model. Essentially, the filtering process is a mathematical manipulation of the real Navier-Stokes equations to eliminate eddies smaller than the filter criteria. The idea is to parameterise less of the turbulence but resolve this in a more complete manner. It requires a significantly finer mesh than traditional RANS simulations and requires a large number of time steps to obtain stable statistics of the flow being modelled. Thus, the computational time and required memory are far greater than RANS.

Detached Eddy Simulation (DES) was developed in the field of aeronautics with the intention to combine the useful features of both RANS and LES. In this approach the simpler RANS model is used to simulate the majority of the flow, while LES is employed to the regions of separated flow. The concern with using DES for structural wind engineering applications is that the entire fluid domain could be highly turbulent. This is in contrast to aeronautic analyses involving flow around an aeroplane wing, since this flow is not separated. Hence, it may not be applicable for highly complex flows in densely built-up environments.

3. Development of CFD for Surface Pressures

The majority of benchmark studies to assess the application of CFD for flow features and surface pressures refer to basic cubic shapes, usually exposed to wind acting normal to its face. This is chosen for two main reasons; the simplicity of its shape and the numerous experimental results available for comparison (Stathopoulos, 1997). For example, Rodi (1997) used a surface mounted cube to model the 3D flow for various turbulence models. The results, along with data from a LES workshop, identified that RANS consistently over-predicted the wake reattachment length, while LES gave a good prediction of the overall flow behaviour. It was also found that RANS severely under-predicted the level of turbulent kinetic energy, k – sometimes by as much as 66% lower than the experimental data. The solution time for the LES simulation was 160 hours, where as the RANS solutions ranged between 15 minutes and 6 hours.

The CWE 2000 competition challenged applicants to model the flow around a 6m cube using an appropriate turbulence model, with the results compared against full-scale data. Two angles of attack were considered; 0° and 45°. It concentrated on the performance of various two-equation RANS models, with results suggesting that the *RNG* k-ε solution achieved the closest agreement with the full-scale results. Of the other turbulence models, the industry standard k-ε was found to under-predict the flow separation, while the *MMK* model over-predicted this criterion.

Knapp (2007) reviewed the recent advances in turbulence modelling to predict wind speeds and surface pressures for a simple cube. This was extended to the complex grandstand at the Ascott horse racing track to the west of London in Berkshire, UK. A full set of data for each direction was obtained from a scaled wind tunnel analysis of the grandstand. Thus, to allow direct comparison with these wind tunnel results, the CFD simulations were run at model scale (1:300). The procedure comprised of both steady and unsteady RANS as well as DES. The steady RANS solution did not achieve convergence – as was to be expected from the irregular geometry of the problem. The DES results displayed a notable improvement in the prediction of mean loads and showed slightly more complex unsteady behaviour. However, the conclusions stated the slight superiority of DES was not worth the huge increase in computational times.

Recent publications show LES has produced a very strong correlation to field measurement data for complex building forms and environments. For example, research by Nozu *et al*. (2008) successfully implemented a LES model, assisted by graphical information system (GIS) data, to obtain the flow field and building pressure distribution for two buildings in the centre of Tokyo, with many surrounding tall buildings. The results found LES to be capable of reproducing complex flow patterns and could qualitatively calculate distorted pressure distributions in an urban environment. It did, however, highlight the difficulty in generating an optimum mesh for such cases since the flow field around the buildings was not predictable in advance. There is no mention of the computational time, which would have been substantial despite the extensive computational power available. These requirements exceed the capabilities of the majority of industry practitioners, a situation which is likely to continue for several years. The Architectural Institute of Japan (AIJ) has published more definitive recommendations on the use of CFD for wind loads on buildings (Tamura *et al*., 2008). This research assessed the success of both LES and a modified RANS in calculating the surface pressures of a low-rise (1:1:0.5) and high-rise (1:1:4) building. The outputs were compared against experimental data from wind tunnel tests. The conclusions stated that LES was essential for full wind load estimation, while RANS should be limited to calculating mean wind forces on buildings. The results lead to the development of a flow chart (Fig. 1) to outline the procedure for predicting wind loads on structural frames using CFD. Reference to Fig. 1 emphasises that the RANS model is limited to alongwind loads and should be supplemented with an appropriate AIJ-published gust effect factor. Thus, the benefits of RANS for tall building problems are restricted since it is generally the acrosswind and torsional loads that are critical.

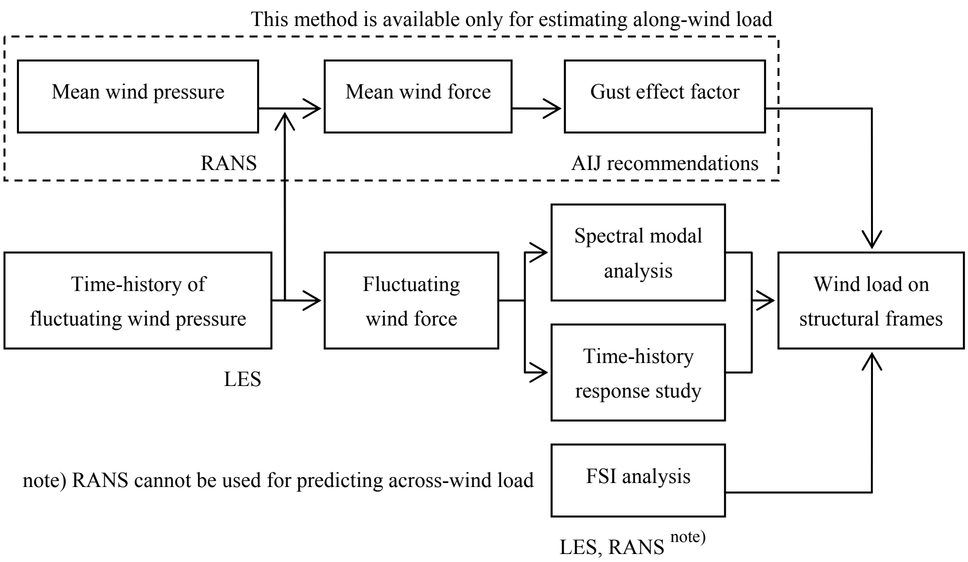

Fig. 1. CFD procedure for estimating wind loads on structural frames (Tamura *et al.*, 2008).

Sun *et al*. (2009) developed a fully coupled FSI problem to consider an appropriate turbulence model in the field of bridge deck aero-elasticity. A similar interface could prove very useful for the aeroelastic response of flexible tall buildings. Simulations were performed in both 2D and 3D for a rectangular cylinder ($B/D = 4$). Comparisons between the force coefficients and Strouhal numbers indicated that the 2D and 3D results were identical. This is simply because most RANS models assume isotropic turbulence so the 3D simulation lost the 3D vortical structures in the spanwise direction. Furthermore, they concluded RANS models are significantly more efficient than LES and, more specifically, the k-ω model gives superior performance than k-ε for near-wall flow simulation. It regards LES as being too advanced for general FSI studies.

4. User-Defined Criteria

Aside from concerns related to the choice of turbulence model, it has been well documented that significant errors are possible from the user-defined criteria. This includes but is not limited to the boundary conditions and grid specification (Miles and Westbury 2003). In addition, the fluid domain must be such that it captures all the important flow characteristics and does not constrain the flow.

The boundary conditions must be able to produce a fully developed atmospheric boundary layer. This profile is dependent on inputs including the turbulence characteristics, roughness length and inlet velocity. The target profile, derived from experiment or a form of the logarithmic law, should be satisfied upstream of the structure prior to any local velocity or pressure predictions being performed.

The size of the mesh is required to be fine enough to resolve the local flow component around the bluff body. Thus, the minimum mesh size should be implemented in the surrounding vicinity of the structure to a downstream distance equal to the end of the recirculation zone (Knapp 2007). Grid refinement, either manual or automatic, should be performed for RANS simulations to ensure a fully grid-independent solution. Also, the mesh size has a significant bearing on achieving convergence – too coarse a meshing scheme would be unable to completely resolve the flow instability.

The time-step duration is another important parameter, although there is no definitive guide for their size. This function is more significant for LES than RANS, since the time-step contributes to its explicit filtering process of the turbulence. Thus, the value may require modification in order to achieve satisfactory convergence. Too large a time-step may not capture the fluctuating instability of inherently turbulent flows. Analyses towards predicting structural loads should use a time-step lower than the typical response time of the structure under consideration.

5. Potential Simplifications for Tall Buildings

For a tall building, a measurable inaccuracy may be accepted in a CFD simulation of the wind velocities and pressures. This depends on the criterion being assessed and the ability to account for the error. The critical design concern is resisting the pressures throughout the height of the structure, while the serviceability limit states must be satisfied for the transient dynamic interaction with the building. Thus, these criteria must be satisfied to prevent loss of life or substantial financial loss.

The effects of neighbouring buildings are less important for tall buildings located in an environment of significantly smaller structures. In such cases, flow instability caused by the building itself is of primary importance. Therefore, a possible simplification for such instances may be to investigate the impact of ignoring these smaller buildings in the CFD analyses. If a conclusive agreement could be validated then it would dramatically reduce the computational costs of both RANS and LES simulations.

Tall building developments are quite often designed with a consistent plan spanning its height. For such cases, reducing the CFD simulation from a full 3D analysis to a simplified 2D model would also dramatically reduce the computational costs. However, flow around buildings is quite obviously a three-dimensional phenomenon. Tall buildings can cause high-level wind to deflect downwards (Fig. 2) to produce strong bursts of pressure. Similarly, upward sweeps of sudden accelerated flow can occur at the tip. Turbulence is another phenomenon that demonstrates full 3D characteristics, with a high level of interaction between turbulent eddies in all three axes. Thus, a 2D plan section of a tall building would not account for this behaviour.

A simple 3D case study should be compared with a layered 2D model to assess the impact of this assumption. The structure could be subdivided into a series of idealised 2D layers at specific heights. The corresponding wind speeds would then be obtained directly from the atmospheric boundary layer profile to obtain the mean surface pressures. As well as the restrictions outlined above, these 2D models would give no indication of the differing phase shifts of the time-dependent surface pressures throughout the height. Collectively, the limitations suggest that the most advantageous outcome would be for predicting the alongwind mean loads only. Loads on the roof would have to be calculated using code prescribed methods.

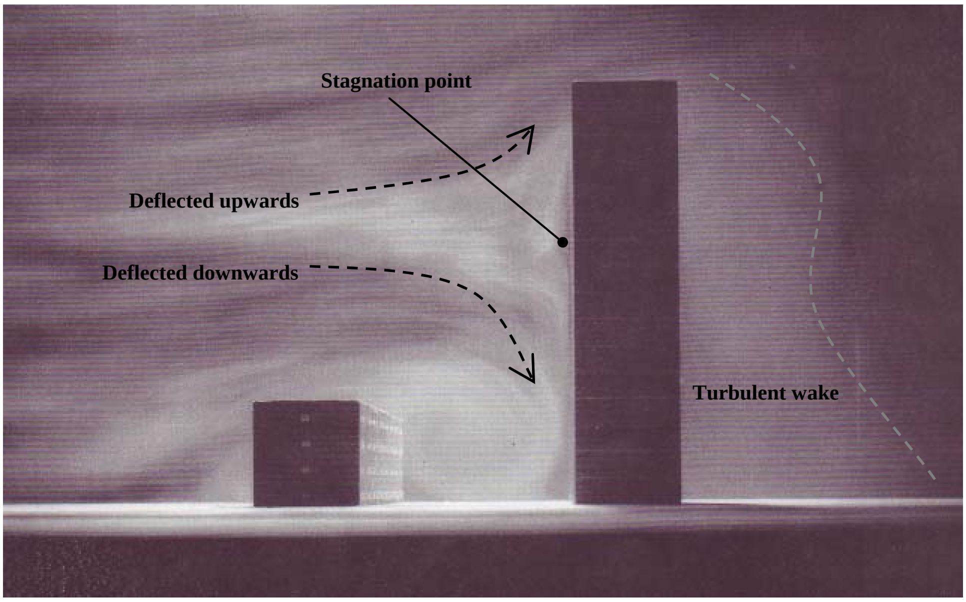

Fig. 2. Wind deflected by a tall building. Wind flowing left to right. (Building Research Establishment, 1994)

If the results between the 2D and 3D analyses differ significantly, a successful conclusion may yield the development of a factoring system to be applied to the predicted pressures at each height. These factors would vary based on site conditions and would have to ensure the resulting loads were always conservative. This is a similar method to that offered by current standards but is valuable because it could incorporate a much broader spectrum of geometric forms. The selected turbulence model must be capable of acceptably modelling the flow instabilities – particularly for irregular plan sections. The literature suggests an unsteady modified RANS model (*RNG k-ε*) or the more advanced LES would offer the optimum predictions. However, this methodology may be more useful for the loading of smaller, stiffer structures that are less sensitive to unsteady wind flow.

These proposed simplifications are still fundamental concepts and would require complete investigation to assess their possible application. Successfully validating either or both of these proposals would reduce computational costs and improve the practicality of CFD for predicting wind loads on tall buildings. These methods could be implemented as a predictive tool at an early design stage to assess a broader range of design alternatives at lower costs. Wind tunnel testing could be used later to finalise the decisions and provide time-dependent pressures for the detailed structural analysis.

6. Conclusions

The aim of this paper was to research the development of CFD in its application to calculating wind generated surface pressures on tall buildings. It is quite apparent that CFD has huge potential and its development should remain of great interest to structural wind engineers. Despite its progress, it is not in a position to replace wind tunnel testing for calculating complete wind loads. It does, however, offer encouraging evidence that it can be used as a complimentary analytical tool. A particular benefit is its potential to develop a full FSI model, as is currently performed for bridge deck studies. The structural model would be significantly more advanced than the aeroelastic models used in wind tunnel testing, which could account for the influence of higher natural modes and nonlinear mode shapes.

The research shows that full LES demands a high computational cost and its implementation will exceed the capabilities of many industrial practices for several years. Consequently, while it appears LES may offer the closest computational alternative to traditional wind tunnel testing, RANS is certainly the most practical for typical structural engineering purposes. By limiting RANS applications to mean alongwind loads, as prescribed by AIJ, it is difficult to assess its value for tall building problems since it is the acrosswind and torsional loadings that are often critical. Nonetheless, the literature does clearly suggest that a statistically improved version of *k-ε* or the standard *k-ω* model will produce the optimum RANS results for predicting flow features and surface pressure distributions.

The majority of the user-defined criteria, particularly related to grid specification, are iterative processes that require alteration based on the flow instabilities found. This can prove a very time consuming process. This feature must be improved in order to progress CFD towards the standard set by wind tunnel testing.

Accepting certain limitations of CFD could allow notable progress to be made towards the practical application of CFD for surface pressures on tall buildings. The paper has suggested some potential concepts but stressed the importance of ensuring these simplified pressures are validated and always on the conservative (safe) side.

References

Allsop, A. BS EN 1991-1-4 Tall Buildings. In: ICE (Institution of Civil Engineers), *New Eurocode on Wind Loading*. (London, UK, 11 May 2009).

ANSYS Inc. *FLUENT 12.0 Theory Guide*. (2009)

Building Research Establishment. *Wind around tall buildings*, BRE Digest 390, (BRE: Watford, UK, 1994).

British Standards Institution. *BS 6399-2:1997 Loadings for buildings – Part 2: Code of practice for wind loads*. (BSI: London, UK, 1997).

British Standards Institution. *BS EN 1991-1-4:2005 Eurocode 1: Actions on structures – Part 1-4: General actions – Wind actions*. (BSI: London, UK, 2005).

Castro, I.P. CFD for External Aerodynamics in the Built Environment. *The QNET-CFD Network Newsletter*. Vol. 2: No. 2 – July 2003, pp 4-7 (2003).

Davenport, A.G. What makes a structure wind sensitive? in *Proc. The Jubileum Conference on Wind Effects on Buildings and Structures*. Porto Alegre, Brazil 25-29 May 1998, eds. A. A. Balkema, (Rotterdam, Netherlands, 1998) pp 1-14.

Irwin, P.A. and Baker, W.R. The Burj Dubai Tower Wind Engineering. *Structure Magazine*, June 2006 pp 28-31 (2006).

Knapp, G.A. *Improved methods for structural wind engineering*. Ph. D. University of Nottingham (2007).

Mehta, K.C. Wind load standards. in *Proc. The Jubileum Conference on Wind Effects on Buildings and Structures*. Porto Alegre, Brazil 25-29 May 1998, eds. A. A. Balkema, (Rotterdam, Netherlands, 1998) pp 307-314.

Miles, S. and Westbury, P. Practical tools for wind engineering in the built environment, *QNET-CFD Network Newsletter*, Vol. 2: No. 2 – July 2003, pp11-14 (2003).

Nozu, T., Tamura, T., Okuda, Y., and Sanada, S. LES of the flow and building wall pressures in the center of Tokyo. *Journal of Wind Engineering and Industrial Aerodynamics*, Elsevier, **96** pp 1762-1773 (2008).

Rodi, W. Comparison of LES and RANS calculations of the flow around bluff bodies. *Journal of Wind Engineering and Industrial Aerodynamics*, Elsevier, **69-71** pp 55-75 (1997).

Stathopoulus, T. Computational wind engineering: Past achievements and future challenges. *Journal of Wind Engineering and Industrial Aerodynamics*, Elsevier, **67&68** pp 509-532 (1997).

Sun, D., Owen, J.S., and Wright, N.G. Application of the k-ω turbulence model for a wind-induced vibration study of 2D bluff bodies, *Journal of Wind Engineering and Industrial Aerodynamics*. Elsevier, **97** pp 77-87 (2009).

Tamura, T., Nozawa, K., and Kondo, K. AIJ guide for numerical prediction of wind loads on buildings, *Journal of Wind Engineering and Industrial Aerodynamics*, Elsevier, **96** pp 1974-1984 (2008).

Taranath, B.S. *Steel, Concrete, & Composite Design of Tall Buildings*, 2^{nd} Edition (New York: McGraw-Hill, 1998).