Wind loading on high-rise buildings and the comfort effects on the occupants

Ramtin Avini, Prashant Kumar*, Susan J. Hughes

^aGlobal Centre for Clean Air Research (GCARE), Department of Civil and Environmental Engineering, Faculty of Engineering and Physical Sciences, University of Surrey, Guildford GU2 7XH, United Kingdom

Abstract

The design of low to medium-rise buildings is based on quasi-static analysis of wind loading. Such procedures do not fully address issues such as interference from other structures, wind directionality, across-wind response and dynamic effects including acceleration, structural stiffness and damping which influence comfort criteria of the occupants. This paper studies wind loads on a prototype, rectangular cross-section building, 80m high. Computational Wind Tunnel (CWT) tests were performed using Autodesk Flow Design with the buildings located in London and New York City. The analysis included tests with and without the surrounding structures and manual computation of wind loads provided data for comparison. Comfort criteria (human response to building motion) were assessed from wind-induced horizontal peak accelerations on the top floor. As expected, analytical methods proved conservative, with wind pressures significantly larger than those from the CWT tests. Surrounding structures reduced the mean component of the wind action. As for comfort criteria, across-wind direction governed the horizontal accelerations with wind targeted on the building's narrow face. CWT tests provide a cheaper alternative to experimental wind tunnel tests and can be used as preliminary design tools to aid civil engineers, architects and designers with high-rise developments in urban environments.

*Corresponding author. Address as above. Email: p.kumar@surrey.ac.uk; Prashant.Kumar@cantab.net

Keywords: Wind Loading; Tall Buildings; Design Standards; Wind Tunnel; Comfort Criteria

Research highlights

- Wind loads for tall buildings studied by codes and Computational Wind Tunnel (CWT)
- Design Standards gave rise to larger surface pressures than CWT estimates
- Complex terrain led to more fragmented vortices in New York City
- Shielding effect was crucial for depleting the mean component of the load
- Building's low damping level resulted in surpassing comfort limits in some cases

1. Introduction

Wind is a phenomenon of significant complexity due to the generation of many flow situations stemming from its interaction with structures. It comprises a large number of eddies of different sizes and rotational characteristics flowing in a general stream of air acting with respect to the surface of the earth; these eddies are the main reason behind the stormy and turbulent nature of wind. The leading rationale behind the gustiness of strong winds in the lower atmosphere emerges from its contact with the surface features. The mean wind speed, over a duration of ten minutes or greater, has a linear relationship with increasing height in contrast with turbulence which tends to reduce with height (Mendis et al., 2007).

The wind vector at a particular point may be defined as the addition of two segments: (i) the static component, the mean wind vector, and (ii) the dynamic component, the turbulence. The dynamic loading on a structure is largely dependent on the size of eddies due to the turbulence. Eddies that are large in size with dimensions proportionate to the structure, will lead to well-coordinated pressures as they surround the structure. Conversely, small-scale eddies bring about pressures on diverse sections of a structure, making them uncoordinated due to the distance of separation (Mendis et al., 2007). The effect of wind loads can result in a dynamic response of some structures, especially those which are tall or slender. Some factors which contribute to this action are buffeting, vortex shedding, galloping, and flutter.

Slender structures such as tall buildings are known to be susceptible to the dynamic response due to turbulence buffeting. Vortex shedding or galloping are associated with transverse or crosswind responses whereas a combined motion of bending and torsion (which causes instability) is linked with flutter (Mendis et al., 2007). In the present study with reference to buildings, flutter and galloping are not of concern.

High-rise buildings subject to wind actions are normally regarded as ‘vibrant bluff bodies’ with numerous plan dimensions that oscillate in the along-wind, across-wind, and torsional directions (Lin et al., 2005; Holmes, 2007). Bluff bodies are represented by large regions of separated flow, significant drag forces, and the formation of vortex shedding, in contrast with streamlined bodies, where the flow streamlines track the body (Roshko, 1993). The along-wind load is the resultant pressure fluctuation imposed parallel to the wind as opposed to the across-wind load, which is the response of the building perpendicular to the wind direction (Kim, 2013). The twisting motion of a bluff body occurs when the pressure distribution around the wall faces, is non-uniform. This loading system has been widely investigated with the aid of aerodynamic load measurements in wind tunnel tests carried out on bluff bodies with contrasting shapes, the presence of other interfering bodies, and various angles of approaching flow (Boggs et al., 2000).

The aerodynamics of tall buildings are complex and affected by a variety of key parameters; the most crucial of which are the approaching boundary layer flow, the building geometry, and the surroundings of the building (Kim, 2013). The boundary layer flow approaching a high-rise building is best described with the aid of average wind speed and turbulence intensity profiles in terms of height. These control the corresponding magnitudes of the mean and root mean square (RMS) values of the aerodynamic forces on the building. The RMS value of the oscillating wind speed over the corresponding mean wind speed defines the turbulence intensity.

The boundary layer flow is directly reliant on the terrain roughness, represented by the roughness length. Flatter and smoother terrain conditions lead to an increase of mean wind speed at a given height with a corresponding decrease in turbulence intensity. Hence, with the utilisation of the quasi-steady theory, a rise in the roughness length reduces the mean component of load but increases the oscillating component (Kim, 2013).

Vickery (1968) and Sakamoto (1985) investigated bluff bodies in turbulent flows, under fluctuating loads. All the bluff bodies were rectangular with a range of plan dimensions and heights. It was found that the flow patterns surrounding the bodies and correspondingly the drag and lift forces on the bodies were influenced by the various geometries. Their results indicated a decrease in mean drag force with the reduction of B/D or $2H/B$; where B and D were the plan dimensions perpendicular and parallel to the flow direction, respectively, and H was the height of bluff body. The mean drag force coefficients increased with surface area of the bluff body (Vickery, 1968).

High-rise buildings are usually situated in metropolitan and developed regions, surrounded by buildings of similar dimensions. The impacts of the neighbouring buildings on the aerodynamics of a tall building are recognised as the interference and shielding effects (Khanduri, 1998). These effects are governed by several factors such as shape and dimensions; the gap between the interfering building and the target structure; the orientation of both surrounding and downstream buildings; lateral disposition of the interfering building with respect to the target building and wind flow direction; the number of surrounding buildings; wind attack angle, and terrain roughness (Bailey, 1985; Kareem 1987; Taniike, 1991; Khanduri, 1998; Gu, 2005; Xie, 2009). The effects of interference result in the increase of the fluctuating load but depletion of the mean load, where this reduction of the wind load initiated by the surrounding buildings is known as the “shielding effect” (English, 1990; English, 1993; Khanduri, 1998). There are limitations on the number of studies undertaken on

shielding and interference effects of grouped buildings, due to the complexity of multiple building layouts and sizes. However, there are studies cited on buildings of equal sizes, for example, Gu et al. (2005) and Xie and Gu (2009) examined the along-wind and across-wind interference effects of two and three high-rise building arrangements, respectively. Lam et al. (2008) investigated the interference effects of tall buildings positioned in a row. Lam and Zhao (2008) explored the interference effects of tall buildings placed in L- and T-shapes. Stone (1987) studied the interference induced by different orientations of a group of equal-sized buildings. Finally, Soliman (1976) and Hussain and Lee (1980) carried out inspections regarding the interference effect instigated by a large group of buildings. Conclusions from the studies described above determined that a tall building overwhelmed by a large group of neighbouring buildings is exposed to a smaller wind loading than the same building in an isolated or stand-alone environment condition.

The end result of this work can help to optimise and enhance the design of high-rise buildings by using the CWT technique. This technique can provide a more sustainable and quicker approach for determining wind loads than that of the standards. Furthermore, by controlling the criteria for the comfort of the occupants due to the wind-induced horizontal accelerations, any potential discernible or displeasing vibrations at the upper levels of the buildings can be avoided. By carefully undertaking CWT studies, smarter cities and more resilient environments can be formed and important urban design/planning regulations can be realised.

The recent years have seen the rapid rise of “smart sustainable cities” topic as a positive reply to the challenge of urban sustainability (Bibri and Krogstie, 2017). For example, Broekhuizen (2016) investigated how urban planners can better integrate wind simulation in the design process with the aid of three CFD programs such as Autodesk CFD, Autodesk Flow Design, and DesignBuilder. Due to the lack of knowhow regarding technical side of CFD simulations, there is generally a lack of motivation among urban planners for using such techniques. This

paper demonstrates a tool (Autodesk Flow Design) which does not require comprehensive recognition of CFD simulations to employ. Stern et al. (2001) carried out extensive, pragmatic approach to V&V (verification and validation) methodology and procedures for determining errors and uncertainties for industrial CFD. Reiter (2008) performed a pedestrian comfort study using CFD simulations (ANSYS Fluent). The validation process was done through a comparison between the simulated results and wind tunnel investigations for three different configurations; a lone-standing building, the interaction between two buildings, and an urban condition. Chakravarthy and Akdag (2015) examined issues concerning meshing, which has a significant effect on the accuracy and convergence of numerical simulations, since computational simulations of fluid dynamics phenomena is solved by taking into account different constituents of the simulation, i.e. the computational model (including the mesh). Maragkogiannis et al. (2014) undertook a CFD approach for the study of urban thermal environment (a public square), which considered the key parameters that form the square's microclimate; radiation, turbulence models for the study of the flows, and a well-defined computational grid.

The aim of this paper is to advance the current techniques of forecasting design wind loads on high-rise buildings. This was achieved through a comparative study in which wind loads were predicted using a numerical wind tunnel model simulator and compared with those manually calculated using design codes. The wind-induced responses obtained from the design codes were used to both validate the simulator methodology and assess the accuracy of both methods. In order to make the two techniques comparable, the same high-rise building of known dimension and structural design was used in both methods. The simulated wind tunnel testing was performed for the target structure with and without the presence of surrounding structures. When surrounding structures were included, two different scenarios were simulated, namely a London and New York City landscape. These provided two different

topographies with distinct preliminary design wind loads. It was anticipated that these authentic configurations would give rise to more realistic results compared with those obtained from an isolated tall building modelled remote from its surroundings.

2. Methodology

Two methods of analysis were adopted: (i) the use of two design standards (namely EN1991.1.4:2005 and ASCE 7-10), and (ii) implementation of a Computer-generated Wind Tunnel (CWT) simulator using Autodesk Flow Design configured for both London and New York City locations. Autodesk Flow Design runs a transient, incompressible flow solver that uses a finite volume method. Turbulence is solved using a Smagorinsky Large Eddy Simulation (LES) model similar to other CFD models. However, Flow Design was developed to be extremely geometry tolerant and does not require geometry “cleanup” (in terms of meshing) like other products (such as ANSYS Fluent). Its meshing technology is designed to accept geometry from the most widely used design packages. It accommodates both surface and solid 3D models and is not sensitive to small geometry imperfections (Autodesk, 2014).

In order to perform the analysis, a high-rise building of known dimension and structural system was modelled. For the numerical simulation, the building was incorporated in the CWT model; firstly, within a London landscape and secondly within New York City. After the completion of both CWT simulations, the results were compared with the calculations carried out using EN1991.1.4:2005 (Eurocode Standards adopted by UK) and ASCE 7-10 (American Standards) for London and New York City, respectively.

2.1 Validation of the Computational Fluid Dynamics (CFD) method for turbulent wind flows

CFD applied to wind engineering, is a numerical technique used for the simulation, prediction or quantification of fluid flow phenomena in which the relevant equations of

motion are solved at a distinct set of points (Tamura and Kareem, 2013). The last two decades have seen extensive use of CFD in wind engineering, often termed as Computational Wind Engineering (CWE). Despite its high usage, there has been a shortage of assurance over the interpretation of this method for quantitative and qualitative predictions. The major challenges associated with the practical use of CWE are the lack of precise data, for which the user has to input many physical and numerical parameters (Franke et al., 2004). Hence, the CFD models require rigorous validation before they can be used with confidence. Validation is typically achieved by comparing CFD outputs with those of physical models, so that a comprehensive picture of both the physical aspects and numerical parameters can be made available to the user for adoption in the CFD model. This way the flow can be simulated more accurately and the reliability of this technique can be acknowledged.

In this study, the modelled CFD results were compared with corresponding experimental results for a two-dimensional square prism. Drag coefficients (C_d) were calculated for a range of Reynolds numbers (Re) and both techniques were compared in order to indicate model performance compared with experimental values. Furthermore, working in terms of C_d also allows the model numerical simulation results to be compared with the empirical formulas adopted by the design specifications.

The literature behind the modelling of drag forces of two-dimensional bodies immersed in a turbulent wind flow are outlined below:

2.1.1 The Bernoulli Equation

The Bernoulli equation for viscous-free, steady flow along a streamline is shown in Equation (1) in which the first and second terms in the equation represent the static and dynamic pressures, respectively (Simiu and Scanlan, 1996):

$$p + \frac{1}{2}\rho V^2 = \text{constant} \quad (1)$$

196 where

197 V = velocity, in m/s

198 p = static pressure, in Pa

199 ρ = fluid density, in kg/m³

200 **2.1.2 Reynolds number, Re**

201 The Re is important to obtain the extent to which boundary layers are laminar or
202 turbulent. This determines their capability to resist pressure gradients, or their sensitivity to
203 separate from the surface of a body. Re is defined as the ratio of inertia forces to viscous
204 forces acting on the fluid particles (Steerter and Wylie, 1997):

$$205 \quad Re = \frac{\text{Inertia force}}{\text{Viscous force}} = \frac{\text{mass} \times \text{acceleration}}{\text{shear stress} \times \text{area}} = \frac{Vl\rho}{\mu} = \frac{Vl}{\nu} \quad (2)$$

206 where

207 μ = dynamic air viscosity, in Pa s

208 ρ = air density, in kg/m³

209 ν = kinematic viscosity, in m²/s

210 l = characteristic length, in m

211 In wind engineering, when viscous forces are counted, the Re becomes important. Based on
212 (Simiu, 2011), for air flow at 21°C under atmospheric pressure conditions, the kinematic
213 viscosity (ν) is approximately 15×10^{-6} m²/s, thus the Re is:

$$214 \quad Re = 67000Vl \quad (3)$$

215 Where V and l are expressed in terms of m/s and m, respectively. The latter is the travelled
216 length of the fluid (leading edge of the boundary layer as it approaches the prism).

217 **2.1.3 Drag coefficient, C_d**

218 The drag is termed as the force component that acts parallel to the corresponding
219 approach velocity imposed on the body by the moving fluid. The drag coefficient (C_d) is a

non-dimensional coefficient, which relates the resultant drag force to the mean dynamic pressure ($\frac{1}{2}\rho V^2$), of the upstream flow (Shames, 1982):

$$C_d = \frac{\text{Drag force}}{\frac{1}{2}\rho V^2 A} \quad (4)$$

Where A is the area of the structure perpendicular to flow (m^2). Generally, C_d is a function of the geometry and roughness of the submerged body. C_d is also dependent on Re in laminar and transition flows and is predicted either through wind tunnel experiments or numerical CFD techniques (Steerter and Wylie, 1997; Shames, 1982).

2.1.4 Framework of CFD simulation

2.1.4.1 Boundary conditions

Unlike wind tunnel simulations where boundary layers are covered in a physical manner, artificial boundary conditions should be applied in the CFD model due to the finite nature of the computational domain. Improper boundary layers produce unreliable results which can cause failure in the numerical process (Tamura and Kareem, 2013). In order to counter this, the simulation volume should be chosen to be the largest possible (Reiter, 2008).

2.1.4.2 Turbulence model

In civil engineering applications, the incompressible turbulent wind flow is known as the most frequent phenomenon (Simiu and Scanlan, 1996). The ever-turbulent nature of the upstream flow to the structure is because the structure is immersed in a turbulent boundary layer as well as the flow having a high Re. Phenomena concerned with the flow, such as vortex formation, interaction between upcoming turbulent flow and separated shear layer, and dynamic wake development define the turbulent conditions (Tamura and Kareem, 2013). Two types of turbulence models, which are adopted to dissipate the turbulent energy at high frequency through turbulence viscosity, are used.

1. Reynolds Averaged Navier-Stokes (RANS) equations: These determine the turbulence effect using supplementary equations of turbulence statistics. They are mostly effective if the mean values of the fluid flow are of interest (Tamura and Kareem, 2013). However, the main drawback of the RANS equations are their inadequacy for $Re \geq 20,000$ and most practical applications of turbulent fluid flow are $>10^6$.
2. Large Eddy Simulation (LES): This solves the large scale units of the fluid flow directly with the modelled small scale components. LES carries out 3D calculations but requires long times series data for the flow quantities in order to conduct a dependable analysis. However, it determines the flow quantities at all the grid points and evaluates both mean and peak values. Therefore, in wind engineering, LES is more applicable and therefore the preferred choice to RANS (Tamura and Kareem, 2013).

It should also be noted that the use of the LES turbulence model is highly influenced by numerical mesh number, spatial resolution and time scale. In Autodesk Flow Design, the model wind tunnel is transparently divided into a grid of small pieces named “voxels”. For the calculations, the software transparently performs this operation by automatically determining the voxel sizes based on the model under investigation and the wind tunnel. The resolution of the analysis grid, which controls the size of the voxels, was set to 100% from a possible range of 50%-400%, where the higher value indicates a more detailed but a time-consuming simulation. For this 2D analysis, higher resolutions didn’t produce more detailed results.

In addition, prior to using the LES model in the software, the optimal computational grid size was determined to ensure that the results were grid independent. A suite of tests was carried out to ascertain the optimal grid size, with the mesh size being halved between each run. The optimal grid size was selected when a further halving of the grid spacing had minimal effects

on the results. In this software, time scale is obtained automatically when the model is imported and the wind tunnel is configured.

2.1.5 Validation procedure

Wind tunnel tests can be used for the validation of CFD simulations, since they produce more accurate results (for the same configuration), as the measurements are taken on a “real site” with static boundary conditions. In order to validate the Autodesk Flow Design software as a tool for simulating wind around buildings, a simulation was carried out for a two-dimensional square prism and the results compared with those from equivalent wind tunnel tests available in the literature.

2.1.5.1 Empirical results

The experimental results, from the literature cited, are listed in Table 1. It can be seen that for a square prism, the drag coefficient (C_d) varies across the range of Re numbers presented. C_d was between 2.0- 2.05 for an attack angle of 0° ($Re\ 10^4$ - 10^5) but slightly lower at ~ 1.6 for an attack angle of 45° ($Re\ 10^4$ - 10^5).

2.1.5.2 Extracted results from Autodesk Flow Design

The CFD results were obtained using Autodesk Flow Design. Two-dimensional flow was modelled around a square (1x1m) prism for a range of wind speeds and 2 attack angles (0 and 45°). The wind speeds were incremented from 0-95m/s in 5m/s increments so that all the domains of Re number and turbulence effects in the critical region were covered. The results using the Autodesk Flow Design model are presented in Figures 1-3, showing agreement with the experimental data and hence the model is validated (albeit for a square prism) but that the structures in the study here have a similar geometry. Hence the model can be used.

2.2 Site 1: London, United Kingdom

2.2.1 Design code (EN1991.1.4:2005)

EN1991.1.4:2005 design codes were used to determine the wind-induced pressures acting on each face of the building. The procedure initially required the fundamental wind velocity ($V_{b,0}$) corresponding to the site. This was extracted by locating the velocity corresponding to London on the map produced by UK National Annex ($V_{b,map}$) and multiplying it by the relevant altitude correction factor (C_{alt}). The basic wind velocity, V_b was then obtained using $V_{b,0}$ and the values for the season and direction factors (conservatively taken as 1). This value was then multiplied by the conforming roughness and orography factors to compute the mean wind velocity. High-rise buildings comprise two parts (i) for windward walls, when $b < h \leq 2b$ where h and b represent the height and width of the building (Supplementary Information, SI Figure S1), and (ii) for leeward and sidewalls where the reference height should be taken as the height of the building. The method was then followed by finding the peak velocity pressures for both the reference heights and determining the pressure on each face of the building with the correct use of external pressure coefficients (SI Figure S2a).

2.2.2 Computational Wind Tunnel (CWT)

The target building was regarded as having administrative usage. This meant that office blocks were incorporated into the structure. The building's structure was modelled in FEA software ETABS as shown in Figure 4a. The building was 80 m high with a rectangular plan cross-section, 41 m $\times$ 26 m (Figure 4b) in order to investigate different scenarios with varying wind directions. This would produce different results to a building whose plan dimensions were equal.

Figure 5 shows the site of the proposed building. It was located in Canary Wharf, London which comprises buildings that are generally tall (average height, ~200 m). Consequently, there was a considerable height difference between the target building and the surrounding buildings. This height difference was chosen to provide shielding against the wind in some wind directions, while the congestion of the site would promote the occurrence of channelling effects. Therefore, this site configuration would allow the emergence of aerodynamics effects from the surrounding buildings that would provide useful insights, particularly when comparing the CWT method with the design code method.

Wind loads can be significantly different when comparing different topographies since topographic effects on the flow of the wind near the ground surface are important. Nonetheless, the majority of standards only address the most adverse situations when wind speeds approach the crest of hills and ridges (Maharani, Lee and Lee, 2009). In order to determine the most appropriate wind speed at the building site, Computer-Aided Design (CAD) Mapper was used to establish the exact topography of the site. This was then used in the CWT model so that the wind speed could be adjusted accordingly.

In order to accurately determine the design wind loads in the CWT, the presence of the interfering structures was fully incorporated. CAD Mapper, which is the primary tool for producing CAD models, was implemented to produce the model shown in Figure 6a in conjunction with Autodesk Revit. After the creation of the downstream building and the surroundings using Autodesk Revit and CAD Mapper, the CWT was set up with the aid of Autodesk Flow Design as displayed in Figure 6b.

2.3 Site 2: New York City, United States of America

2.3.1 Design code (ASCE 7-10)

Design codes from ASCE 7-10 provided the procedures for obtaining surface pressures when the location was New York City. The initial step was to determine the basic wind speed using the map of United States corresponding to the relevant occupancy category. The velocity pressure at height z was then computed, using the basic wind speed, the wind directionality factor, K_d , the exposure coefficient, K_z and the topographic factor, K_{zt} . The wind pressures were obtained for windward, leeward, and sidewalls using the relevant external pressure coefficient, C_p and taking into account the gust effect factor. Finally, the internal pressures were subtracted from the external ones by incorporating an internal pressure coefficient (positive and negative values of GC_{pi}) as recommended by ASCE 7-10 (SI Figure S2b).

2.3.2 Computational Wind Tunnel (CWT)

Figures 7 shows the aerial view of the proposed building in New York City. Similar to the London location, it comprises buildings which typically exceed 80 m height thus providing a significant height difference between the target building and the surrounding buildings. Again this was deliberately chosen to maximise shielding and channelling effects. Figure 8 shows the computer model and the CWT for the New York City simulation. CAD Mapper was used to produce the model shown in Figure 8a in conjunction with Autodesk Revit and the CWT model was set-up using Autodesk Flow Design as shown in Figure 8b.

2.4 Comfort Criteria

For high-rise structures, it is important to determine whether the horizontal accelerations at the top occupied floors of the buildings, due to the building movement under (turbulent) wind action, are displeasing and/or unacceptable for the occupants. Acceleration levels, which are acceptable to people, are reliant on many physiological factors and consequently are subjective to some degree.

Consideration must also be made regarding the planned usage of the building: residential buildings demand more restrictive acceleration criteria compared with office (administrative) buildings. Based on previous research, when the amplitude of acceleration is in the range of 0.5%-1.5% of acceleration due to gravity (9.81 m/s^2), movement of buildings becomes *perceptible* to the majority of building occupants. According to this, an acceleration limit of 1%-3% of gravity is recommended, where the lower value is considered suitable for *residential* buildings, while the upper value is appropriate for *office* buildings (Taranath, 2010).

2.4.1 Wind-induced horizontal peak accelerations

The most decisive concept for complying with the occupant's comfort is the peak acceleration they are likely to experience. The maximum lateral deflection normally occurs in a direction parallel to wind (along-wind direction), however, the maximum acceleration giving rise to human perception of motion or discomfort may be induced in a direction perpendicular to the wind (across-wind direction). Therefore, it is important to determine the maximum accelerations in both the *along-wind* and *across-wind* directions. Both EN1991.1.4:2005 and ASCE 7-10 provide an approach for estimating along-wind responses, including peak acceleration, but neither specify a procedure for obtaining an across-wind response. The National Building Code of Canada (NBCC 2005) does consider such a procedure (Taranath, 2010), thus, NBCC 2005 procedures have been adopted to determine these accelerations as described in Section 2.4.2.

2.4.2 Estimation of accelerations

NBCC 2005 gives the following expressions, Eqs. (5) and (6), for determining the across- and along-wind accelerations, based on wind tunnel studies:

The across-wind acceleration (a_w) is given by:

$$a_W = n_W^2 g_p \sqrt{WD} \left(\frac{a_r}{\rho_B g \sqrt{\beta_W}} \right) \quad (5)$$

The along-wind acceleration (a_D) is given by:

$$a_D = 4\pi^2 n_D^2 g_p \sqrt{\frac{KsF}{C_{eH}\beta_D} \frac{\Delta}{C_g}} \quad (6)$$

$$\frac{a_D}{g} = g_p \sqrt{\frac{KsF}{C_{eH}\beta_D} \left(\frac{3.9}{2 + \alpha} \right) \left(\frac{C_{eH}q}{D\rho_B} \right)} \quad (7)$$

(Taranath, 2010)

where

W, D = across-wind width and along-wind depth, respectively, in m

a_W, a_D = peak accelerations in across-wind and along-wind directions, respectively, in m/s^2

β_W, β_D = critical damping ratio in across-wind and along-wind directions, respectively

n_W, n_D = fundamental natural frequencies in across-wind and along-wind directions, respectively, in Hz

g = acceleration due to gravity = 9.81 m/s^2

g_p = statistical peak factor for the loading effect

K = factor related to surface roughness coefficient of terrain

s = size reduction factor

F = gust energy ratio

C_{eH} = exposure factor

α = power coefficient related to C_{eH}

q = reference wind pressure, in kPa = $650 \times 10^{-6} \times \bar{V}^2$, ($\bar{V}$ is reference wind speed, in m/s)

ρ_B = mass density of the building, in kg/m^3

The maximum wind-induced lateral displacement in the along-wind direction, Δ is usually calculated computationally. Substitution of this value in Eq. (6) establishes the best estimation of the acceleration in the along-wind direction, a_D . However, as an approximation for preliminary evaluations, the maximum deflection (Δ) can be assumed to be taken as $H/450$, which is the drift index generally implemented in wind design of tall buildings. By undertaking a linear modal representation for the building motion, Δ can be linked to the fundamental frequency of the building (Taranath, 2010). The resulting expression is presented in Eq. (7) for the ratio a_D/g and was used to calculate the along-wind acceleration instead of Eq. (6).

3. Results and Discussion

3.1 Site 1: London, United Kingdom

3.1.1 Wind actions obtained by EN1991.1.4:2005 and London CWT Model

Table 2 shows the wind-induced pressures on the target building calculated using EN1991.1.4:2005. These were determined for wind directions of 0° and 90° , by using the peak velocity pressure and the appropriate external pressure coefficients.

For the CWT testing, the dominant wind direction was ascertained using Autodesk Revit's virtual weather station. Figure 9 shows that for this site, the prevailing wind direction was south westerly and for the New York City site, it was north westerly.

The building wind loads were then obtained for *8 angles (45° increments)* using Autodesk Flow Design. Figure 10 shows the wind pressure pattern on the target building with a wind velocity of 21.99 m/s from the west. This value represented the basic wind velocity taken from the UK National Annex map. In order to determine the surface pressures due to wind action, the pressures belonging to each face were proportionately correlated with their corresponding values in pre-defined ranges (as indicated in the legends). Any pressures

which spanned the ranges were obtained using multiple linear interpolations until a representative value was obtained. For example, in the case of the narrow (26 m) face which was windward (i.e. zone D), the surface pressures comprised a 90:10 split between 2 ranges.

The total wind pressure was determined as follows:

Wind pressure (zone D) = $0.9(151.8^a) + 0.1(247.1) = 161.3$ Pa; ^a151.8 Pa was obtained through linear interpolation between 183.6 Pa and 120.0 Pa, in which the former was determined from interpolation between 247.1 Pa and 120.0 Pa.

This process was repeated for all wind directions and for both faces of the target building and the results are summarised in Table 3.

Wind direction had a significant impact on the external surface pressures. The highest windward pressures of 181.1 Pa and 161.3 Pa were experienced with winds from the South and West, respectively. These two wind directions provided the smoothest and least obstructive path for the wind to hit the target building. Conversely, the SW direction produced the lowest windward pressure, 66.2 Pa, which could be attributed to the presence of the surrounding structure facing upwind of the building. The negative pressures associated with North, NW, and NE wind directions, gave rise to leeward pressures, in particular, for wind directions from the NW and NE, as these resulted in insignificant windward pressures. This was due to the configuration of the surrounding structures, which effectively shielded the windward face of the target building when the wind was from those directions. The fact that the neighbouring terrain had structures, which were mostly outsizing the target building, resulted in the wind-induced pressures being negative rather than positive for NW and NE directions.

3.2 Site 2: New York City, United States of America

3.2.1 Wind actions obtained by ASCE 7-10 and New York City CWT Model

Wind pressures on surfaces based on ASCE 7-10 are summarised in Table 4, for which pressures were obtained for positive and negative GC_{pi} for both 0° and 90° directions. The dominant wind direction for this site was from the NW direction as evident from Figure 9b. Figure 11 shows the results from the New York City CWT test where the wind was applied from the NW with a velocity of 51 m/s. This value was the basic wind speed taken from the USA map in ASCE 7-10. Similar to the London scenario, when the pressures on the building faces overlapped the pre-defined ranges, the same technique (i.e. proportional correlation and multiple linear interpolations) described in Section 3.1.1, was used to obtain representative values. For the windward face, this proportional representation was a 70:20:10 split between ranges and for the leeward face it was a 70:30 split. Consequently, for these 2 faces, the total wind pressures on the target structure were calculated as follows:

$$\text{Wind pressure (windward face)} = 0.7(-477.0) + 0.2(-23.1^*) + 0.1(430.7) = -295.5 \text{ Pa}$$

$$\text{Wind pressure (leeward face)} = 0.7(-477.0) + 0.3(-23.1^*) = -340.8 \text{ Pa}$$

*-23.1 Pa was obtained through linear interpolation between 430.7 Pa and -477.0 Pa.

The remaining CWT results for all wind directions tested are presented in Table 5.

The emergence of negative pressures for the windward surfaces of the structure were evident in the majority of scenarios. This contrasted sharply with the results from the London site, in which only positive pressures were associated with the windward face of the building. As both target buildings had the same dimensions, these differences were clearly due to differences in wind velocity and differences in the terrain, congestion and dimensions of the structures surrounding the target building. In the New York City landscape, the condensed nature of the region surrounding the building meant that vortex shedding was so disorganised and fragmented that it led to the occurrence of varying pressure distributions on each face of the building. The highest positive windward pressure originated from the SW direction (321.1 Pa). Furthermore, this direction together with the SE direction (111.8 Pa) represented the only

two positive windward pressures. The negative leeward pressure on the building reached its peak with wind from the North (-641.6 Pa). This direction attracted the wind load diagonally to its surfaces, hence was more severe than the NE direction which acted perpendicularly to the building faces. The NW direction, which was identified as the dominant direction of wind for this site, did not produce the highest pressures. Hence, in this test environment, other wind directions proved more critical for wind loading. This emphasises the importance of testing a range of wind directions to ensure that the “worst case” scenario in terms of wind loading is identified. Figure 12 displays the complex wind flow surrounding the target building for both sites in which more disorganized vortices were existent in the New York City than London.

3.3 Wind actions obtained by CWT model without surrounding structures

Finally, the target building was modelled, stand-alone, in the CWT test environment. These simulations were carried out so that the effect of the surrounding structures could be deduced when compared with the earlier simulations. Three different wind directions (0° , 45° , and 90°) were modelled using wind velocities of ~ 22 m/s and 51 m/s for London and New York City, respectively.

Figure 13 shows the surface pressure distribution for the building exposed to London wind speeds and Figure 13b shows the same distribution on each of the building faces. The non-uniform nature of the pressure distribution was evident on the windward face of the building, whereas a far more homogenous (and lower) pressure trend was observed on the leeward and sidewall surfaces. The wind-induced pressures on all the faces of the building were calculated using the same method described in Section 3.1.1 and were found to be 437.7 Pa for the windward face, -249.6 Pa for the leeward face, and -252.2 Pa for the sidewall. As expected, the magnitudes of these pressures were considerably higher than those predicted when the surrounding structures were taken into account. The absence of any tall neighbouring

buildings (interfering buildings) increases the mean wind on the target building, hence there was no “shielding effect”. Shielding has an inverse relationship with the interference factor: for buildings of identical height, the shielding effect increases with increasing breadth whereas the interference factor decreases.

Figure 14 demonstrates the sophisticated flow regime represented by instantaneous velocity vectors when wind targeted the building’s broad face (0°). The dispersed flow at the dominant corners on the sidewalls and roof and the consequent near-wake turbulent zone were evident. Furthermore, vertical displacement of the arch vortex in the wake region and the alternate shedding development were visible. A highly stormy wake territory and a horseshoe vortex were formed as shown in Figure 14c, in which the horseshoe vortex formed upwind and swept around the edges of the building.

The average pressure distribution on the windward face, Figure 13b, shows the anticipated symmetry throughout the vertical centre line. It can be seen that the maximum drag pressure was at the stagnation point, which in this scenario, occurred at approximately $\frac{1}{2}H$, where H is the building height. Above this point, the flow transferred to the top of the building, whereas below this point the flow shifted downwards until it touched the ground. When investigating the lower section of the windward face, a higher pressure could be seen, which initiated the development of the horseshoe vortex. The disparity of the pressure was fairly even on the sidewall face, however, the pressure distribution on the leeward face was more asymmetric due to the disorganised vortex shedding in the near-wake.

As part of the validation process, drag plots were produced which showed the expected trends of drag coefficient (C_d) with respect to direction of flow. C_d value was at its peak when wind targeted the broader face, while it became lower when wind blew to the narrow face and diagonally at both faces, respectively (see SI Section S1).

3.4 Comparison of results for London and New York City sites

Figure 15 shows the significant difference between the results, in which the pressures obtained by EN1991.1.4:2005 and ASCE 7-10 were significantly higher than those obtained by the CWT testing. This variation in results shows the conservatism of the design standards, in which the following factors do play a significant role. The CWT results involve wind directionality, in contrast with the code evaluation where no directionality is considered. CWT testing takes into account the building geometry and the surroundings in greater detail than the design codes. This would lead to more accurate wind loading of both cladding and structural frames of high-rise buildings, in contrast with code analytical approaches that allow for preliminary design, but produce conservative wind loads on most occasions. The building is located in an area where all the surrounding structures are significantly greater than the target building size-wise. Consequently, the “shielding effect”, will reduce the mean wind load as a result of nearby buildings. This effect significantly decreases the surface pressures compared with the design codes where shielding is not taken into consideration. The severity of this impact relies on building orientation with surroundings, shape of the interfering buildings, and quantity of the neighbouring structures, all of which were prominent in the test cases. Codes such as the AS/NZS 1170.2:2011, as opposed to EN1991.1.4:2005 and ASCE 7-10, do reflect shielding effects by considering a ‘shielding multiplier (M_s)’ when obtaining the design velocity pressure.

As expected, examination of the results showed that the New York City site with 51 m/s of basic wind speed imposed greater wind pressures on the principal building than the London site with a wind speed of ~22 m/s (Figure 15). Other factors which may have attributed to the differences include ASCE 7-10 being based on a 3-second gust wind speed (700-year return period), while EN1991.1.4:2005 was based on a 600-second speed (50-year return period) (SI Table S4). The impact of the surrounding structures was prominent as the wind pressures

with the absence of the nearby buildings were much greater than when the building was analysed with their presence. This theory can be better understood by looking at Figure 15, where the varying margins between the scenarios are observed. In all of the cases (especially without surroundings) on both sites, wind impacting on the windward broad face (41 m) of the building resulted in greater pressures than wind moving towards the building's narrow face (26 m), as area is in direct relationship with load. This theorem was mostly applicable in the London model due to the simplicity of the terrain and the orientation of the building. One of the greatest windward pressures for the building located in London was obtained when the wind direction was on the narrow face due to the placement of two buildings adjacent to each other which initiated the "channelling effect".

The main differences between CWT studies and design codes are: (i) the design code analytical methods are usually defined as minimum design loads; in which they represent an upper envelope taking into account the majority of scenarios for standard building shapes (RWDI Consulting Engineers and Scientists); and (ii) CWT tests account for distinct factors such as impact of the adjoining buildings and topography, consideration of wind directionality effects, the aerodynamic effect of the actual shape of the building, and aero-elastic interaction between the structural motion and airflow (RWDI).

3.5 Comfort criteria

In the majority of cases, the results for the target building, satisfy the comfort criteria for the building's occupants. Scenarios a_1 and b_1 (Figure 16), with wind impacting on the broad face of the building, showed the controlling acceleration to be in the direction of the along-wind response due to the slenderness of the structure in that direction. Conversely, in scenarios a_2 and b_2 , when wind impacted on the narrow face, peak accelerations occurred in the across-wind direction, (i.e. perpendicular to the wind). This was due to the plan cross-

section of the building being narrower in this direction. In general, the across-wind accelerations govern the design for all buildings which are slender about both plan dimensions. This normally happens when $\sqrt{WD/H}$ is smaller than 1/3, where W and D stand for the across- and along-wind plan dimensions, respectively, and H is the building's height (Taranath, 2010).

The National Building Code of Canada (2005) recommends 10-year accelerations of up to 3% of acceleration due to gravity (g) for *office* buildings. The predicted 10-year peak accelerations at the top occupied level of the building ranged from 1.399 %- g (percentage of gravity acceleration) (for the London landscape) to 3.469 %- g (for the New York City landscape) taking into account a damping of 2%.

For the London landscape and its associated wind conditions, the computed accelerations were well below the allowable limit set out by NBCC 2005. For the New York City site, the accelerations were sufficiently high in some scenarios that the movement of the building at the top floor would be perceptible by occupants of the building. Most of the predicted accelerations were in the upper tolerable range for office usage, with two of them (3.469 %- g and 3.405 %- g for wind blowing to the broad and narrow face, respectively) considered unacceptable as they were within 15% of agreed limit (3 %- g). The value, 3.221 %- g for New York City for wind targeting the broad face (Figure 16, b₁) was, however, deemed safe as the exceedance was within 10% of the limit which was considered negligible. One probable explanation for the exceedance of the accelerations in the upper levels of the New York building was the comparatively low damping level of the structure. The susceptibility of a building to horizontal accelerations may be reduced by: (i) increasing the stiffness/frequency of the building, as the human's perception of building motions/accelerations is highly dependent on frequency of vibration; (ii) increasing the building's mass (Courtesy of Wacker

Ingenieure Wind Engineering, 2016); and (iii) provision of supplementary damping such as Tuned Mass Damper (TMD) and Distributed Viscous Dampers (Mendis et al., 2007).

4. Summary and Conclusions

This work has involved a comparison of design code calculations and CWT predictions for wind loading on an 80 m tall building. The numerical simulations were performed with the building located in 2 different landscapes, namely London and New York City, whilst the design codes were carried out on the same building with minimal surrounding effects. Horizontal peak accelerations due to the action of the wind on the top floor of the structure, were also calculated and discussed in terms of "comfort criteria". The aims were to (i) investigate the existing methods of obtaining wind loads on tall buildings and examine their applicability; (ii) study the effect of surrounding structures and wind directionality in the determination of wind loads in modern-day building design; and (iii) assess the along- and across-wind response of high-rise buildings under the induced action of wind with respect to comfort of its occupants.

The key conclusions drawn from this study are as follows:

- As expected, wind pressures on surfaces evaluated from EN1991.1.4:2005 and ASCE 7-10 were higher than those obtained from CWT testing. This was due to the fact that the design code results embody an upper envelope covering the majority of scenarios. They produce conservative results on most occasions since they offer minimum design loads. CWT tests, in contrast, take into account the influence of the surrounding structures, the true shape of the structure, and comprehensive wind directionality effects.
- Design codes EN1991.1.4:2005 and ASCE 7-10 do not incorporate shielding effects in chaotic environments such as London and New York City. This phenomenon is

included in other design codes such as AS/NZS 1170.2:2011 in the form of a “shielding multiplier” which also results in the reduction of wind loads if conducted in such terrains.

- CWT testing performed with the absence of the nearby structures demonstrated significantly greater surface pressures than the case with the surrounding terrain. The results with London wind velocities were on average closer to the predictions made by the corresponding code of practice (EN1991.1.4:2005) analytical approach, while those tested under New York City velocities exceeded the values of ASCE 7-10.
- Design standards with low averaging times and high return periods, such as ASCE 7-10, bring about wind speeds higher than those determined based on greater averaging time and lower return periods, like EN1991.1.4:2005 since they pursue two different approaches to obtain the basic wind speed.
- When the wind impacted on the broad face of the building, peak accelerations occurred in the along-wind direction, while when the wind impacted on the narrow face of the building, the crosswind direction produced greater accelerations.

CWT testing offers a relatively cheap and accurate tool which could be complimentary to full boundary layer experimental wind tunnel tests. It enables a wide range of buildings of varying dimensions and shapes to be quickly investigated in order to gain a better understanding the factors affecting the aerodynamics of these buildings, especially when modelled in-situ. This could provide a useful information for designers, planners or architects as part of their initial design phase and enable them to investigate a range of scenarios. The results can be validated or combined with traditional boundary layer wind tunnel tests subject to cost and availability. Geometrically-scaled models can be tested in the wind tunnel, where attention can be focussed on the principal building which can include the minor structural and

architectural details of the building. This would further enhance the quality and precision of the study.

5. References

- Ahlborn, B., Seto, M. and Noack, B. (2002). On drag, Strouhal number and vortex-street structure. *Fluid Dynamics Research* 30, 379-399.
- American Society of Civil Engineers Standard 7-05. *Minimum Design Loads for Buildings and Other Structures*. (2005).
- American Society of Civil Engineers Standard 7-10. *Minimum Design Loads for Buildings and Other Structures*. (2010).
- Australian/ New Zealand Standard 1170.2:2011. *Structural Design Actions. Part 2: Wind Actions*. (2011).
- Autodesk (2014). Flow Design Preliminary Validation Brief. *Autodesk Flow Design*.
- Bailey, P. (1985). Interference Excitation of Twin Tall Buildings. *Journal of Wind Engineering and Industrial Aerodynamics* 21, 323-338.
- Bashor, R. and Kareem, A. (2009). Comparative Study of Major International Standards. In: *The Seventh Asia-Pacific Conference on Wind Engineering*. Notre Dame, IL. 6.
- Bibri, S. and Krogstie, J. (2017). Smart sustainable cities of the future: An extensive interdisciplinary literature review. *Sustainable Cities and Society* 31, 183-212.
- Boggs, D., Hosoya, N., Cochran, L. (2000). Sources of Torsional Wind Loading on Tall Buildings: Lessons from the Wind Tunnel. In: *Structures Congress 2000: Advanced Technology in Structural Engineering*. Reston, VA: American Society of Civil Engineers. Reston, VA. 103.
- Broekhuizen, I. (2016). *Integrating Outdoor Wind Simulation in Urban Design*. Postgraduate. Luleå University of Technology.
- Chakravarthy, S. and Akdag, V. (2015). Importance of accuracy in CFD simulations. In: *6th BETA CAE International Conference*. p.1.
- English, E. (1990). Shielding Factors from Wind-Tunnel Studies of Prismatic Structures. *Journal of Wind Engineering and Industrial Aerodynamics* 36, 611-619.

English, E. (1993). Shielding Factors for Paired Rectangular Prisms: An Analysis of Along-Wind Mean Response Data from Several Sources. In: *Proceession of 7th US National Conference of Wind Engineering*. Los Angeles, CA. 193-201.

Eurocode 1: Actions on Structures. Part 1-4: General Actions – Wind Actions. (2005).

Franke, J., Hirsch, C., Jensen, A., Krüs, H., Schatzmann, M., Westbury, P., Miles, S. and Wisse, J. (2004). Recommendations on the use of CFD in wind engineering. Cost Action C14, C1. Available from:
<https://www.kuleuven.be/bwf/projects/annex41/protected/data/Recommendations%20for%20CFD%20in%20wind%20engineering.pdf> (accessed 29.08.2018)

Gu, M., Xie, Z., Huang, P. (2005). Along-Wind Dynamic Interference Effects of Tall Buildings. *Advances in Structural Engineering* 8, 623-636.

Holmes, J. (2007). *Wind Loading of Structures*. 2nd ed. New York: Taylor and Francis Group.

Hussain, M. and Lee, B. (1980). A Wind Tunnel Study of the Mean Pressure Forces Acting on a Large Group of Low-Rise Buildings. *Journal of Wind Engineering and Industrial Aerodynamics* 6, 207-225.

Kareem, A. (1987). Effect of Aerodynamic Interference on the Dynamic Response of Prismatic Structures. *Journal of Wind Engineering and Industrial Aerodynamics* 25, 365-372.

Khanduri, A., Stathopoulos, T., Bedard, C. (1998). Wind-Induced Interference Effects on Buildings – A Review of the State-of-the-Art. *Engineering Structures* 20, 617-630.

Kim, B. (2013). *Prediction of Wind Loads on Tall Buildings: Development and Applications of an Aerodynamic Database*. The University of Western Ontario. 5-27.

Lam, K., Leung, H., Zhao, J. (2008). Interference Effects on Wind Loading of a Row of Closely Spaced Tall Buildings. *Journal of Wind Engineering and Industrial Aerodynamics* 96, 562-583.

Lam, K. and Zhao, J. (2008). Interference Effects in a Group of Tall Buildings Closely Arranged in an L- or T-Shaped Pattern. *Wind and Structures* 11, 1-18.

701 Lin, N., Letchford, C., Tamura, Y., Liang, Bo, Nakamura, O. (2005). Characteristics of Wind
702 Forces Acting on Tall Buildings. *Journal of Wind Engineering and Industrial Aerodynamics*
703 93, 217-242.

704 Maharani, Y., Lee, S. and Lee, Y. (2009). *Topographical Effects on Wind Speed Over*
705 *Various Terrains: A Case Study for Korean Peninsula*. Taiwan. 1.

706 Maragkogiannis, K., Kolokotsa, D., Maravelakis, E. and Konstantaras, A. (2014). Combining
707 terrestrial laser scanning and computational fluid dynamics for the study of the urban thermal
708 environment. *Sustainable Cities and Society* 13, 204-216.

709 Mendis, P., Ngo, T., Haritos, N., Hira, A., Samali, B. and Cheung, J. (2007). Wind Loading
710 on Tall Buildings. *EJSE International*. 41-53.

711 *National Building Code of Canada, Volume 1, Division B*. (2005).

712 Reiter, S. (2008). Validation Process for CFD Simulations of Wind Around Buildings.
713 European Built Environment CAE Conference.

714 Roshko, A. (1993). Perspectives on Bluff Body Aerodynamics. *Journal of Wind Engineering*
715 *and Industrial Aerodynamics* 49, 79-100.

716 RWDI Consulting Engineers and Scientists. The Advantages of Using Wind Tunnel
717 Testing. *Technotes*. 1-3.

718 Sakamoto, H. (1985). Aerodynamic Forces Acting on a Rectangular Prism Placed Vertically
719 in a Turbulent Boundary Layer. *Journal of Wind Engineering and Industrial Aerodynamics*
720 18, 131-151.

721 Shames, I. (1982). *Mechanics of fluids*. 2nd ed. McGraw-Hill.

722 Simiu, E. and Scanlan, R. (1996). *Winds Effects on Structures: Fundamentals and*
723 *Applications to Design*. 3rd ed. John Wiley & Sons.

724 Simiu, E. (2011). *Design of Buildings for Wind: A Guide for ASCE 7-10 Standard Users and*
725 *Designers of Special Structures*. 2nd ed. John Wiley & Sons.

726 Soliman, B. (1976). *Effect of Building Group Geometry on Wind Pressure and Properties of*
727 *Flow*. Report No. BS 29. Department of Building Science. University of Sheffield. Sheffield,
728 U.K.

729 Stern, F., Wilson, R., Coleman, H. and Paterson, E. (2001). *Verification and Validation of*
730 *CFD Simulations* pp. 46. Available from: <http://files.asme.org/divisions/fed/16302.pdf>
731 (accessed 19.08.2018)

732 Streeter, V. (1951). *Fluid Mechanics*. McGraw-Hill, p.207.

733 Streeter, V. and Wylie, E. (1997). *Fluid Mechanics*. McGraw-Hill.

734 Stone, G. (1987). *Aerodynamic Interference Effects on Wind Loads and Responses of Tall*
735 *Buildings*. The University of Western Ontario.

736 Tamura, Y. and Kareem, A. (2013). *Advanced Structural Wind Engineering*. Springer.

737 Taniike, Y. (1991). Turbulence Effect on Mutual Interference of Tall Buildings. *Journal of*
738 *Engineering Mechanics* 117, 443-456.

739 Taranath, B. (2010). *Reinforced Concrete Design of Tall Buildings*. Boca Raton: CRC Press.
740 253-345.

741 *UK National Annex to Eurocode 1: Actions on Structures. Part 1-4: General Actions – Wind*
742 *Actions*. (2005).

743 Vickery, B. (1968). Load Fluctuations in Turbulent Flow. *Journal of Engineering Mechanics*
744 *Division: Proceedings of the American Society of Civil Engineers* 94, 31-46

745 Wacker Ingenieure Wind Engineering, (2016). *Final Report of Wind Tunnel Testing of a*
746 *Tower in Tehran, Iran*.

747 Xie, Z., Gu, M. (2009). Across-Wind Dynamic Response of High-Rise Building under Wind
748 Action with Interference Effects from One and Two Tall Buildings. *The Structural Design of*
749 *Tall and Special Buildings* 18, 37-57.

750

751 **List of Figure Captions**

752
753 **Figure 1** – Variation of Drag coefficient (C_d) against Reynolds number (Re) for a square
754 prism in two-dimensional flow.

755
756 **Figure 2** – Drag plot for a square prism (wind attack angle at 0°) in two-dimensional flow in
757 CWT.

Figure 4 – Visualisation of the target building: (a) building's structure in ETABS, and (b) building modelled with its cladding in Autodesk Revit.

Figure 5 – (a) Aerial view of the London site (Google maps), (b) Close-up view.

Figure 6 – (a) Site showing location of target building in London, (b) CWT testing of the proposed building.

Figure 7 – (a) Aerial view of the New York City site (Google maps), (b) Close-up view.

Figure 8 – (a) Site showing location of target building in New York City, (b) CWT testing of the proposed building.

Figure 9 – Annual wind rose diagrams: (a) London, and (b) New York City.

Figure 10 – Surface pressures on target building with westerly wind direction for London site.

Figure 11 – Surface pressures on target building with northwest wind direction for New York City site.

Figure 12 – Comparison of the wind flow pattern for W and SE directions: (a) London, and (b) New York City.

Figure 13 - (a) Surface pressures for a wind velocity of 21.99 m/s incident on the broad (41 m) face, (b) Mean pressure contours on all faces of the building.

Figure 14 – Wind-induced instabilities: (a) Fragmented arch vortex, (b) Alternate vortex shedding, (c) Turbulent wake and horseshoe vortex.

Figure 15 – Summary of the wind pressures on surfaces (Pa): (a) London, (b) New York City (SI Table S3).

Figure 16 – Wind-induced horizontal peak accelerations; NBCC 2005 procedure: (a₁) Site 1 (London), wind on broad face; (a₂) Site 1 (London), wind on narrow face; (b₁) Site 2 (New York City), wind on broad face; (b₂) Site 2 (New York City), wind on narrow face (SI Table S5).

List of Tables

Table 1 – Empirical data derived for a square prism in two-dimensional flow.

Drag coefficient (C_d)	Reynolds number (Re)	Attack angle	Reference
2.0	3.5×10^4	0°	(Steerter, 1951)
1.6	10^4 to 10^5	45°	
2.04	1.76×10^5	0°	(Ahlborn, Seto and Noack, 2002)
2.05	10^5	0°	

Table 2 – Wind pressures on target building (0° and 90° winds) for the London site. ^a

Target Building Face	Wind Pressures (Pa)
Sidewall (zone A)	-1217.070 ^b
Sidewall (zone B)	-811.380 ^b
Sidewall (zone C)	-507.113 ^b
Windward wall (zone D)	+811.380 (for $z_{e1} = 80$ m) ^b +629.531 (for $z_{e2} = 41$ m) ^b
Leeward wall (zone E)	-612.592 (0°) -555.795 (90°)

^a see SI Table S1; ^b is wind pressure applies to both 0° and 90° winds. * The zones are defined in EN1991.1.4:2005. A, B, and C are the sidewall zones, while, D is the windward and E is the leeward zones. Pressures are the same for 0° and 90° in all zones apart from zone E (leeward zone), where the external pressure coefficient is different.

Table 3 - Wind pressures on target building by CWT for London site. Note: the positive and negative signs represent windward and leeward pressures, respectively.

Wind direction	Wind pressure on target building surfaces (Pa)
0°/360° (N)	-127.7 (broad face)
45° (NE)	-67.4 (broad face), -60.3 (narrow face)
90° (E)	21.4 (narrow face)
135° (SE)	66.4 (broad face), 16.2 (narrow face)
180° (S)	181.1 (broad face)
225° (SW)	147.4 (broad face), 66.2 (narrow face)
270° (W)	161.3 (narrow face)
315° (NW)	-124.7 (broad face), -2.0 (narrow face)

Table 4 – Wind pressures on target building (0° and 90° winds) for the New York City site. ^a

Target Building Face	Wind Pressures (Pa)
Sidewall	-1363.315 ($GC_{pi} = +0.18$) ^b
	-730.033 ($GC_{pi} = -0.18$) ^b
Windward wall	879.558 ($GC_{pi} = +0.18$) ^b
	1512.840 ($GC_{pi} = -0.18$) ^b
Leeward wall	-1064.265 ($GC_{pi} = +0.18$) (0°)
	-430.983 ($GC_{pi} = -0.18$) (0°)
	-890.816 ($GC_{pi} = +0.18$) (90°)
	-257.535 ($GC_{pi} = -0.18$) (90°)

^a see SI Table S2; ^b is wind pressure applies to both 0° and 90° winds. * Pressures are the same for 0° and 90° in all zones apart from zone E (leeward zone), where the external pressure coefficient is different.

908 **Table 5 - Wind pressures on target building by CWT for New York City site.**

Wind direction	Wind pressure on target building (Pa)
0°/360° (N)	-641.6 (broad leeward face), -593.6 (narrow leeward face)
45° (NE)	-306.2 (broad windward face), -397.9 (broad leeward face)
90° (E)	-581.7 (broad leeward face), -188.1 (narrow leeward face)
135° (SE)	111.8 (narrow windward face), -324.3 (narrow leeward face)
180° (S)	-417.9 (broad leeward face), -360.8 (narrow leeward face)
225° (SW)	321.1 (broad windward face)
270° (W)	-291.9 (broad windward face), -268.6 (narrow windward face)
315° (NW)	-295.4 (narrow windward face), -340.8 (narrow leeward face)

909