Cite this article

An K, Hunt JCR and Fung JCH (2019)
Downwind flow behaviour of cuboid-shaped obstacles: modelling and experiments.
Proceedings of the Institution of Civil Engineers – Engineering and Computational Mechanics
172(1): 12–32, <https://doi.org/10.1680/jenclm.18.00024>

Research Article

Paper 1800024

Received 28/03/2018;
Accepted 22/03/2019;
Published online 07/05/2019

Keywords: buildings, structures &
design/computational mechanics/
environment

ICE Publishing: All rights reserved

Downwind flow behaviours of cuboid-shaped obstacles: modelling and experiments

Karl An PhD

Researcher, Division of Environment and Sustainability, Hong Kong
University of Science and Technology, Hong Kong, Hong Kong
(corresponding author: karl.an@connect.ust.hk)
(Orcid:0000-0003-4886-1574)

Julian C. R. Hunt PhD

Emeritus Professor, Department of Earth Sciences, University College
London, London, UK; Fellow, Trinity College, Cambridge, UK

Jimmy C. H. Fung PhD

Professor, Division of Environment and Sustainability, Hong Kong University
of Science and Technology, Hong Kong, Hong Kong; Department of
Mathematics, Hong Kong University of Science and Technology,
Hong Kong, Hong Kong (Orcid:0000-0002-7859-8511)

Buildings or fence-like structures are frequently modelled as cuboids in simulations of environmental assessments. Understanding flows around and downwind of typical isolated buildings, in the form of cuboids, provide insightful building disposition strategies, which are particularly useful in the field of urban planning. Fast analytical models or computational numerical models are essential for sensitivity studies of various design parameters that require validation against field/experimental data. The aim of this study is to compare in detail the robustness of analytical perturbation models and the computational fluid dynamics (CFD) $k-\epsilon$ turbulence models for predicting the mean wake velocity defects along the centre line downwind of the cuboids, both in the near- and far-wake regions. Depending on aspect ratios of cuboids, the decay rate of maximum velocity defect behaves differently. Moreover, the CFD code over-predicts the magnitude of maximum defect in comparison with the perturbation models, whereas it over-predicts or under-predicts when compared to actual measurements. A physical explanation has been proposed in this paper. The steady CFD models, requiring less empirical input, are shown to provide satisfactory results for approximate computations of flows in the near- and far-wake regions of cuboid buildings with appropriate settings of inflow profiles and wall function.

Notation

C_{F_x}	drag coefficient
D_y	eddy viscosity in neutral and stable conditions
D_z	eddy viscosity in neutral and stable conditions
H	height of obstacle
k	turbulent kinetic energy
L	length of obstacle
n	boundary layer velocity profile power-law exponent
p	decay rate of maximum velocity defect
U	mean velocity
u_*	friction velocity
$\hat{u}$	dimensionless magnitude of perturbation
W	width of obstacle
x_0	virtual origin
z_0	roughness parameter
Δu	perturbation velocity
δ	boundary layer thickness
ϵ	turbulent dissipation rate
κ	von Karman constant
λ_y	lateral wake scales
λ_z	vertical wake scales
σ_y	width of scalar plume

1. Introduction

1.1 Background

Bluff body aerodynamics is of considerable practical importance in the fields of urban planning and structural design. Investigation of the wind flow patterns around single or a cluster of buildings (with structural details generally simplified and represented as bluff bodies of various shapes) with different alignments have been emphasised in research studies over the last few decades. Extensive studies using wind tunnel experiments have been carried out on flows over a wide variety of shapes of bluff bodies (Castro, 1973; Castro and Robins, 1975; CODASC, 2008; Counihan, 1971; Lemberg, 1973; Woo *et al.*, 1976), field measurements and theoretical predictions (Counihan, Hunt and Jackson, 1974 (referred to as ‘CHJ’ hereafter); Kothari *et al.*, 1986) and numerical calculations (An and Fung, 2018; Gao and Chow, 2005; Markatos, 1986; Shirasawa *et al.*, 2003; Tominaga and Stathopoulos, 2009; Tominaga *et al.*, 2008; Yoshie *et al.*, 2007a).

These flows are complex due to the interactions between the mean flow and the turbulence generated by the obstacle, by the ground and in the boundary layer above the wake (see

Figure 1. Demonstration of the weak interaction of wake turbulence and external turbulence of flow over a single obstacle with hydrogen bubbles in a wind tunnel experiment at Warren Spring Laboratory, UK

Figure 1). In general, the mean velocity is reduced in the wakes of most obstacles, but for three-dimensional obstacles, the mean vorticity of the approaching flow is bent around the obstacle and distorted by the wake defect. For low turbulence levels, this leads to significant mean trailing vortices and a net increase in the mean velocity on the wake centre line (Kothari *et al.*, 1986; Smith *et al.*, 1977).

Turbulence plays an important role in influencing the behaviour of the flow structure, vorticity distribution and separation in fluid–structure interaction problems. Understanding of turbulent flow structures is one of the preliminary steps to comprehend the nature of fluid–structure interaction problems. Detailed time costly measurements are always one of the good approaches to study turbulent flow structures. However, carrying out some types of experimental measurements are considered not feasible and challenging. As an alternative method of choice, rather measure the required data experimentally, turbulence modelling by numerical calculations is informative in predicting characteristics of turbulent flows, while at the same time capable of accurately simulating the flow field.

1.2 Literature review

Turbulent flows, characterised by low momentum diffusion, high momentum convection and rapid variations in both pressure and velocity, are chaotic in nature but of central importance to bluff body aerodynamics problems and many engineering applications. Despite great efforts by engineers and scientists, accurate predictions of turbulent flows remain as a great challenge. There has been great emphasis on comprehending the physics of turbulent flows with introduction of new hypothesis, concepts and improvements in turbulence

models. The methods attempted by researchers in modelling of turbulence around bluff bodies can be mainly divided into two categories: analytical turbulence theories and numerical simulations.

Analytical turbulence theories, normally developed in Fourier space, form the foundation of various analytically based models (e.g. atmospheric dispersion modelling system (ADMS) model (CERC, 2013)) by focusing on interactions between the various scales of motion. However, analytical turbulence theories are relatively complicated and have not yet been widely applied to complex engineering problems. With the advancement of technology with better computational horsepower, numerical simulations with turbulence models become more prominent. These turbulence models can be further subdivided into categories based on whether the turbulent scales are fully and directly solved, partially solved and modelled or fully modelled.

Direct numerical simulation (DNS) (El Khoury *et al.*, 2013; Geurts, 2004; Lee and Moser, 2015; Trias *et al.*, 2010) attempts to simulate directly all the scales on a mesh with a grid size at least of order of magnitude of the Kolmogorov scale. It is obvious that obtaining the direct solution of the time-dependent Navier–Stokes equation without recourse to modelling is the ideal and robust way to solve turbulent flow problems. Despite its high robustness, the computational cost of DNS increases as the cube of Reynolds number (Andersson *et al.*, 2011; Issakhov, 2013; Tu *et al.*, 2018). Furthermore, the low numerical dissipation schemes should be adopted to allow all the wavelengths of the turbulent flow to naturally persist. Owing to the reason that grid stretching would increase the numerical dissipation of the algorithm, this leads to the higher-order numerical algorithms currently used in DNS are only valid on computational grids that contain uniform spacing. With the above-mentioned constraints, DNS approach remains non-feasible for most engineering applications (Andersson *et al.*, 2011; Yang, 2015).

There are tendencies for academia to adopt sub-grid-scale closure models such as large eddy simulation (LES) (Geurts, 2004; Lesieur *et al.*, 2005; Piomelli, 2014; Pope, 2000, 2004; Yang, 2015) to model turbulent flows. LES directly solve the large eddies within the energy production and the inertia regions of the energy spectrum while obtaining the nearly isotropic small-scale eddies by simple turbulence modelling. Similar to DNS, LES solutions are unsteady, and a low dissipative numerical algorithm is required. Relatively less computational cost than DNS, LES solutions must overcome a large number of time steps to eliminate starting transients in order to acquire a statistically stationary solution, which would result approximate simulation time in order of months. Therefore, although promising, LES solutions still require too much computational time to be useful for engineering applications.

The usage of turbulence transport models are the basis for most engineering applications. Reynolds averaged Navier–Stokes (RANS) turbulence transport models concentrate on seeking simplified modelling of the terms governing the transport of mass and momentum transport. These models treat dynamical quantities as statistically averaged fields and simulate only the gross features of turbulence. This would decrease the computational demands from order of months in LES simulations to hours/days through exclusion in the multiple length scales of eddies involved in turbulent flows.

Aiming to mitigate the main limitation of the incapability in modelling the inherently transient features of the flow field using steady RANS-family models and at the same time retain the advantages of LES models, hybrid RANS/LES methods have been developed in recent decades. This kind of approach is capable of reproducing a RANS-type behaviour near wall boundaries and a LES-type behaviour further away. Hybrid methods can be broadly classified into zonal and non-zonal methods (Hanjalic and Kenjeres, 2008; Hanjalic *et al.*, 2004), with most of hybrid RANS-LES models were initially developed in the framework of a zonal approach (Spalart, 2000; Spalart *et al.*, 1997; Travin *et al.*, 2000). RANS/LES zonal methods rely on a RANS model and a sub-grid-scale model, which are applied in different domains separated by a sharp or dynamic interface (Batten *et al.*, 2004; Davidson and Peng, 2003; Hamba, 2003, 2009; Temmerman *et al.*, 2005; Tessicini *et al.*, 2006). As the interface is empirically set inside the computational domain, the turbulence closure switches from one model to another without continuity when crossing the interface. Special treatments such as introduction of extra terms into the model equations and internal forcing produced by artificial instantaneous random fluctuations are necessary to restore continuity at the cross-flow interfaces between the different domains in order to obtain correct velocity and stress profiles in the boundary layer for zonal-type models (Davidson and Billson, 2006). On the contrary, non-zonal hybrid methods (Chaouat and Schiestel, 2005, 2007; Schiestel and Dejoan, 2005; Speziale, 1998), which assume that the governing equations are smoothly converting from a RANS behaviour to an LES behaviour, are of growing interest in hybrid RANS–LES modelling for simulation of complex turbulent flows within the engineering community.

Apart from typical and hybrid Eulerian methods, vortex particle methods, which are Lagrangian in nature, have apparent applications in academia and by the industry for flow simulations (Chatelain *et al.*, 2008; Coquerelle and Cottet, 2008; Ge and Xiang, 2008; Koumoutsakos and Leonard, 1995; Larsen and Walther, 1998; Ploumhans *et al.*, 2002; Smith and Stansby, 1988; Zhou *et al.*, 2002). The computational elements move like particles of the flow under the influence of the local forcing. As particle-based solvers do not necessarily require the

fulfilment of Courant–Friedrichs–Lewy (CFL) conditions imposed by grid-based solvers, and these methods do not require a grid to describe the flow around complex and deforming geometries, vortex particle methods are particularly suitable for exterior flow simulations (Cottet and Poncet, 2004; Rossinelli *et al.*, 2010). Furthermore, additional time can be saved by avoidance in generation of computational grids required in Eulerian methods by using boundary element methods (Wu, 1976) to impose the no-slip condition at the wall boundaries. However, the most important issue in flow simulations, which leads to inaccurate simulations by using particles method, is the Lagrangian distortion of the computational elements.

Particle mesh methods (Cottet *et al.*, 2000; Koumoutsakos, 1997; Monaghan, 1985) for flow simulations employ a grid to re-construct the particle locations when they get distorted. These methods have better computational speed than traditional vortex particle methods while retaining the robustness to perform efficient simulations. Originally developed in the context of vortex methods, immersed boundary methods (Mittal and Iaccarino, 2005; Sotiropoulos and Yang, 2014; Wang and Zhang, 2011; Yang *et al.*, 2009a) were introduced for particle mesh methods to tackle flow problems on more complex and deforming geometries.

In classical immersed boundary methods, depending on the configuration of immersed boundaries, the forces at these locations are calculated from appropriate constitutive laws. Apart from classical methods, there are improvement variations such as the direct forcing immersed boundary methods (Fadlun *et al.*, 2000; Kempe *et al.*, 2012; Uhlmann, 2005), immersed interface methods (Lee and Leveque, 2003; Li and Lai, 2001; Linnick and Fasel, 2005) and penalisation methods (Angot *et al.*, 1999; Bost *et al.*, 2010).

For the direct forcing immersed boundary methods, the forces at immersed boundaries are calculated based on the temporally discretised momentum equations. The approach for the forces calculation in the immersed interface methods is similar to classical immersed boundary methods where the effects of the immersed interfaces on the surrounding fluids are represented by singular forces.

By a local modification of the governing equations for the fluid, the solid obstacle in penalisation methods would be modelled as porous media with vanishing porosity by way of imposition of fictitious body forces. In order to ensure numerical stability and avoid numerical oscillations (Gazzola *et al.*, 2011; Kolomenskiy and Schneider, 2009), the fictitious body forces need to be distributed smoothly over the immersed boundaries as in the classical and direct forcing immersed boundary method. The main advantage in the usage of penalisation

techniques in particle methods is the simplification in the requirements for the grid structures used for re-meshing and allows computationally intensive flow simulations.

Despite there is a continuous introduction of improved turbulence models and methods in both Eulerian and Lagrangian frames, RANS methods are capable of providing ‘sufficiently accurate’ results and would continue to take up the main role in industrial applications (Leschziner, 1993; Murakami, 1997; Toparlar *et al.*, 2017; Yang, 2015). Moreover, simulation of the flow field by the CFD RANS approach has gradually reached a mature state, with well-developed technical guidelines (Britter and Schatzmann, 2016; Franke, 2006; Franke *et al.*, 2004; Tominaga *et al.*, 2008) and extensive experience through a long history of practice. For environmental and engineering computations to shorten the design cycle, the RANS turbulence models will further play an important role, especially in industrial engineering and environmental computations (Blocken, 2018; Hanjalic, 2005; Toparlar *et al.*, 2017; Yang, 2015).

1.3 Objectives

To deal with the many practical environmental and engineering problems such as estimation of pollutant dispersion (An *et al.*, 2019; Chen *et al.*, 2013; Cheng *et al.*, 2008; Di Sabatino *et al.*, 2007; Gromke and Blocken, 2015a, 2015b; Hang *et al.*, 2012a, 2012b; Ramponi *et al.*, 2015; Robins *et al.*, 1997; Salim, 2011; Thaker and Gokhale, 2016; Tominaga and Stathopoulos, 2012), gustiness and the temperature field (Kothari *et al.*, 1986; Yoshie *et al.*, 2007a, 2007b), and sensitivity studies of various atmospheric and model parameters (Ai and Mak, 2013; An *et al.*, 2013; Blocken, 2015; Blocken *et al.*, 2007a, 2007b; Garcia-Sánchez *et al.*, 2017; Liu *et al.*, 2018; Ricci *et al.*, 2017b; Tominaga and Stathopoulos, 2007, 2013), good descriptions and reasonably fast (in the order of minutes) computations of wake flows downwind of surface obstacles immersed in deep turbulent boundary layers are needed. The computations are sensitive to the assumptions made about the eddy viscosity approximations in analytical or computational models. This paper focuses on two computational models that are widely used to assess flows around buildings over terrains, which are analytically based models, the ADMS model (CERC, 2013) and the CFD $k-\epsilon$ turbulence models. The former is a three-dimensional self-similar perturbation model assuming linearised perturbations and a constant simple eddy viscosity for the perturbation velocity wake, which is determined by the flow around the body, and is approximately independent of the external turbulence (see Figure 1). The analytical model for the wake of a single obstacle, such as a cuboid, predicts how the perturbation mean velocity varies with downwind distance x , as if caused by a line or point obstacle upwind of the actual obstacle (CHJ, 1974).

Kothari *et al.* (1986) extended the theory to include the effects of trailing vortices in the wake, which induce down flow and positive velocity perturbations on the centre line, and has been confirmed in low-turbulence wind tunnel experiments. This study focuses on neutrally stratified flows. The effect of trailing vortices is significant in boundary layers with low turbulence including stably stratified flows, but it is not significant in the high-turbulence flows that occur in cities or over rough terrains.

The computational fluid dynamics (CFD) solver uses computational mesh grids that need to be much smaller than the scale of the target obstacles (Franke *et al.*, 2011; Murakami *et al.*, 1987). Although RANS $k-\epsilon$ turbulence models with eddy viscosity hypothesis have been applied extensively and are capable of modelling the main features of mean velocity fields around isolated obstacles within a decent amount of time, they can only provide approximate estimates for the mean velocity in the wake (e.g. Durbin, 1995). Murakami (1997) showed that the RANS $k-\epsilon$ CFD models tend to overestimate the change in turbulence upwind and above the obstacles, but they under-predict the turbulence (especially the large-scale eddies) in the near wake because the $k-\epsilon$ turbulence models do not accurately model the ‘sheltering’ interactions between the wake turbulence and the external boundary layer turbulence (Hunt and Durbin, 1999). They tend to give relatively non-satisfactory results for the mean velocity in the far wake downwind of the obstacle, and this phenomenon is discussed in this paper.

The aim of this study is to serve as additional literature to provide completeness to the existing research on flows over bluff bodies, by assessing the performance of analytical perturbation models (of CHJ, 1974; Kothari *et al.*, 1986), the perturbation theory-based ADMS model and the $k-\epsilon$ family of turbulence models, in prediction of the mean wake velocity defects along the centre line downwind of cuboids not only at the near wake, but also on the far wake. It should be emphasised here that this paper does not serve as another literature to criticise or review the advantages/disadvantages of $k-\epsilon$ family models, in which there have already been extensive research efforts on such areas.

To the best of the author’s knowledge, due to the diminishing effect of the obstacle to the flow behaviours at locations far away from it, emphasis has been put on flow behaviours predictions at recirculation/near-wake regions immediate downwind of obstacles in most of the studies. Therefore, researchers commonly overlook the wind flow behaviours at far downwind regimes and in terms of completeness, it is considered useful to evaluate the performance of commonly applied two equation eddy viscosity turbulence models or simple analytical models in prediction of the rate of wind profile recovery and understand the wind flow behaviour there.

2. Methodology

2.1 Perturbation theory and ADMS wake model

The perturbation analytical wake theory is based on the observation that an obstacle of height H causes small perturbations $\Delta u(x, y, z)$ to the mean velocity profile $U(z)$ of the turbulent boundary layer. The vertical variations of Δu with z typically have maximum values at $z_{\Delta u_{\max}}$ that increase with distance x downwind of the location of the obstacle (see Figure 2). In the near-wake region, where $x \sim H$ and $z_{\Delta u_{\max}} \sim H$, the surface stress has a small effect on the deficit momentum flux, which is therefore approximately equal to the drag. In this region, Δu is of the order of $U(H)$ (CHJ, 1974), but further downwind, the surface stress acts to reduce the perturbation velocity. Nevertheless, Δu is described by a similarity analysis along the same lines as for wakes in uniform flows. This led to two-/three-dimensional perturbation theory for Δu . The far wake of a three-dimensional bluff body immersed in a deep boundary layer differs from that of a two-dimensional body in the lateral spreading of the wake in the y direction. This paper does not consider shear flows with a low level of turbulence in which longitudinally oriented vortices generated by the body extend far downstream (Kothari *et al.*, 1986; Smith *et al.*, 1977).

2.1.1 Perturbation theory

The theory for the momentum wake of a three-dimensional object in a turbulent boundary layer predicts the mean velocity, which is expressed as the sum of the upwind velocity profile $U(z)$ and a perturbed profile $\Delta u(x, y, z)$ – that is, $u(x, y, z) = U(x, y, z) + \Delta u(x, y, z)$

$$1. \quad U(z) = U(H) \left(\frac{z}{H} \right)^n$$

where $U(z)$ is the mean upwind velocity at a distance z above the ground, and H is the cuboid height. For a typical boundary layer wind profile over rough surfaces, the power-law exponent $n = 0.16$. With a small value of the power-law index n ($n \sim 0.1-0.2$), the maximum velocity deficit decays at a rate of $x^{-1.5}$, where x is the distance travelled downstream in the wake region.

The model for the three-dimensional perturbation velocities can be expressed, following Kothari *et al.* (1986) and Woo *et al.* (1976), as

$$2. \quad \Delta u(x, y, z) = kU(H)F(z_1, y_1)/[(x - x_0)/H]^{(3+n)/(2+n)}$$

where x_0 is the virtual origin of the similarity profile in the order of H , n is the boundary layer velocity profile power-law exponent, $k = 0.21 C_{Fx} (W/H) / \{4\lambda^{1/2} \gamma [2\kappa^2 n / (2+n)]^{(3+n)/(2+n)}\}$, W is the width of the building, λ and γ are coefficients of order 1, which are adjusted to give optimum agreement between the theoretical and experimental results, $\kappa = 0.41$, which is the von Karman constant, and C_{Fx} is the drag coefficient of the obstacle. The function F that determines Δu is expressed in terms of stretched coordinates as

$$3. \quad F(z_1, y_1) = \frac{\{\eta^{1/2+n} \exp\{-[\eta + y_1^2/(1.5 + \eta)]\}\}}{(\eta + 1.5)^{1/2}}$$

$$4. \quad \eta = \frac{z_1^{2+n}}{[2(2+n)\kappa^2 n \gamma]}$$

Figure 2. Configuration and symbol definition of flow over a three-dimensional obstacle showing the typical wake with a velocity defect ($\Delta u > 0$). The diagram also shows how the trailing vorticity produced directly by the obstacle and by the wake defect induces down flow and a limitation of the wake width λ_y . In some cases, such as with low turbulence, a velocity excess ($\Delta u < 0$) may be generated in the far wake, which is not considered in this study (source: modified and adapted from Kothari *et al.*, 1986)

$$5. \quad y_1 = \frac{(y/H)}{[(x-x_0)/H]^{1/(2+n)}} \left(\frac{2+n}{2\gamma\lambda n\kappa^2} \right)$$

$$6. \quad z_1 = \frac{(z/H)}{[(x-x_0)/H]^{1/(2+n)}}$$

2.1.2 ADMS model

A simplified equation for the velocity defect Δu_{ADMS} , which is based on the perturbation theory stated above, has been proposed for practical calculations by Apsley (1988) and is given by

$$7. \quad \Delta u_{ADMS}(x, y, z) = U(H) \hat{u} \left(\frac{W}{2\lambda_y} \right) \left(\frac{H}{\lambda_z} \right)^2 g(\xi) h(\eta)$$

Note that formula (7) is published on the Cambridge Environmental Research Consultants (CERC) website (CERC, 2013).

Here, $\hat{u} = (C_G/\sqrt{\pi})$ is the dimensionless magnitude of the perturbation, where $g(\xi) = (\xi/2) \exp(-\xi^2/4)$ and $h(\eta) = (1/2\sqrt{\pi}) \exp(-\eta^2/4)$, in which $\xi = (z/\lambda_z)$ and $\eta = (y/\lambda_y)$. λ_y and λ_z are the lateral and vertical of the wake scales. They are given as $\lambda_y = [(D_y(x-x_0)/u_0(H))]^{1/2}$ and $\lambda_z = [(D_z(x-x_0)/u_0(H))]^{1/2}$, respectively. D_y and D_z are the eddy viscosities in stable and neutral conditions in which $D_y = \kappa u_* H$ and $D_z = 2\kappa u_* H$, and u_* is the friction velocity. Note that the lateral width λ_y , which increases in proportion to $x^{1/2}$ as a result of the dynamics of the wake, grows more slowly than the width of a scalar plume σ_y , because in the atmosphere, σ_y grows approximately in proportion to x for sources located near the ground (e.g. Pasquill and Smith, 1983).

In this study, the distance of the virtual origin x_0 from the building uses the empirical formula in the ADMS wake model (CERC, 2013). It is chosen to be consistent with the observed condition that $u=0$ at a distance x_r within the recirculating wake. For typical cuboids, $x_r = L/2$, where L is the length of the obstacle in the flow direction, so that the virtual origin is at

$$8. \quad x_0 = \frac{L}{2} - \left(\frac{U(H)}{D_z} \right) \left[\frac{\hat{u}WH^2}{2} \sqrt{\frac{D_z}{D_y}} \max g(\xi) \max h(\eta) \right]^{2/3}$$

The equation for calculating the virtual origin (Equation 8) remains a pragmatic way of determining its location for single block-shaped obstacles (CERC, 2013). For elongated cuboids, where $W \gg H$, the x_0 is located further upwind of the obstacle,

as for two-dimensional wakes (CHJ, 1974), with the velocity defect far downwind following $\Delta u_{\max} \sim (x/H)^{-1}$. To determine more accurately the location of the virtual origin for obstacles with other geometrical shapes, wind tunnel experiments or detailed simulations may be required.

2.2 CFD $k-\epsilon$ family models

A turbulence model needs to be chosen for every CFD simulation of external flows. The most common practice with turbulence models is to use 'two-equation models' such as the $k-\epsilon$ family turbulence models. The most widely used types of $k-\epsilon$ models are named the standard $k-\epsilon$ model, the re-normalisation group (RNG) $k-\epsilon$ model and the realisable $k-\epsilon$ model.

The conclusions of recent academic and industrial research on the use of CFD for flow over buildings (Hanna *et al.*, 2002; Sada and Sato, 2002) is that the standard $k-\epsilon$ model underestimates reverse flow in the separated region at the leeward side of a building due to its overestimation of turbulence energy k at the impinging region of the building wall (Franke *et al.*, 2011; Tominaga *et al.*, 2008). Also, the blocking of large eddies in the approaching boundary layer by the obstacle and the mean shear of the wake are not represented (e.g. Britter *et al.*, 1979; Hunt and Savill, 2005). These deficiencies affect the predictions of the flow structure. The RNG $k-\epsilon$ model proposed by Yakhot *et al.* (1992) has a modified form of the turbulent dissipation equation, which partially accounts for the different scales of motion through changes to the production term. The other $k-\epsilon$ model used here, the realisable $k-\epsilon$ model proposed by Shih *et al.* (1995), ensures more physically realistic turbulence properties because this model modifies the transport equation for the turbulent dissipation rate and also the coefficient C_μ varies.

Parente *et al.* (2011a, 2011b) recently modified the standard $k-\epsilon$ model by redefining the model coefficient C_μ to vary with the friction velocity u_* and inflow turbulent kinetic energy. The addition of a source term correction in the turbulence dissipation equation allows implementation of a more realistic form of turbulent kinetic energy profile in comparison with the constant profile proposed by Richards and Hoxey (1993).

Although RANS $k-\epsilon$ models are still widely applied in the industrial field due to its relatively shorter computation time in comparison with more advanced turbulence models, there are still limitations to these classes of turbulence models due to the exclusion of multiple length scales of eddies involved in complex interactions between the wake turbulence and the external turbulence flows in the modelling process, which contributes to certain discrepancies between simulation results and the real physical situation.

3. Homogeneity of velocity and turbulence quantities in CFD $k-\varepsilon$ models

Self-maintained velocity profiles throughout the domain for CFD simulations are important to ensure an accurate solution in the far wake of obstacles. To ensure stream-wise homogeneous flow within the computational domain, appropriate settings for the CFD model are important. Overlooking of the stream-wise homogeneity of the velocity profile has consequences on the development of the mean stream-wise velocity, turbulent kinetic energy and turbulent dissipation rate and hence affects the prediction of the maximum velocity deficit in the wake of obstacles between the ground and the shear layer. The standard wall function would lead to velocity profile inhomogeneity resulting in a faster velocity deficit decay rate at the far-wake region downstream of the obstacle.

To maintain velocity profile homogeneity, the modified wall function proposed by Balogh *et al.* (2012) was implemented together with the Richards and Hoxey (1993) inflow conditions (Equations 9–11) for the RNG and realisable $k-\varepsilon$ models. For the modified $k-\varepsilon$ model, the modified wall function was used together with the modified turbulent viscosity with the set of inflow profiles stated in Equations 9, 11 and 12. The set of $k-\varepsilon$ models with the modified wall function and the set used the standard wall function were tested on an empty domain. Velocity, turbulent kinetic energy and turbulent dissipation rate profiles were plotted at the centre line of the inflow and outflow faces of the empty domain for comparison.

3.1 Richards and Hoxey inflow conditions

Richards and Hoxey (1993) proposed inflow profiles of mean velocity, turbulent kinetic energy and turbulent dissipation rate for CFD $k-\varepsilon$ model simulations and are stated as

$$9. \quad U(z) = \frac{u_*}{\kappa} \ln \left(\frac{z + z_0}{z_0} \right)$$

$$10. \quad k(z) = \frac{u_*^2}{\sqrt{C_\mu}}$$

$$11. \quad \varepsilon(z) = \frac{u_*^3}{\kappa(z + z_0)}$$

where u_* is the friction velocity and z_0 is the roughness parameter, $\kappa = 0.41$ and $C_\mu = 0.09$. The disadvantage of this set of inflow profiles is that the inflow turbulent kinetic energy profile remains a constant value and does not match the

realistic situation in which the turbulent kinetic energy would decrease with the increment of height.

3.2 Inflow conditions for the modified $k-\varepsilon$ model

To input a more realistic turbulent kinetic energy profile, Parente *et al.* (2011a, 2011b) adopted the turbulent kinetic energy profile in the form of

$$12. \quad k(z) = C_1 \ln(z + z_0) + C_2$$

where C_1 and C_2 are adjusting parameters that determine the shape of the turbulent kinetic energy profile. A similar form of turbulent kinetic energy profile as in Equation 12 was also proposed by Yang *et al.* (2009b).

By further modification of the model constant $C_\mu = (u_*^4/k^2)$ to relate with friction velocity and inflow turbulent kinetic energy together with the addition of an extra source term

$$S_\varepsilon = \frac{u_*^4}{(z + z_0)^2} \left[\frac{(C_{\varepsilon 2} - C_{\varepsilon 1}) \sqrt{C_\mu}}{\kappa^2} - \frac{1}{\sigma_\varepsilon} \right]$$

where $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$ and $\sigma_\varepsilon = 1.3$, to the turbulent dissipation rate equation, Equations 9–12 become a set of valid inflow profiles for the modified $k-\varepsilon$ model. However, the constants C_1 and C_2 in Equation 12 should be carefully chosen to avoid large values of C_μ , which would lead to numerical convergence issues in the CFD simulations.

3.3 Modified wall function

The standard wall function is defined as

$$13. \quad \frac{u_+}{u_*} = \frac{1}{\kappa} \ln(Ez^+)$$

where $E = 9.793$ and $z^+ = (\rho C_\mu^{0.25} k_w^{0.5} z_w / \mu)$.

The modified wall function (Balogh *et al.*, 2012) introduced a new wall function constant $E' = (\mu / z_0 u_*)$ and non-dimensional wall distance $z'^+ = (u_*^* (z + z_0) / \mu)$, which was simply a shifting of z^+ by the roughness parameter z_0 . This redefinition in the modified wall function removes the disadvantages of the standard wall function while maintaining it in equation form.

3.4 CFD simulation on an empty domain

The modified $k-\varepsilon$ model, RNG $k-\varepsilon$ model and realisable $k-\varepsilon$ model together with different types of wall functions were tested on an empty computational domain to investigate the model performances in maintaining homogeneity of velocity and turbulent quantities. The empty computational domain was of the below size 14 m (L) $\times$ 4 m (W) $\times$ 0.6 m (H).

The first cell size from the ground wall was 0.005 m in height. The two lateral sides of the domain were set as symmetry boundary conditions, the top face was set with velocity and turbulent quantities at domain height, the outflow face was set as a zero gradient condition, the ground was set as the wall boundary condition and the inflow face was set as the velocity inlet condition with inflow profiles defined in Equations 9–12 depending on different models with parameters $u_* = 0.2263$ m/s, $z_0 = 0.00003$ m, $C_1 = -0.036$ and $C_2 = 0.15$.

Second-order upwind schemes were used in the advection terms for the calculations of momentum, turbulent kinetic energy and turbulent dissipation rate. The green Gauss node-based gradient scheme was applied for the diffusion term, and the convergence criteria for the residuals of the quantities are set to 1×10^{-5} . Figure 3 shows the velocity, turbulent kinetic energy and turbulent dissipation rate profiles at the centre line of the inflow and outflow faces resulting from different combinations of the $k-\varepsilon$ turbulence models and wall functions.

From Figure 3, it is observed that the modified $k-\varepsilon$, RNG $k-\varepsilon$ and realisable $k-\varepsilon$ models together with the modified wall function perform satisfactorily in maintaining velocity profile homogeneity, whereas the RNG $k-\varepsilon$ and realisable $k-\varepsilon$ models with the standard wall function slightly over-predict near the ground at the outflow location. For the turbulent kinetic energy and turbulent dissipation rate homogeneity, the profiles match well at the inflow and outflow locations for the models with modified wall function. In addition, turbulent kinetic energy homogeneity is satisfactorily achieved for the RNG $k-\varepsilon$ and realisable $k-\varepsilon$ models input with the Richards and Hoxey (1993) inflow conditions. However, these two models coupled with the modified wall function perform slightly better than those coupled with the standard wall function near the ground level. To summarise, models coupled with the modified wall function perform relatively better than those coupled with the standard wall function in terms of profile homogeneity, particularly near the ground.

4. Geometry and CFD parameter settings for the simulation cases

4.1 Geometry settings

Four single block obstacles with different dimensions were created. The first case (case A) was a cubic block that followed the dimension of the block used in the meteorological wind tunnel experiment performed at the Fluid Dynamics and Diffusion Laboratory, Colorado State University, by Kothari *et al.* (1986) in a simulated boundary layer with thickness $\delta = 0.508$ m and a cube with dimensions $H = L = W = 0.0762$ m. Case B was a cuboid elongated parallel to the flow direction with $L/W \approx 3$. The block in case C was rotated 90° degrees to that in case B. Finally, the block in case D was a cuboid

elongated perpendicular to the flow direction with $W/L \approx 40$. The wake velocity defects of the four cases along the centre line were calculated applying the perturbation theory, the ADMS and the CFD $k-\varepsilon$ models and were compared in this study. The geometry and dimensions of the obstacles are shown in Figure 4, while the parameter settings for the perturbation theory and the ADMS model for the investigated cases are tabulated in Table 1. It should be emphasised that no comparison has yet been made between the $k-\varepsilon$ and perturbation models for obstacles with $W > 10H$, which are of great practical interest for pollution/sound barriers or for estimating sand blown over dunes. The variations in the wake thicknesses of these obstacles differ. For the cube (case A), the vertical and lateral wake thicknesses grow with x from the same virtual origin at approximately the same rate, but for the other three obstacles, the virtual origin is different for the growth of the thicknesses in the y and z directions. These aspects have not been considered in detail previously.

4.2 Computational domain settings and boundary conditions

The first task before carrying out a CFD simulation is to determine the size and shape of the computational domain and the corresponding boundary conditions imposed on the different faces of that domain. The influence of side walls demarcating the lateral side can be reduced by using a large enough computational domain size (Blocken *et al.*, 2007a, 2007b; Li *et al.*, 2006). The computational domain size was set according to the CFD guidelines (Franke *et al.*, 2011; Tominaga *et al.*, 2008), which recommend that the lateral and top boundaries can be set at $5H$ or further from the target building, where H is the height of the tallest building. The outflow boundary should also be set at least $10H$ behind the target building (Mochida *et al.*, 2002; Shirasawa *et al.*, 2003).

It is worthwhile to mention here that flow behaviours at the wake regions would vary according to the geometric properties of cuboid obstacles. The commonly adopted definition of near and far wake would be based on the flow behaviour characteristics at the wake. However, in this study, it was instead characterised based on flow characteristics, the near-/far-wake regions were defined based on the obstacle height H with $5H < x < 20H$ being the near- to mid-wake regime and $x > 20H$ being the far-wake regime. The computational domain size (see Figure 5) in this study extended the distances $26H$ upwind of the obstacle, greater than $\pm 26H$ in the lateral directions (because, as explained above, the wake dynamics limit lateral spread) and $8H$ in the vertical direction. Because the object of this study is to investigate the behaviour of the velocity defect as x varies from the near-wake region ($1H$ – $5H$ distance from the obstacle) to the far-wake region (distance more than $20H$ from the target cuboid), the domain extends a distance $158H$ in the downstream direction to ensure a far enough buffer distance from the outflow boundary.

Figure 3. Profiles of velocity, turbulent kinetic energy and turbulent dissipation rate at the centre line of inflow and outflow faces for different $k-\epsilon$ models coupled with different wall functions and inflow turbulent kinetic energy profiles

Figure 4. Dimensions of the block obstacles used in the study

For computation, the two side boundaries of the computational domain were set with symmetry boundary condition. The top boundary was set with velocity and turbulent quantities at the domain height. The ground boundary was set as a rough wall with modified wall function, the inflow boundary condition was set as the velocity inlet boundary condition and the outflow condition was adopted at the outlet boundary as shown in Figure 5.

4.3 Inflow profiles and numerical schemes

For any of the turbulence models chosen, one must impose an initial inflow wind profile, turbulent kinetic energy profile and turbulent dissipation rate profile at the inlet face of the computational domain. A logarithmic wind profile, as stated in Equation 9, is input as the inflow wind profile, with the parameters $u_* = 0.2263 \text{ m/s}$ and $z_0 = 0.00003 \text{ m}$, which fits well the power-law wind profile as stated in Equation 1 with power-law exponent $n = 0.16$. The $k-\varepsilon$ family steady-state turbulence models also require the appropriate estimation of a turbulent kinetic energy profile at the inflow boundary to carry out the numerical simulations.

For the RNG $k-\varepsilon$ and realisable $k-\varepsilon$ models, the inflow turbulence profile proposed by Richards and Hoxey (1993) as stated in Equation 10 was used, whereas for the modified $k-\varepsilon$ model, the inflow turbulence profile as stated in Equation 12 was applied with constants $C_1 = -0.036$ and $C_2 = 0.15$. The inflow turbulent dissipation rate profile was estimated as $\varepsilon = u_*^3 / \kappa(z + z_0)$, where u_* is the friction velocity, and z_0 is the roughness parameter.

Second-order upwind schemes were used in the advection terms for the calculations of momentum, turbulent kinetic energy and turbulent dissipation rate. The green Gauss node-based gradient scheme was applied for the diffusion term. COST Action C14 (Franke *et al.*, 2004) suggests that scaled residuals in iterations should be less than 1×10^{-4} . Throughout the simulations, scaled residuals of each of the transport equations were monitored to determine whether the solution has converged. In this study, the convergence criteria of the residuals were 1×10^{-5} (Casey and Wintergerste, 2000). Table 2 summarizes the settings for the CFD model.

4.4 Resolution of the computational grid

A grid-sensitivity test was performed on three types of grids: a coarse grid, a basic grid and a fine grid. The basic grid has approximately 2 million structured mesh grid cells. The grid expansion ratio in both the horizontal and vertical directions was at most 1.2 with the smallest grid cell in the vertical direction being approximately 1/25 of the cuboid height. The coarse grid has approximately 1.4 million cells, with the grid cell number being approximately 30% less than that of the basic grid while the fine grid has approximately 2.9 million cells, with the grid amount more than 30% than the basic grid. Grid sensitivity analysis was carried out for all the $k-\varepsilon$ models adopted in this study. For the aim of brevity, only the results of the mesh sensitivity study on case A are presented, similar results have also been obtained for the rest of the cases (i.e. cases B, C and D).

Figure 6 illustrates the views of the basic grid. Figure 7 depicts a comparison of the results of the velocity magnitude profiles based on the three different grids by various $k-\varepsilon$ models for

Table 1. Parameter settings for the perturbation theory and the ADMS model

	Case A	Case B	Case C	Case D
H : m	0.0762	0.0625	0.0625	0.0625
Dimension of block obstacle ($L \times W \times H$)	$H \times H \times H$	$2.54H \times 0.78H \times H$	$0.78H \times 2.54H \times H$	$0.5H \times 20H \times H$
$U(H)$: m/s	4.27	4.26	4.26	4.26
Boundary layer height (δ): m	0.508	0.508	0.508	0.508
Power-law index of wind profile	0.16	0.16	0.16	0.16
Drag coefficient C_{F_x}	1.3	1.3	1.4	1.3
Adjustable constant λ, η	1.5	1.15	1.2	1.5
Virtual origin x_0	Obtained from the ADMS model (Equation 8)			$\sim 8H$

Figure 5. Diagram illustrating the computational domain and boundary conditions (case A)

case A. There are no observable differences between the simulation results among the three grids for the downwind velocity magnitude profiles at $x/H = 5$, $x/H = 10$, $x/H = 20$, $x/H = 40$ and $x/H = 80$ behind the block obstacle. No significant grid sensitivity was identified; therefore, the resolution of the basic grid is considered sufficient and has been adopted for the cases in this study.

5. Performance of the models

In the theory for the momentum wake of a three-dimensional object, where the boundary layer velocity profile has a small

power-law index ($n \sim 0.1 - 0.2$), the maximum value of the mean velocity defect $\Delta u_{\max}$ decays as x^{-p} , where x is the distance travelled downstream in the wake region and p varies over finite range from 1 to 1.5 – that is, $1 < p < 1.5$ (CHJ, 1974; Hosker, 1984).

In this study, the experimental results for maximum velocity defect $\Delta u_{\max}$ along the centre line were first plotted in Figure 8 for the cube in case A. In the same figure the data are compared with the results of the perturbation theory, the ADMS wake model and the $k-\varepsilon$ family models. In Figure 9, the experimental and modelled results are compared using self-similar plots of the vertical profiles of the defect velocity $\Delta u(x, y=0, z)$ for the cube (case A).

For case A, cube obstacles, the experimental data in Figure 8 for $\Delta u_{\max}$ from Lemberg (1973), Counihan (1971), Woo *et al.* (1976) and Kothari *et al.* (1986) demonstrate approximate self-similarity on a logarithmic scale plot for the maximum defect of velocity along the centre line of the wake, the observed decay rate being approximately $x^{-1.5}$ for $x \geq 2H$. Comparing the experimental data with modelling results, the perturbation theory and ADMS wake model results for $\Delta u_{\max}$ are proportional to $x^{-1.3}$ for $x > 5H$. This decay rate continues until about 20 building heights downwind and the decay rate further slightly increases to $x^{-1.4}$ from 20 building heights till 100 building heights. The decay rate of approximately $x^{-1.5}$ is also valid after 20 building heights for the enhanced $k-\varepsilon$ turbulence models (i.e. the RNG $k-\varepsilon$ and realisable $k-\varepsilon$ models). The decay rate transits from approximately $x^{-1.2}$ at near- to mid-wake regime (i.e. $5H < x < 20H$) to approximately $x^{-1.6}$ at

Figure 6. Computational mesh grids for case A (cube)

Figure 7. Downwind wind profiles along centre line for various $k-\varepsilon$ models under different grid resolutions (case A – cube)

Table 2. Summary of settings for the CFD model

CFD model	
Computational domain size	26H inflow buffer and 26H for lateral region 158H downstream region, 8H in vertical region
Boundary conditions	Symmetry condition for the two side boundaries Top specified stress Wall boundary condition for the ground Velocity inlet condition for the inflow boundary Outflow condition for the outlet boundary
Turbulence models	Modified $k-\varepsilon$ model RNG $k-\varepsilon$ model Realisable $k-\varepsilon$ model
Numerical scheme	Second order for the pressure term Second-order upwind schemes for advection terms
Wall function	Modified wall function
Convergence criteria	Scaled residuals dropped to below 1×10^{-5}

the far-wake regime (i.e. $x > 20H$) for the RNG $k-\varepsilon$ model, while for the realisable $k-\varepsilon$ model, decay rate transition from approximately $x^{-1.1}$ at near- to mid-wake regime to approximately $x^{-1.5}$ at the far-wake regime is observed. However, the modified $k-\varepsilon$ model predicts a slower decay rate proportional to around $x^{-0.8}$ at near- to mid-wake regimes and transits to a decay rate of only x^{-1} , which deviates far away from the theory and observations.

To summarise, for cubic-shaped obstacles, in terms of maximum velocity deficit decay rate, there is a good agreement between experimental results and the modelled results obtained from the perturbation theory and the ADMS model. The enhanced versions of $k-\varepsilon$ family models have better matching to the theoretical hypothesis (i.e. the maximum value of the mean velocity defect $\Delta u_{\max}$ decays as $x^{-1.5}$, where x is the distance travelled downstream in the wake region) in comparison to the modified $k-\varepsilon$ model at the far-wake regimes. Although at the near- to mid-wake regimes, the enhanced versions of

Figure 8. Logarithmic scale plot of maximum wind velocity defect along centre line of wake (case A – cube)

Figure 9. Vertical profiles of the velocity defect plotted in terms of similarity variables. Comparison of perturbation theory, ADMS model and $k-\epsilon$ family models along centre line with various experiment data – case A (cube)

$k-\epsilon$ family models obtain slightly slower mean velocity defect decay rates of approximately $x^{-1.1}$ to $x^{-1.2}$, these models still have satisfactory performance as compared with the modified $k-\epsilon$ model. Note that the specification of the virtual origin x_0 in the perturbation theory model and ADMS model affects the comparisons with the experiments. The values used in

Equations 2 and 7 were optimised based on experimental data, which include those as adopted in Figure 8. It is clear that the CFD models (which have no empirical adjustments for these data sets) over-predict the magnitude of the maximum velocity defect by a factor of 3 averaged from 4 to 14 building heights, in comparison with the perturbation theory and the ADMS

wake model. The RNG $k-\varepsilon$ and realisable $k-\varepsilon$ models perform better than the modified $k-\varepsilon$ model, having a closer match in the maximum velocity defect with the perturbation theory. However, closer to the obstacle, the $k-\varepsilon$ turbulence models have similar performance with over prediction in the maximum velocity defect.

In Figure 9, self-similar plots of the vertical profiles of the mean velocity defect Δu are presented, following the theoretical forms in Equations 2 and 7. The measurements of Counihan (1971) and Lemberg (1973) are consistent with this form of the self-similar plot and the theoretical profiles. The experimental data of Castro (1973) matches the ADMS model better than the perturbation theory. By contrast, the $k-\varepsilon$ models not only over predict the maximum defect, but also under predict by about 30% the height where the maximum defect of the velocity profiles appear. This indicates that the magnitude of the eddy viscosity is too small in the separated flow region, and is consistent with many other studies of wakes of bluff bodies (e.g. Durbin, 1995). In addition, not much difference has been spotted for the various $k-\varepsilon$ models.

For case B, the cuboid was elongated in the direction along the wind and for cases C and D, they were elongated perpendicular to the wind. The model results for the maximum velocity defect $\Delta u_{\max}$ are plotted on a logarithmic scale in Figures 10–12. Measurement data from a two-dimensional rectangular block which is the closest available data that

matches the geometry of the cuboid in case B are plotted for comparative purpose. For cases C and D, measurements for wakes behind two-dimensional triangular ‘hills’ perpendicular to the flow and behind porous fences in atmospheric winds have been provided for comparison with the models.

It is noticeable that for case B, the results for $\Delta u_{\max}$, derive with both types of models, can be represented as power-law decay formulae – that is, $\Delta u_{\max} \sim (x/H)^{-p}$, provided that p has two values over different ranges; in the near- to mid-wake ($5H < x < 20H$) where $p \sim 1$, and in the far wake ($x > 20H$) where $p \sim 1.5$. A likely explanation is that the shear layer thickness λ_z is small where the flow passes over the top of the block but increases rapidly just downwind of the obstacle (i.e. $x > 0$), where the large eddies form in the separated flow. However, the lateral thickness λ_y of the shear layer along the sides of the block is determined by the trailing vortices along the sides, which move downwind without much outwards growth. These vortical structures merge with the eddying in the separated region ($x > 0$), and therefore it takes a further distance before λ_y increases significantly above its value at $x = 0$. Thus, p is determined by the vertical growth of the near-/mid-wake with $p \sim 1$. Further downwind the wake turbulence causes a similar growth rate in the vertical and lateral directions – that is, $p \sim 1.5$.

Note that in the perturbation and ADMS models, the logarithmic plot of velocity defect is affected by the location of the

Figure 10. Logarithmic scale plot of maximum wind velocity defect along centre line of wake (case B – cuboid elongated in direction parallel to flow, $L = 3W$)

Figure 11. Logarithmic scale plot of maximum defect of velocity along centre line of wake (case C – cuboid elongated in direction perpendicular to flow, $W=3H$)

Figure 12. Logarithmic scale plot of maximum defect of velocity along centre line of wake (case D – cuboid with greater elongation in direction perpendicular to flow, $W=20H$)

virtual origin x_0 , located at the upwind of the near wake – that is, $x_0 < 0$. The exponent p is small near $x=0$, and will increase as x increases, reaching an approximately constant value when $x > 2L$. The result in Figure 10 shows a similar variation

in p for both types of model (with more obvious two different self-similar regions observed for the CFD models), despite some differences in the underlying assumptions of eddy viscosity formulation between the analytical and $k-\epsilon$ models.

For case C, where the cuboid was elongated in a direction perpendicular to the flow direction, two self-similarity regions as in case B are observed. The logarithmic plot of $\Delta u_{\max}$ in Figure 11 shows that in the far wake ($x > 20H$), for the analytical model results, the power-law decay rate is approximately $x^{-1.3}$ and $x^{-1.5}$ for the enhanced $k-\varepsilon$ models, while at the near- to mid-wake ($5H < x < 20H$), the power-law decay rate is approximately x^{-1} . However, if an alternative parameterisation of the virtual origin x_0 was chosen in the perturbation model, following Kothari *et al.* (1986), a significant difference is observed just behind the obstacle ($x < 2H$); however, similar decay rate prediction is observed at the near-, mid- and further extension into the far-wake regime.

For the wake of the block in case D, illustrated in Figure 12, the analytical models show a similar value of the algebraic decay rate with $p \sim 1$. However, this type of nearly two-dimensional wake is more sensitive to the interactions between the higher amplitude decaying wake turbulence and the external boundary layer turbulence. By comparison of the near-wake regions, the results show a significant difference in the decay rates of $\Delta u_{\max}$, in which for the cuboid in case C acquires a decay rate of $p \sim 1$, whereas for case D, p has a value of much smaller than 1 (i.e. $p \ll 1$). The reason behind is that in the former case, the separated near-wake flow is relatively shorter and at further downwind regions, the wake turbulence spreads upwards and the vertical thickness grows with the downstream distance x . With the wider cuboid as in case D, where the reverse flow extends over a longer distance downwind, the vertical thickness λ_z grows more slowly. These differences are somehow consistent with the self-similar analytical model, with appropriate specification of the virtual origin x_0 .

Measurements of the maximum velocity defect $\Delta u_{\max}$ at the far wakes downwind of various two-dimensional surface bluff obstacles in turbulent boundary layers are also presented in Figures 11 and 12 for comparison. These data include a fence (Mons and Sforza, 1968; Sforza and Mons, 1970) and two-dimensional wedges (Plate and Lin, 1965) in a wind tunnel boundary layer, and also thin and thick screens in the atmospheric boundary layer (Nägeli, 1953).

As shown by CHJ (1974), all these experimental results for $\Delta u_{\max}$ have self-similar decay rates with $p \sim 1$. The significance of the model results for longer elongated cuboids, that are not completely two dimensional, is that the decay rates of $\Delta u_{\max}$ for far wakes behind obstacles ($x/H > 20$), as predicted by the theoretical models and the $k-\varepsilon$ models are in fact similar to the theory and experiments for two-dimensional obstacles.

Furthermore, for cases C and D, the comparisons between the predicted magnitudes by the analytical and turbulence models

of $\Delta u_{\max}$ have shown that the turbulence model results exceed the analytical model ones for the defect velocity in the far wakes by about 30% or more. However in the near wake, neither models is well adapted, the turbulence model results are greater or less than the analytical models, respectively.

6. Conclusions

This paper has supplemented the available research on prediction performance of perturbation models and CFD $k-\varepsilon$ turbulence models on flows over bluff body at both near- and far-wake regimes. The merits of perturbation-based analytical and numerical models for the prediction of flow structures starting from the near regime continuously extended to far-wake regimes downwind of the cuboid-shaped obstacles have been compared. The wakes downwind of four surface-mounted cuboids with different dimensions (denoted as cases A–D) and orientated either parallel or perpendicular to a turbulent wind have been investigated. The available data from various wind tunnel experiments for the wakes of cubes (case A), and other wake experiments for two-dimensional obstacles in wind tunnel flows and atmospheric turbulent boundary layer were presented and compared with model and theory results. The comparisons have shown that the broad features of wakes of typical building shapes could be estimated with both simple formulae models and CFD models.

The analytical models are generally consistent with the experimental results for the velocity defect in the near- to mid-wake (where the perturbation velocity Δu was comparable with the upwind velocity U) and in the far wake (where Δu was small compared with U , but was larger than the turbulence fluctuation u' in the upwind flow) for the cases in this study. Self-similar structure for the velocity defect in the far wake has been obtained, defined by an algebraic decay $\Delta u_{\max} \sim (x/H)^{-p}$ that persisted for a considerable distance of about 20 building heights and further downwind with small variation in the values of $p \sim 1.3 - 1.4$ for three-dimensional obstacles, such as the cube (case A) considered here. These values of p are consistent with those in published papers (Counihan, 1971; CHJ, 1974.)

A new finding here is that when the models are applied to elongated obstacles such as case B (where the obstacle dimension of $L=3W$), cases C (where obstacle dimension of $W=3H$) and D (where obstacle dimension of $W=20H$), there exist two separate self-similar regions of the wake, where p has different values. In cases B and C, $p \sim 1$ for $x/H < 20$ and increased to values of $p \sim 1.1 - 1.6$ for $x/H > 20$, depending on the type of models adopted. For approximately two-dimensional obstacles (case D), $p \ll 1$ at near-wake regions and $p \sim 1$ has been observed at the far-wake region.

The results of the $k-\varepsilon$ family of models showed broadly similar regions of self-similarity and similar values for the exponent p , yet there are differences. For the wakes of cuboids in cases A, B and C, there was a more rapid rate of decay of $\Delta u_{\max}$ for $x > 20H$, respectively, leading to the exponents p exceeding the values of p from the analytical models. For the greater elongated cuboids perpendicular to the wind flow direction as in case D, departure from self-similarity was notable for the $k-\varepsilon$ turbulence models, leading to a reduction in the resulting value of $p \sim 1.0$ from the analytical models.

Despite the broad agreement about the self-similar decay rates between the analytical and turbulence models, the results for the $k-\varepsilon$ family models have proven that there was an over-prediction (relative to the analytical models) in magnitude of the maximum velocity defect by a factor of 2–3 for the cube (case A). For both cases B and C, the magnitude of the maximum velocity defect predicted by CFD models exceeded by an order of 30% in comparison with the results from perturbation models, but when compared with results from various experiments, slight under-prediction in $\Delta u_{\max}$ has been observed. However, for the nearly two-dimensional cuboid in case D, the magnitude was slightly under predicted in the near wake, but significantly over-predicted (by up to 100%) in the far wake for the three $k-\varepsilon$ turbulence models in comparison with the perturbation theory-based models.

Both types of models would provide adequate robustness in prediction of velocity decay rates, yet perturbation theory-based analytical models have displayed better prediction in magnitude of maximum velocity defect ($\Delta u_{\max}$) till far-wake regimes (20 building heights and further downwind) in comparison with $k-\varepsilon$ turbulence models. The advantage of the steady CFD models using the implicit eddy viscosity hypothesis is that wake flows can be modelled for arbitrary-shaped obstacles and arbitrary inlet conditions. Broadly $k-\varepsilon$ models over-predict turbulent mixing upwind (due to too much dependence on straining), slightly over/under-predict in the near wake (depending on cuboid shapes) and over-predict in the far wake (because the wake shear is not diffused by the boundary layer turbulence as much as predicted by $k-\varepsilon$ model). However, numerous studies with these limitations are being conducted using unsteady direct simulations and unsteady turbulence models for sub-grid scale motions (e.g. Braza *et al.*, 2006).

It is notable that the fast approximate analytical methods require specifying eddy diffusivity and defining, for the self-similar velocity defect profiles, the virtual origin x_0 in relation to the dimensions of the obstacle and the magnitude of the perturbation ($\bar{u}$). Despite this empiricism, the perturbation and ADMS wake models have been optimised to provide useful estimates for the wakes of a wide range of obstacle shapes.

To conclude, the perturbation models can provide rapid calculation with adequate robustness on broad features of wakes for typical building shapes, whereas the steady CFD $k-\varepsilon$ models are useful in understanding wake behaviour of structures with arbitrary shapes without requiring any empirical input, but with a slight sacrifice of computational time compared with the analytical models. Both model types are essential and provide insights for understanding the flow behaviour in practical structural design problems and environmental air ventilation studies. Nevertheless, it should be noted that typically, the analytical models are not generally applicable in the region of recirculating flow downwind of an obstacle. More tests and further parameterisation are needed to apply the models to buildings located at arbitrary orientations and in incident boundary layers with varying stability.

Acknowledgements

This work was conducted at the Hong Kong University of Science and Technology and by Professor Julian Hunt at the University of Hong Kong (as a distinguished visiting professor). It was also supported by the Cambridge Environmental Research Consultants (CERC), UK. The authors would like to thank Dr. David Carruthers, director of CERC, for his valuable advice during the preparation and modelling stage of the study.

REFERENCES

- Ai ZT and Mak CM (2013) CFD simulation of flow and dispersion around an isolated building: effect of inhomogeneous ABL and near-wall treatment. *Atmospheric Environment* **77**: 568–578.
- An K and Fung JCH (2018) An improved SST $k-\omega$ model for pollutant dispersion simulations within an isothermal boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics* **179**: 369–384.
- An K, Fung JCH and Yim SHL (2013) Sensitivity of inflow boundary conditions on downstream wind and turbulence profiles through building obstacles using a CFD approach. *Journal of Wind Engineering and Industrial Aerodynamics* **115**: 137–149.
- An K, Wong SM and Fung JCH (2019) Exploration of sustainable building morphologies for effective passive pollutant dispersion within compact urban environments. *Building and Environment* **148**: 508–523.
- Andersson B, Andersson R and Håkansson L (2011) *Computational Fluid Dynamics for Engineers*. Cambridge University Press, Cambridge, UK.
- Angot P, Bruneau C and Fabrie P (1999) A penalization method to take into account obstacles in incompressible viscous flows. *Numerische Mathematik* **81**(4): 497–520.
- Apsley DD (1988) *A Model for Dispersion in the Wake of Large Buildings*. Central Electricity Generating Board, UK, CEGB Report RD/L/3359/R88.
- Balogh M, Parente A and Benocci C (2012) RANS simulation of ABL flow over complex terrains applying an enhanced $k-\varepsilon$ model and wall function formulation: implementation and comparison for FLUENT and OpenFOAM. *Journal of Wind Engineering and Industrial Aerodynamics* **104–106**: 360–368.

- Batten P, Golberg U and Chakravarthy S (2004) Interfacing statistical turbulence closures with large-eddy simulation. *American Institute of Aeronautics and Astronautics* **42**(3): 485–492.
- Blocken B (2015) Computational fluid dynamics for urban physics: importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations. *Building and Environment* **91**: 219–245.
- Blocken B (2018) LES over RANS in building simulation for outdoor and indoor applications: a foregone conclusion? *Building Simulation* **11**(5): 821–870.
- Blocken B, Carmeliet J and Stathopoulos T (2007a) CFD evaluation of wind speed conditions in passages between parallel buildings-effect of wall-function roughness modifications for the atmospheric boundary layer flow. *Journal of Wind Engineering and Industrial Aerodynamics* **95**(9–11): 941–962.
- Blocken B, Stathopoulos T and Carmeliet J (2007b) CFD simulation of the atmospheric boundary layer: wall function problems. *Atmospheric Environment* **41**(2): 238–252.
- Bost C, Cottet G and Maître E (2010) Convergence analysis of a penalization method for the three-dimensional motion of a rigid body in an incompressible viscous fluid. *SIAM Journal of Numerical Analysis* **48**(4): 1313–1337.
- Braza M, Perrin R and Hoarau Y (2006) Turbulence properties in the cylinder wake at high Reynolds number. *Journal of Fluids and Structures* **22**(6–7): 757–771.
- Britter R and Schatzmann M (2016) *Model Evaluation Guidance and Protocol Document*. COST Office, Brussels, Belgium. See <http://www.cost.eu/media/publications/07-64-Model-Evaluation-Guidance-and-Protocol-Documents> (accessed 08/01/2016).
- Britter RE, Hunt JCR and Mumford JC (1979) The distortion of turbulence by a circular cylinder. *Journal of Fluid Mechanics* **92**(2): 269–301.
- Casey M and Wintergerste T (2000) *ERCOTAC Special Interest Group on Quality and Trust in Industrial CFD, ERCOTAC Best Practice Guidelines, European Research Community on Flow, Turbulence and Combustion*. ERCOTAC, London, UK.
- Castro IP (1973) *An Exponential Investigation of the Flow Around a Surface Mounted Cube in a Uniform Free Stream*. Central Electricity Generating Board, UK, CEGB Rep. R/M/N687.
- Castro IP and Robins AG (1975) *The Effect of a Thick Incident Boundary Layer on the Flow Around a Small Surface Mounted Cube in a Uniform Free Stream*. Central Electricity Generating Board, UK, CEGB Rep. R/M/N795.
- CERC (Cambridge Environmental Research Consultants) (2013) *Modelling of Building Effects in ADMS*. Cambridge Environmental Research Consultants, Cambridge, UK.
- Chaouat B and Schiestel R (2005) A new partially integrated transport model for subgrid-scale stresses and dissipation rate for turbulent developing flow. *Physics of Fluids* **17**(6): 1–19.
- Chaouat B and Schiestel R (2007) From single-scale turbulence models to multiple-scale and subgrid-scale models by Fourier transform. *Theoretical and Computational Fluid Dynamics* **21**(3): 201–229.
- Chatelain P, Curioni A, Bergdorf M et al. (2008) Billion vortex particle direct numerical simulations of aircraft wakes. *Computer Methods in Applied Mechanics and Engineering* **197**(13–16): 1296–1304.
- Chen B, Liu S, Miao Y, Wang S and Li Y (2013) Construction and validation of an urban area flow and dispersion model on building scales. *Acta Meteorologica Sinica* **27**(6): 923–941.
- Cheng WC, Liu CH and Leung YC (2008) Computational formulation for the evaluation of street canyon ventilation and pollutant removal performance. *Atmospheric Environment* **42**(40): 9041–9051.
- CODASC (Concentration Data of Street Canyons) (2008) *Concentration Data of Street Canyons*. Laboratory of Building and Environmental Aerodynamics, Karlsruhe Institute of Technology, Karlsruhe, Baden-Württemberg, Germany.
- Coquerelle M and Cottet GH (2008) A vortex level set method for the two way coupling of an incompressible fluid with colliding rigid bodies. *Journal of Computational Physics* **227**(21): 9121–9137.
- Cottet GH and Poncet P (2004) Advances in direct numerical simulation of 3D wall-bounded flows by vortex-in-cell methods. *Journal of Computational Physics* **193**(1): 136–158.
- Cottet GH, Koumoutsakos P and Ould-Salihi ML (2000) Vortex methods with spatially varying cores. *Journal of Computational Physics* **162**(1): 164–185.
- Counihan J (1971) *An Experimental Investigation of the Wake Behind a Two-Dimensional Block and Behind a Cube in a Simulated Boundary Layer Flow*. Central Electricity Research Laboratories, Leatherhead, Surrey, UK, note number RD/LN 115/71.
- Counihan J, Hunt JCR and Jackson PS (1974) Wakes behind two-dimensional surface obstacles in turbulent boundary layers. *Journal of Fluid Mechanics* **64**(3): 529–564.
- Davidson L and Billson M (2006) Hybrid LES-RANS using synthesized turbulent fluctuations for forcing in the interface region. *International Journal of Heat and Fluid Flow* **27**(6): 1028–1042.
- Davidson L and Peng SH (2003) Hybrid LES-RANS modelling: a one-equation SGS model combined with a $k-\omega$ model for predicting recirculating flows. *International Journal for Numerical Methods in Fluids* **43**(9): 1003–1018.
- Di Sabatino S, Buccolieri R, Pulvirenti B and Britter R (2007) Simulations of pollutant dispersion within idealized urban-type geometries with CFD and integral models. *Atmospheric Environment* **41**(37): 8316–8329.
- Durbin PA (1995) Separated flow computations with the $k-\epsilon-v_2$ model. *American Institute of Aeronautics and Astronautics* **33**(4): 659–664.
- El Khoury GK, Schlatter P, Noorani A et al. (2013) Direct numerical simulation of turbulent pipe flow at moderately high Reynolds numbers. *Flow Turbulence and Combustion* **91**(3): 475–495.
- Fadlun E, Verzicco R, Orlandi P and Mohd-Yusof J (2000) Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations. *Journal of Computational Physics* **161**(1): 35–60.
- Franke J (2006) Recommendations of the COST action C14 on the use of CFD in predicting pedestrian wind environment. *The Fourth International Symposium on Computational Wind Engineering, Yokohama, Japan*, pp. 529–532.
- Franke J, Hirsch C, Jensen AG et al. (2004) Recommendations on the use of CFD in wind engineering. *Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics*. In van Beeck JPAJ (ed.). COST Action C14, Impact of Wind and Storm on City Life Built Environment, von Karman Institute, Sint-Genesius-Rode, Belgium, pp. C.1.1–C.1.11.
- Franke J, Hellsten A, Schlunzen K and Carissimo B (2011) The COST 732 best practice guideline for CFD simulation of flows in the urban environment: a summary. *International Journal of Environment and Pollution* **44**(1–4): 419–427.
- Gao Y and Chow WK (2005) Numerical studies on air flow around a cube. *Journal of Wind Engineering and Industrial Aerodynamics* **93**(2): 115–135.
- García-Sánchez C, Van Tendeloo G and Gorlé C (2017) Quantifying inflow uncertainties in RANS simulations of urban pollutant dispersion. *Atmospheric Environment* **161**: 263–273.
- Gazzola M, Chatelain P, van Rees WM and Koumoutsakos P (2011) Simulations of single and multiple swimmers with non-divergence

- free deforming geometries. *Journal of Computational Physics* **230**(19): 7093–7114.
- Ge YJ and Xiang HF (2008) Computational models and methods for aerodynamic flutter of long-span bridges. *Journal of Wind Engineering and Industrial Aerodynamics* **96**(10–11): 1912–1924.
- Geurts BJ (2004) *Elements of Direct and Large-Eddy Simulation*. Edwards, Philadelphia, USA.
- Gromke C and Blocken B (2015a) Influence of avenue-trees on air quality at the urban neighborhood scale. Part I: quality assurance studies and turbulent Schmidt number analysis for RANS CFD simulations. *Environmental Pollution* **196**: 214–223.
- Gromke C and Blocken B (2015b) Influence of avenue-trees on air quality at the urban neighborhood scale. Part II: traffic pollutant concentrations at pedestrian level. *Environmental Pollution* **196**: 176–184.
- Hamba F (2003) A hybrid RANS/LES simulation of turbulent channel flow. *Theoretical and Computational Fluid Dynamics* **16**(5): 387–403.
- Hamba F (2009) Log-layer mismatch and commutation error in hybrid RANS/LES simulation of channel flow. *International Journal of Heat and Fluid Flow* **30**(1): 20–31.
- Hang J, Li Y, Sandberg M, Buccolieri R and Di Sabatino S (2012a) The influence of building height variability on pollutant dispersion and pedestrian ventilation in idealized high-rise urban areas. *Building and Environment* **56**: 346–360.
- Hang J, Li Y, Sandberg M, Buccolieri R and Di Sabatino S (2012b) On the contribution of mean flow and turbulence to city breathability: the case of long streets with tall buildings. *Science of the Total Environment* **416**: 362–373.
- Hanjalic K (2005) Will RANS survive LES? A view of perspectives. *Journal of Fluids Engineering* **127**(5): 831–839.
- Hanjalic K and Kenjeres S (2008) Some developments in turbulence modeling for wind and environmental engineering. *Journal of Wind Engineering and Industrial Aerodynamics* **96**(10–11): 1537–1570.
- Hanjalic K, Hadziabdic M, Temmerman L and Leschziner M (2004) Merging LES and RANS strategies: zonal or seamless coupling?. In *Direct and Large-Eddy Simulation V* (Friedrich R, Geurts B and Métais O (eds)). Springer, Dordrecht, Netherlands, pp. 451–464.
- Hanna SR, Tehrani S, Carissimo B, Macdonald RW and Lohner R (2002) Comparisons of model simulations with observations of mean flow and turbulence within simple obstacle arrays. *Atmospheric Environment* **36**(32): 5067–5079.
- Hosker RP (1984) Flow and diffusion near obstacles. In *Atmospheric Science and Power Production* (Randerson D (ed.)). Technical Information Center, Office of Scientific and Technical Information, U.S. Department of Energy, Oak Ridge, Tennessee, USA, pp. 241–326.
- Hunt JCR and Durbin PA (1999) Perturbed vortical layers and shear sheltering. *Fluid Dynamics Research* **24**(6): 375–404.
- Hunt JCR and Savill AM (2005) Guidelines and criteria for the use of turbulence models in complex flows. In *Turbulence Modelling* (Hewitt GF and Vassilicos JC (eds)), Cambridge University Press, Cambridge, UK, pp. 291–343.
- Issakhov A (2013) Numerical methods for solving turbulent flows by using parallel technologies. *Journal of Computer and Communications* **1**(1): 1–5.
- Kempe T and Fröhlich J (2012) An improved immersed boundary method with direct forcing for the simulation of particle laden flows. *Journal of Computational Physics* **231**(9): 3663–3684.
- Kolomenskiy D and Schneider K (2009) A Fourier spectral method for the Navier–Stokes equations with volume penalization for moving solid obstacles. *Journal of Computational Physics* **228**(16): 5687–5709.
- Kothari KM, Peterka JA and Meroney RN (1986) Perturbation analysis and measurements of building wakes in a stably stratified turbulent boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics* **25**(1): 49–74.
- Koumoutsakos P (1997) Inviscid axisymmetrization of an elliptical vortex ring. *Journal of Computational Physics* **138**(2): 821–857.
- Koumoutsakos P and Leonard A (1995) High-resolution simulation of the flow around an impulsively started cylinder using vortex methods. *Journal of Fluid Mechanics* **296**: 1–38.
- Larsen A and Walther JH (1998) Discrete vortex simulation of flow around five generic bridge deck sections. *Journal of Wind Engineering and Industrial Aerodynamics* **77–78**: 591–602.
- Lee L and Leveque RJ (2003) An immersed interface method for incompressible Navier–Stokes equations. *SIAM Journal on Scientific Computing* **25**(3): 832–856.
- Lee M and Moser RD (2015) Direct numerical simulation of turbulent channel flow up to $Re_\tau=5200$. *Journal of Fluid Mechanics* **774**: 395–415.
- Lemberg R (1973) *On the Wakes Behind Bluff Bodies in a Turbulent Boundary Layer*. PhD thesis and Boundary Layer Wind Tunnel Report No. BLWT-3-73, University of Western Ontario, London, Ontario, Canada.
- Leschziner MA (1993) Computational modelling of complex turbulent flow – expectations, reality and prospects. *Journal of Wind Engineering and Industrial Aerodynamics* **46–47**: 37–51.
- Lesieur M, Metais O and Comte P (2005) *Large-Eddy Simulations of Turbulence*. Cambridge University Press, Cambridge, UK.
- Li Z and Lai MC (2001) The immersed interface method for the Navier–Stokes equations with singular forces. *Journal of Computational Physics* **171**(2): 822–842.
- Li QS, Fu JY, Xiao YQ et al. (2006) Wind tunnel and full-scale study of wind effects on China’s tallest building. *Engineering Structures* **28**(12): 1745–1758.
- Linnick MN and Fasel HF (2005) A high-order immersed interface method for simulating unsteady incompressible flows on irregular domains. *Journal of Computational Physics* **204**(1): 157–192.
- Liu S, Pan W, Zhao X et al. (2018) Influence of surrounding buildings on wind flow around a building predicted by CFD simulations. *Building and Environment* **140**: 1–10.
- Markatos NC (1986) The mathematical modelling of turbulent flows. *Applied Mathematical Modelling* **10**(3): 190–220.
- Mittal R and Iaccarino G (2005) Immersed boundary methods. *Annual Review of Fluid Mechanics* **37**: 239–261.
- Mochida A, Tominaga Y, Murakami S et al. (2002) Comparison of various $k-\epsilon$ models and DSM applied to flow around a high-rise building-report on AIJ cooperative project for CFD prediction of wind environment. *Wind and Structures: An International Journal* **5**(2–4): 227–244.
- Monaghan JJ (1985) Particle methods for hydrodynamics. *Computer Physics Report* **3**(2): 71–124.
- Mons RF and Sforza PM (1968) *The Three-Dimensional Wake Behind an Obstacle on a Flat Plate*. Department of Aerospace Engineering and Applied Mechanics, Polytechnic Institute of Brooklyn, New York, NY, USA, AFOSR Scientific Report, OSR 68-1666, PIBAL Report No. 68-20.
- Murakami S (1997) Current status and future trends in computational wind engineering. *Journal of Wind Engineering and Industrial Aerodynamics* **67–68**: 3–34.
- Murakami S, Mochida A and Hibi K (1987) Three-dimensional numerical-simulation of air-flow around a cubic model by means

- of large eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics* **25**(3): 291–305.
- Nägeli W (1953) Untersuchungen über die Windverhältnisse im Bereich von Schilfrohrwänden. *Mitteilungen der Schweizerischen Anstalt für das Forstliche Versuchswesen* **29**(2): 213–266 (in German).
- Parente A, Gorle C, van Beck J and Benocci C (2011a) Improved k - ϵ model and wall function formulation for the RANS simulation of neutral ABL flows. *Journal of Wind Engineering and Industrial Aerodynamics* **99**(4): 267–278.
- Parente A, Gorle C, van Beck J and Benocci C (2011b) A comprehensive modelling approach for the neutral atmospheric boundary layer: consistent inflow conditions, wall function and turbulence model. *Boundary Layer Meteorology* **140**(3): 411–428.
- Pasquill F and Smith FB (1983) *Atmospheric Diffusion*. Ellis Horwood, (John Wiley & Sons), Chichester, UK.
- Piomelli U (2014) Large eddy simulations in 2030 and beyond. *Philosophical Transactions of the Royal Society A* **372**: 1–13, <https://doi.org/10.1098/rsta.2013.0320>.
- Plate EJ and Lin CY (1965) *The Velocity Field Downstream From a Two Dimensional Model Hill*. Fluid Dynamics and Diffusion Laboratory, Department of Civil Engineering, Colorado State University, Fort Collins, CO, USA, Rep. CER 65 EJP 14.
- Ploumhans P, Winckelmans GS, Salmon JK, Leonard A and Warren MS (2002) Vortex methods for direct numerical simulation of three dimensional bluff body flows: applications to the sphere at $Re=300$, 500 and 1000. *Journal of Computational Physics* **178**(2): 427–463.
- Pope SB (2000) *Turbulent Flows*. Cambridge University Press, Cambridge, UK.
- Pope SB (2004) Ten questions concerning the large eddy simulation of turbulent flows. *New Journal of Physics* **6**(1): 35.
- Ramponi R, Blocken B, de Coo LB and Janssen WD (2015) CFD simulation of outdoor ventilation of generic urban configurations with different urban densities and equal and unequal street widths. *Building and Environment* **92**: 152–166.
- Ricci A, Kalkman I, Blocken B et al. (2017b) Local-scale forcing effects on wind flows in an urban environment: impact of geometrical simplifications. *Journal of Wind Engineering and Industrial Aerodynamics* **170**: 238–255.
- Richards PJ and Hoxey RP (1993) Appropriate boundary conditions for computational wind engineering models using the k - ϵ turbulence model. *Journal of Wind Engineering and Industrial Aerodynamics* **46–47**: 145–153.
- Robins A, Carruthers D and McHugh C (1997) The ADMS building effects module. *International Journal of Environment and Pollution* **8**(3–6): 708–717.
- Rossinelli D, Bergdorf M, Cottet GH and Koumoutsakos P (2010) GPU accelerated simulations of bluff body flows using vortex particle methods. *Journal of Computational Physics* **229**(9): 3316–3333.
- Sada K and Sato A (2002) Numerical calculation of flow and stack-gas concentration fluctuation around a cubical building. *Atmospheric Environment* **36**(35): 5527–5534.
- Salim SM (2011) *Computational Study of Wind Flow and Pollutant Dispersion Near Tree Canopies*. PhD thesis, Division of Environment, University of Nottingham Malaysia Campus, Semenyih, Malaysia.
- Schiestel R and Dejoan A (2005) Towards a new partially integrated transport model for coarse grid and unsteady turbulent flow simulations. *Theoretical and Computational Fluid Dynamics* **18**(6): 443–468.
- Sforza PM and Mons RF (1970) Wall-Wake: flow behind a leading edge obstacle. *American Institute of Aeronautics and Astronautics* **8**(12): 2162–2167.
- Shih T, Liou W, Shabbir A and Zhun J (1995) A new k - ϵ eddy viscosity model for high Reynolds number turbulent flows – model development and validation. *Computers and Fluids* **24**(3): 227–238.
- Shirasawa T, Tominaga T, Yoshie R et al. (2003) Development of CFD method for predicting wind environment around a high-rise building: Part 2: the cross comparison of CFD results using various k - ϵ models for the flowfield around a building model with 4:4:1 shape. *AIJ Journal of Technology and Design* **9**(18): 169–174.
- Smith PA and Stansby PK (1988) Impulsively started flow around a circular cylinder by the vortex method. *Journal of Fluid Mechanics* **194**: 45–77.
- Smith FT, Sykes RI and Brighton PWM (1977) A two-dimensional layer encountering a three dimensional hump. *Journal of Fluid Mechanics* **83**(1): 163–176.
- Sotiropoulos F and Yang X (2014) Immersed boundary methods for simulating fluid–structure interaction. *Progress in Aerospace Sciences* **65**: 1–21.
- Spalart PR (2000) Strategies for turbulence modelling and simulations. *International Journal Heat and Fluid Flow* **21**(3): 252–263.
- Spalart PR, Jou W, Strelets M and Allmaras SR (1997) Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach. In *Advances in DNS/LES* (Liu C and Liu Z (eds)). Greyden Press, Columbus, USA, pp. 137–147.
- Speziale CG (1998) Turbulence modeling for time-dependent RANS and VLES: a review. *American Institute of Aeronautics and Astronautics* **36**(2): 173–184.
- Temmerman L, Hadziabdic M, Leschziner MA and Hanjalic K (2005) A hybrid two-layer URANS-LES approach for large eddy simulation at high Reynolds numbers. *International Journal Heat and Fluid Flow* **26**(2): 173–190.
- Tessicini F, Temmerman L and Leschziner MA (2006) Approximate near-wall treatments based on zonal and hybrid RANS-LES methods for LES at high Reynolds numbers. *International Journal Heat and Fluid Flow* **27**(5): 789–799.
- Thaker P and Gokhale S (2016) The impact of traffic-flow patterns on air quality in urban street canyons. *Environmental Pollution* **208** (Part A): 161–169.
- Tominaga Y and Stathopoulos T (2007) Turbulent Schmidt numbers for CFD analysis with various types of flow field. *Atmospheric Environment* **41**(37): 8091–8099.
- Tominaga Y and Stathopoulos T (2009) Numerical simulation of dispersion around an isolated cubic building: comparison of various types of k - ϵ models. *Atmospheric Environment* **43**(20): 3200–3210.
- Tominaga Y and Stathopoulos T (2012) CFD modeling of pollution dispersion in building array: evaluation of turbulent scalar flux modeling in RANS model using LES results. *Journal of Wind Engineering and Industrial Aerodynamics* **104–106**: 484–491.
- Tominaga Y and Stathopoulos T (2013) CFD simulation of near-field pollutant dispersion in the urban environment: a review of current modeling techniques. *Atmospheric Environment* **79**: 716–730.
- Tominaga Y, Mochida A, Yoshie R et al. (2008) AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics* **96**(10–11): 1749–1761.
- Toparlar Y, Blocken B, Maiheu B and van Heijst GJF (2017) A review on the CFD analysis of urban microclimate. *Renewable and Sustainable Energy Reviews* **80**: 1613–1640.

- Travin A, Shur M, Strelets M and Spalart P (2000) Detached-eddy simulations past a circular cylinder. *Flow Turbulence and Combustion* **63**(1–4): 293–313.
- Trias FX, Gorobets A, Verstappen RWCP, Soria M and Oliva A (2010) Turbulent flow around a wall-mounted cube: direct numerical simulation and regularization modelling. In *Parallel Computational Fluid Dynamics: Recent Advances and Future Directions* (Biswas R (ed.)). DEStech Publication Inc., Lancaster, PA, USA, pp. 483–496.
- Tu J, Yeoh GH and Lin C (2018) *Computational Fluid Dynamics: A Practical Approach*. Butterworth-Heinemann, Oxford, UK.
- Uhlmann M (2005) An immersed boundary method with direct forcing for the simulation of particulate flows. *Journal of Computational Physics* **209**(2): 448–476.
- Wang S and Zhang X (2011) An immersed boundary method based on discrete stream function formulation for two- and three-dimensional incompressible flows. *Journal of Computational Physics* **230**(9): 3479–3499.
- Woo HGC, Peterka JA and Cermak JE (1976) *Wind-Tunnel Measurements in the Wakes of Structures*. Fluid Dynamics and Diffusion Laboratory, Department of Civil Engineering, Colorado State University, Fort Collins, CO, USA, Rep. CER 75-76 HGCW-JAP. JEC40.
- Wu JC (1976) Numerical boundary conditions for viscous flow problems. *American Institute of Aeronautics and Astronautics* **14**(8): 1042–1049.
- Yakhot V, Orszag SA, Thangam S, Gatski TB and Speziale CG (1992) Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids* **4**(7): 1510–1520.
- Yang ZY (2015) Large-eddy simulation: past, present and the future. *Chinese Journal of Aeronautics* **28**(1): 11–24.
- Yang X, Zhang X, Li Z and He GW (2009a) A smoothing technique for discrete delta functions with application to immersed boundary method in moving boundary simulations. *Journal of Computational Physics* **228**(20): 7821–7836.
- Yang Y, Gu M, Chen S and Jin X (2009b) New inflow boundary conditions for modelling the neutral equilibrium atmospheric boundary layer in computational wind engineering. *Journal of Wind Engineering and Industrial Aerodynamics* **97**(2): 88–95.
- Yoshie R, Mochida A, Tominaga Y et al. (2007a) Cooperative project for CFD prediction of pedestrian wind environment in the Architectural Institute of Japan. *Journal of Wind Engineering and Industrial Aerodynamics* **95**(9–11): 1551–1578.
- Yoshie R, Tanaka H and Shirasawa T (2007b) Technique for simultaneous measurement of fluctuating concentration, velocity and temperature in non-isothermal flow. *Proceedings of the 12th International Conference on Wind Engineering, Cairns, Australia*, pp. 1399–1406.
- Zhou Z, Chen A and Xiang H (2002) Numerical assessment of aerodynamic derivatives and critical wind speed of flutter of bridge decks by discrete vortex method. *Journal of Vibration Engineering and Technologies* **15**(3): 327–331.

How can you contribute?

To discuss this paper, please email up to 500 words to the editor at journals@ice.org.uk. Your contribution will be forwarded to the author(s) for a reply and, if considered appropriate by the editorial board, it will be published as discussion in a future issue of the journal.

Proceedings journals rely entirely on contributions from the civil engineering profession (and allied disciplines). Information about how to submit your paper online is available at www.icevirtuallibrary.com/page/authors, where you will also find detailed author guidelines.