Since January 2020 Elsevier has created a COVID-19 resource centre with free information in English and Mandarin on the novel coronavirus COVID-19. The COVID-19 resource centre is hosted on Elsevier Connect, the company’s public news and information website.

Elsevier hereby grants permission to make all its COVID-19-related research that is available on the COVID-19 resource centre - including this research content - immediately available in PubMed Central and other publicly funded repositories, such as the WHO COVID database with rights for unrestricted research re-use and analyses in any form or by any means with acknowledgement of the original source. These permissions are granted for free by Elsevier for as long as the COVID-19 resource centre remains active.
New inflow boundary conditions for modeling twisted wind profiles in CFD simulation for evaluating the pedestrian-level wind field near an isolated building

A.U. Weerasuriya\textsuperscript{a}, Z.Z. Hu\textsuperscript{b,c}, X.L. Zhang\textsuperscript{a}, K.T. Tse\textsuperscript{a}, S. Li\textsuperscript{d,e}, P.W. Chan\textsuperscript{d}

\textsuperscript{a} Department of Civil and Environmental Engineering, Hong Kong University of Science and Technology, Clear Water Bay, Kowloon, Hong Kong
\textsuperscript{b} Department of Civil Engineering, Tsinghua University, Beijing, PR China
\textsuperscript{c} Division of Ocean Science and Technology, Graduate School at Shenzhen, Tsinghua University, Shenzhen, PR China
\textsuperscript{d} Hong Kong Observatory, Kowloon, Hong Kong

\textbf{ARTICLE INFO}

Keywords:
CFD
Isolated building
Pedestrian level wind environment
Twisted wind profiles
Wind tunnel test

\textbf{ABSTRACT}

The hilly topography of Hong Kong influences oncoming winds and gradually changes their wind directions along the profiles' height. The vertical variation in wind directions, or the twist effect, significantly influences the Pedestrian Level Wind (PLW) field in urban areas of Hong Kong, thus it is a topic demanding systematic investigations. In this study, a new set of inflow boundary conditions are proposed to model twisted wind flows in Computational Fluid Dynamic (CFD) simulations. The new inflow boundary condition derived based on the horizontal homogeneous assumption, specifies a vertical profile of lateral wind speeds at the inlet boundary to sustain the twist effect in the empty computational domain. The proposed boundary conditions are used to simulate the PLW fields near three isolated buildings with different Height-to-Width ratio using two CFD codes; OpenFOAM, and FLUENT. The results reveal that OpenFOAM is more reliable in simulating PLW fields in twisted wind flows using the new set of boundary conditions. The three-dimensional flow field provided by the OpenFOAM simulation shows sparse streamlines downstream the buildings, indicating lack of organized eddies in the building far wake, which negatively affects the dispersion of air pollutants in twisted winds.

\section{1. Introduction}

As both wealth and population amass in urban centers, the land becomes scarce and high-rise buildings fill up cityscapes [1]. High-rise buildings are a major cause of wind nuisance at the pedestrian level: on the one hand, pedestrians could be in danger when exposed to intense downwash flows and sudden wind gusts induced by the high-rise buildings [2–6]. On the other hand, when spaced closely together, high-rise buildings obstruct natural breezeways, subsequently limiting the near-ground air circulation, potentially leading to undesirable low wind speed areas between buildings that form a cluster [7,8]. In a tropical metropolis such as Hong Kong, low wind speeds in urban areas may pose a serious threat to public health [9]; if winds do not have high enough speed, they cannot help regulating the body temperatures of pedestrians. Poorly regulated body temperatures may rise, and once reaching a critical level, they may result in heat stroke or other medical conditions [10]. Also, low winds speeds cannot effectively carry air pollutants away and disperse them from the street level, resulting in poor air quality in city centers [1]. Unimproved poor air quality amplifies the risks of an epidemic such as the outbreak of the Severe Acute Respiratory Syndrome (SARS) in 2003. SARS, having claimed many lives, prompted the government of Hong Kong and local scholars to appraise the state and quality of outdoor air ventilation. Their combined effort has led to the introduction of the important protocol named Air Ventilation Assessment (AVA), whose aim is to prevent any further deterioration of the air quality in Hong Kong. The AVA demands that any land developments and redevelopments in Hong Kong need to pass a wind tunnel test-based or Computational Fluid Dynamic simulation-based assessment to ensure that they maintain a minimum wind speed at the pedestrian level no less than 1.5 m s\textsuperscript{-1}. Given the importance of the Pedestrian Level Wind (PLW) environment in modern cities such as Hong Kong, many studies have focused not only on achieving acceptable wind environments for pedestrians but also on the factors that affect the wind conditions in built-up areas.

The buildings in a built-up neighborhood are not the only factor that affects the PLW field in that area. Several other factors such as upstream terrain, atmospheric stability, and anthropogenic heat emissions may also have significant influence on the PLW field. For example, the hilly
terrain in Hong Kong is a major factor that governs the local wind climate [11]. Moreover, the properties of wind flows including velocity, direction, and turbulence modified by the hilly terrain should be considered when calculating wind loads on structures [12] and assessing the urban wind environment in Hong Kong [11]. Tse et al. [11] revealed how the hilly terrain of Hong Kong causes large variations in wind directions in a wind profile, subsequently modifying the PLW fields around both isolated buildings and arrays of buildings. Directional variations in a wind profile occur when winds detour as they are passing over hilly terrains. The influence of a hill on winds is maximum near the surface of the hill, gradually diminishing with height [13,14].

A wind profile containing wind directions that vary with height has a spiral shape, thus giving the profile the commonly seen term “twisted wind profile”; contrary to a twisted wind profile, a conventional wind profile has constant wind direction all along the profile’s height. How much wind direction varies—also known as wind twist effect—at a given height z, can be expressed as yaw angle (θ) by Equation (1).

$$\theta = \arctan \left( \frac{v}{u} \right)$$  

(1)

In Equation (1), v is the lateral wind velocity and u is the longitudinal wind velocity at height z.

Given that most urban areas in Hong Kong are located near mountains, and yaw angles have been observed to reach as much as 40°, Tse et al. [11] proposed to employ twisted wind profiles in wind tunnel tests when the urban PLW field of Hong Kong is concerned. To follow up on this, Tse et al. [15,16] conducted a series of wind tunnel tests to investigate the PLW fields around isolated buildings and arrays of buildings with different dimensions. Their results [15] revealed noticeable flow modifications in the PLW field under the influence of twisted wind flows. In a twisted wind flow, for example, the building’s far wake was observed to deviate from the centerline of the building, and asymmetric corner streams and a new low wind speed zone attached to the wall on the left side of the building were observed. Moreover, in a twisted wind flow, the PLW fields around arrays of buildings showed noticeable modifications: for example, dissimilar wind speeds were found inside passages between the buildings [16]. Although these studies have shed light on how twisted winds influence the PLW fields around both an isolated building and arrays of buildings, details of the flow fields have been obscured. This shortcoming is partly because of the data of the three-dimensional flow field were not obtained from the wind tunnel tests [15]. Without three-dimensional flow field data, Tse et al. [15,16] made assumptions on possible flow mechanisms to explain the modifications in the PLW field caused by the twist effect, but the explanation lacks the support of solid evidence such as simultaneous measurements of wind speed and flow field in the PLW field. One of the main objectives of this paper is, therefore, to reevaluate the explanation proposed by Tse et al. [15] in a more direct and illustrative way using the Computational Fluid Dynamics (CFD) technique.

Since the late 1980s, CFD techniques have been applied to investigate PLW fields in built-up areas [17]. The main advantage of CFD simulation is its ability to provide simultaneous data on both wind speed and flow circulation at the pedestrian level, avoiding the hassle of the two-step approach common in wind-tunnel tests [18]. CFD simulations also have proven accuracy in simulating PLW fields around isolated buildings [19–21], around arrays of buildings [22–24], in idealized city models [25–27], and in real urban areas [28–30]. All these studies highlighted the importance of proper inflow conditions for accurate results from CFD simulations. Improper inflow boundary conditions, which likely create a non-equilibrium boundary layer, would cause a CFD simulation to deviate from reality [31]. Given the importance of an equilibrium boundary layer in CFD simulations of atmospheric flows, several sets of inlet boundary conditions have been proposed for Reynolds Average Naiver Stokes (RANS) simulations, particularly for the standard k-ε turbulence model [32–35]. In the derivation of profiles of u, k and ε for the equilibrium boundary layer, the commonly used shear stress model eventually leads to that u is a logarithmic function of heights [36]. Juretic and Kozmar discussed the numerical set-ups to run a CFD simulation with vertically decreasing turbulent shear stresses [37]; Yan et al. investigated the specification of k and ε which could create an equilibrium boundary layer with an arbitrary shear stress model [36]. Recently, these inlet boundary conditions have been extended to cover other k-ε turbulence models [38] and the SST k-ω model (Shear Stress Transport k-ω model) [39]. Other than the inlet boundary condition, the consistent top boundary condition is discussed by O’Sullivan et al. [39], which facilitate the generation of the equilibrium and neutral boundary layer. In addition to the proposal of inlet boundary conditions, modifications are made to the two-equation turbulence model to create the equilibrium boundary layer [34,35,40]. Besides the specification of the boundary conditions and modifications to the turbulence models, dynamic approaches are also suggested, which include balanced forcing terms in the CFD simulations to produce an equilibrium boundary layer [41]. This study aims to propose a new set of inflow boundary conditions to produce an equilibrium boundary layer with twisted wind profiles.

In the present study, CFD techniques are employed to investigate the PLW field around an isolated building, focusing on the flow modifications induced by the twist effect and flow mechanisms driving such modifications. In order to accurately and reliably simulate the PLW field in the twisted wind flow, it is necessary to explore the inlet boundary conditions, which could sustain a twisted wind profile in a CFD simulation. Although plenty of studies have been conducted to provide inlet boundary conditions in the simulation of the atmospheric boundary layer flow, no systematic investigations, to the best knowledge of the authors, have been carried out to show the sustainability of the twisted wind profiles in the CFD computational domain. The present study therefore carries the first attempt to include the twist effect in a CFD simulation of the atmospheric boundary layer flow. Furthermore, the numerical simulation of the PLW field around an isolated building reveals the three-dimensional flow field in the twisted wind flow in the vicinity of the building at the pedestrian level and above, which helps identify mechanisms leading to the flow modifications in the twisted flows observed from a corresponding wind-tunnel test. The findings made in examining the CFD simulation results, in turn, could provide suggestions for the implementation of the AVA, and hence for the urban planning of a metropolitan in tropical or subtropical zone, such as Hong Kong.

Section 2 of this paper presents the derivation of the new set of inflow boundary conditions, whose sustainability has been evaluated in an empty domain. Section 3 describes the wind-tunnel tests whose data have been used to verify the CFD simulations of the PLW fields influenced by wind twist effects. Section 4 presents the numerical settings used to run the CFD simulations and discusses the simulated PLW fields, putting particular focus on (a) the comparison between results of the CFD simulations and data from wind-tunnel tests, and (b) details of the flow field leading to PLW field modifications. Section 5 offers some concluding remarks.

2. New inlet boundary conditions for twisted wind flows

2.1. Derivation of the inlet boundary conditions

The new inflow boundary conditions are developed based on the RANS equations, in which the turbulent diffusivity is estimated according to the standard k-ε model [42]. It is noted that the variants of the standard k-ε model have been widely used in CFD simulations of atmospheric flows, and therefore the proposed inflow boundary conditions provide a good starting point for developing similar inlet boundary conditions for other turbulence models. It is crucial that the flow conditions specified at the inlet should be maintained throughout the computational domain or until they reach the object of interest. In
Table 1
Boundary conditions for the computational domain.

| Location | Type       | Profiles/conditions                                      |
|----------|------------|---------------------------------------------------------|
| Inlet    | Velocity   | $u(z) = \frac{u}{v} \left( \frac{z}{z_0} \right)$, $v(z) = C_{10} u(z) + C_{12}$, standard $k-\varepsilon$ model; $k(z) = C_{3}(z) + C_{30}$. |
| Outlet   | Outflow    | $z(z) = \sqrt{C_{5} k(z)} \left( \frac{u_{*}}{v} \right)^2 + \left( \frac{u_{*}}{v} \right)^2 |
| Right    | Velocity   | $u(z) = \frac{u}{v} \left( \frac{z}{z_0} \right)$, $v(z) = C_{10} u(z) + C_{12}$, standard $k-\varepsilon$ model; $k(z) = C_{3}(z) + C_{30}$. |
| Left     | Velocity   | $u(z) = \frac{u}{v} \left( \frac{z}{z_0} \right)$, $v(z) = C_{10} u(z) + C_{12}$, standard $k-\varepsilon$ model; $k(z) = C_{3}(z) + C_{30}$. |
| Top      | Free slip  | $w = 0$, $u_{*} = u, v, k, z = 0$                        |
| Ground   | Wall       | Rough wall modification with roughness height $K_s = 0.00032$ m and roughness constant $C_F = 0.5$. |

**Fig. 1.** Inflow profiles of $u$, $v$ and $k$ measured from the wind tunnel tests and employed in the CFD simulations corresponding to the TWP13 profile.

Other words, the flow entering the computational domain through the inlet boundary should be in a horizontal homogeneous state. Under this horizontal homogeneous assumption ($\partial u/\partial x = 0$ and $\partial v/\partial y = 0$) and that assumption that $w = 0$, which has also been made as shown in the previous publications [32,33], the RANS equation and the transportation equations of turbulent kinetic energy ($k$) are reduced to Equations (2)–(4). In the equations, $K$ is the vertical turbulent diffusivity, $u$ is the longitudinal wind velocity, $v$ is the lateral wind velocity, $w$ is the vertical wind velocity, $z$ is the vertical coordinate, and $k$ is the turbulent kinetic energy. The transportation equation of $k$ should include the generation and dissipation of the turbulent kinetic energy, which are absent from Equation (4). In fact, the local equilibrium assumption, which is expressed in Equation (5), is implied in the derivation of Equation (4) to formulate a close set of equations. The generation of turbulence, which can only originate from the shear of the mean flow, is assumed equal to the dissipation rate of the turbulent kinetic energy ($\varepsilon$) as shown in Equation (5). Other publications concerning the creation of equilibrium boundary layer in CFD simulations derived similar sets of governing equations for various turbulence models. It should be noted that in the derivations shown in the works of Yang et al. [31,33], Richards and Hoxey [32], the horizontal homogeneous assumption and the shear stress model unavoidably leads to the logarithmic shape of $u$, and hence is not explicitly included in the derivation.

\[
\frac{\partial}{\partial z} \left( K \frac{\partial u}{\partial z} \right) = 0 \tag{2}
\]

\[
\frac{\partial}{\partial z} \left( K \frac{\partial v}{\partial z} \right) = 0 \tag{3}
\]

\[
\frac{\partial}{\partial z} \left( K \frac{\partial k}{\partial z} \right) \tag{4}
\]

\[
\varepsilon = G_k = \left( \frac{\partial u}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial z} \right)^2 \tag{5}
\]

Within the framework of the standard $k-\varepsilon$ model, the turbulent diffusivity $K$ is computed as [32],

\[
K = C_F \frac{k^2}{\varepsilon} \tag{6}
\]

and $C_F$ is a model constant. A similar set of equations have been derived in previous studies focusing on the sustainable inlet boundary conditions [31–33]. In contrast to the conventional CFD applications, in which the lateral wind velocity is often assumed zero at the inlet, the CFD simulation of twisted flows has the condition $v(z) \neq 0$. Hence,
Equation (3) should be included in the derivation of the sustainable profiles of \( u, k \) and \( \varepsilon \). Under the condition that the vertical variation of the longitudinal wind velocity, i.e., \( u(z) \), has been found from solving Equation (1) \[31–33\], Equations (3)–(5) stipulate a set of inlet boundary conditions for the standard \(-k_\varepsilon\) model, in which the model constant \( \sigma_1 = \sqrt{\varepsilon C_k} \), \( C_{k_1} = + v z C u z \) \( C_{k_2} \) \( (\frac{\partial u}{\partial z})^2 \) \( (\frac{\partial v}{\partial z})^2 \) and \( (\frac{\partial \varepsilon}{\partial z})^2 \) \( \sqrt{\varepsilon C_k} \) \( (\frac{\partial u}{\partial z})^2 \) \( (\frac{\partial v}{\partial z})^2 \) \( (\frac{\partial \varepsilon}{\partial z})^2 \). (9)

While Equations (8) and (9) compute vertical profiles of \( k \) and \( \varepsilon \), equation (7) computes the vertical profile of \( v \) according to the profile of \( u \). Under the horizontal homogeneous assumption, the logarithmic function of heights is suggested to model the \( u \) profile \[31–33\]. According to Equation (7), the \( v \) profile can also be described by a logarithmic function of heights if the \( u \) profile is logarithmic, and the influence of ground frictions, shown by the aerodynamic roughness length \( (z_0) \), is included in both the profiles of \( u \) and \( v \). In other words, the shears in the profiles of \( u \) and \( v \) are majorly induced by the ground friction and marginally induced by the vertical variation of wind directions if the logarithmic function is adopted to model the \( u \) profile.

As illustrated in the derivation, Equations (7)–(9) are the analytical solutions to the RANS equations and the transportation equation of \( k \) under a series of assumptions. The transportation equation of \( \varepsilon \) is, however, not satisfied when \( v, k \) and \( \varepsilon \) are specified according to Equations (7)–(9). In fact, the residual of the transportation equation of \( \varepsilon \) is controlled by the values of the constants \( C_{k_1} \) and \( C_{k_2} \), and the influence of the residual on the sustainability of the inlet profiles is limited when turbulence intensities of the simulated wind field are within a reasonable range \( (<50\%) \).

2.2. Evaluation of the inlet boundary conditions

The horizontal homogeneity of the proposed inflow boundary condition is evaluated in an empty domain using two CFD software; OpenFOAM and FLUENT. OpenFOAM is an open-source CFD simulation tool that has gradually gained popularity by virtue of its flexibility in adjusting the code for various purposes \[43\]. FLUENT is one of the widely adopted commercial CFD codes used by researchers and engineers because of its user-friendly interface and embedded numerical models for different types of simulations. However, the lack of
flexibility in modulating various auxiliary models is a definite drawback of FLUENT [44].

An empty computation domain is simulated in OpenFOAM and FLUENT, whose dimensions are 4.052.71.35 m in length, width, and height, respectively. The domain dimensions are specified to simulate the flow at the wind-tunnel scale. The mesh of the computational domain grows in sizes in the longitudinal and lateral directions from the center of the domain to mimic a simulation with the object of interest located in the center of the domain. In the vertical direction, finer grids appear close to the bottom boundary and gradually expands in the

![Figure 4](image1.png)

Fig. 4. Two wooden vanes systems: (a) with the maximum guide angle of 15°; (b) with the maximum guide angle of 30°.

![Figure 5](image2.png)

Fig. 5. (a) Experimental set-up in the wind tunnel of WWTF and (b) Irwin sensor arrangement in the measurement area.

| Building | Dimensions (mm) | Aspect ratio (H/W) | Remarks               |
|----------|-----------------|-------------------|-----------------------|
| S1       | 600 150 100     | 4:1               | Tall and slender      |
| S2       | 225 150 100     | 1.5:1             | Intermediate         |
| S3       | 225 450 100     | 0.5:1             | Short and wide       |

Table 2
Dimensions of the building models.

![Figure 6](image3.png)

Fig. 6. Dimensions of the computational domain.
The mesh contains 200, 40 and 100 cells in longitudinal, lateral and vertical directions for both simulations, which make the total number of cells reaching 0.8 million.

Table 1 lists the boundary conditions used for the simulations of the empty domain. It should be noted that the CFD codes, OpenFOAM and FLUENT, share a set of boundary conditions and other numerical settings. In particular, the proposed inlet boundary conditions are employed to specify the profiles of $u$, $v$, $k$ and $\varepsilon$ at the inlet and lateral boundaries of the OpenFOAM and FLUENT simulations. The parameters to be determined in Equations (7)–(9) and the profile of wind velocities are specified according to the two twisted wind profiles simulated by Tse et al. [11] in a wind tunnel test. The two twisted wind profiles have the maximum yaw angle ($\theta^\circ$) of 13° and 22° at the lowest measurement height of 0.01 m and are hereafter referred to as TWP13 and TWP22. Fig. 1 shows the profiles measured from the wind tunnel tests and the profiles used to specify the inlet boundary conditions for the CFD simulation.

It is evident from Fig. 1 that the profiles employed in the CFD simulations, calculated according to Equations (7)–(9), are similar to the profiles measured from the wind tunnel test. Since the profile of $u$ measured from the wind tunnel test is modulated using the logarithmic model as the target ($u^* = 0.2738 \text{ m s}^{-1}$, $z_o = 0.000012 \text{ m}$), it is reasonable that Fig. 1(a) shows agreement between the profiles of $u$ measured from the wind tunnel test and the CFD simulation. The profiles of $v$ and $k$, on the other hand, measured in the wind tunnel test are not modulated using equations (7)-(9), and therefore deviate slightly from the profiles used in the CFD simulation.

From comparing the profiles of $u$ and $v$ extracted from the CFD simulations at the center and near the outlet boundaries to the profiles specified at the inlet boundary (shown in Figs. 2 and 3), it can be discerned that the proposed inflow boundary conditions of $v$ and $k$ successfully model the twisted wind profile in both CFD simulations. In fact, there are no observable differences between the profiles of $u$ and $v$ at different locations in the empty domain. The profile of $k$, however, deviates slightly from the specification at the inlet boundary along the stream wise direction, and the maximum differences between $k$ profiles at the inlet and near the outlet boundaries are under 0.06 m$^2$/s$^2$ (<6%). The differences in the $k$ profiles presented in Figs. 2 and 3 should be attributed to the near-wall treatments employed in the OpenFOAM and FLUENT simulations. Since the sustainability of the profiles of $u$ and $v$ is the main concern of the proposed inflow boundary condition and the deviation in the $k$ profiles is within the acceptable range, both the empty domain simulations using OpenFOAM and FLUENT substantiate that the proposed inflow boundary conditions have successfully maintained the profiles of wind velocities with twist effects in the CFD simulation. Consequently, it is feasible to use the proposed inflow boundary conditions in a CFD simulation focusing on the influence of twisted wind flows on the PLW environment around an isolated building.
3. The wind-tunnel experiment

To quantitatively assess the accuracy and reliability of CFD simulations of the PLW field around an isolated building, the wind-tunnel measurements are used as criteria in a systematic evaluation of the CFD simulation results. All the corresponding wind-tunnel tests were conducted in the CLP Power Wind/Wave Tunnel Facility (WWTF) at the Hong Kong University of Science and Technology (HKUST). Two types of wooden vanes with the maximum guide angles of 15° and 30° at the ground level, as shown in Fig. 4, were installed in the low-speed section of the wind tunnel to simulate the two twisted wind profiles: TWP13 and TWP22. The building models were installed at the center of the turntable, which was 4m downstream the vane systems (Fig. 5(a)). The mean wind speeds measured at 10mm in the wind-tunnel, which corresponds to a 2m height in the full-scale according to the length scale of 1: 200, were taken as the wind speeds at the pedestrian level. The wind speeds were measured at more than 200 locations using Irwin sensors installed around the building models as shown in Fig. 5(b). In detail, the Irwin sensors covered an area of 1.218 m² with the minimum resolution of 0.075 m0.100 m near the buildings. Three building models with the dimensions as listed in Table 2 are tested in the wind tunnel to present the PLW fields around a tall and slender, a short and wide, and an intermediate isolated building. All the other details on the wind-tunnel experiments of the PLW field around the isolated building can be found in the authors’ previous publications [15,16].

4. Results and discussion

The PLW fields around the three building models; S1, S2, and S3, under the influence of the two twisted wind flows (TWP13 and TWP22) are simulated using OpenFOAM and FLUENT. In addition, a conventional wind profile, which is hereafter referred to as CWP, is adopted to conduct the CFD simulation as the control case. The CWP profile has similar wind speeds and turbulence intensities as the profiles of TWP13.
and TWP22 but contains no vertical variations in wind directions. For a given building model, the computational domain dimensions, mesh configurations, and the boundary conditions are same for both OpenFOAM and FLUENT simulations. More specifically, the building model is located at the distances of $8H$ and $6H$ away from the inlet and the sidewalls of the computational domain and are surrounded by a finer mesh close to the walls of the buildings. The dimensions of the domain and the mesh close to the building model are shown in Figs. 6.

Fig. 9. The simulated wind velocities along the lines cutting through the computational domain laterally (a) and longitudinally (b) produced by the OpenFOAM simulation. The solid lines, dashed lines, dots and crosses correspond to the QUICK, SFCD, MUSCL and UMIST schemes.
and 7. The horizontal size of the cell close to the building model is 2mm and increases to 10mm at the boundary of the core area as shown in Fig. 7(a). The heights of cells close to the ground, on the other hand, are reduced (shown in Fig. 7(b)) to ensure that there are more than 5 layers of cells within pedestrian level [45], which is 0.01m for the simulations reported in the present study. The height of cells increases along the height of the building model and reduces when approaching the building top. The height of cells near the ground and close to the building top is around 0.1mm, which makes the total number of cells exceeding 0.9 million for both OpenFOAM and FLUENT simulations. The boundary conditions used to run the simulations are identical to the cases with the empty domain, which have been summarized in Table 1. The QUICK scheme [46] is used to discretize the momentum equation in both OpenFOAM and FLUENT simulation, and the SIMPLE algorithm [47] is employed to solve the governing equations. The simulations are kept for 12000 iterations to ensure that the converged results are obtained. The simulated wind velocity at the pedestrian level along the lines laterally penetrating the computational domain \((X = −225\text{mm} \text{and } X = 225\text{mm})\) are monitored through the simulation process, and the fluctuations in the simulated wind velocities on the monitor during the last 2000 iterations for the OpenFOAM and FLUENT simulations are under the levels of 1.3% and 1.5%, respectively, which substantiated that the converged results are obtained.

4.1. Verification of CFD simulations

Before further verifying the CFD simulation results based on wind-tunnel measurements, it is crucial to check the independency of simulation results from both the meshes and numerical schemes used to discretize the momentum equation. Therefore, a finer mesh, with the total number of cells exceeding 1.7 million, is employed to run the simulation of the same domain using the same numerical setting as the simulation with coarse mesh (containing 0.9 million cells). The finer meshes are produced through reducing the horizontal sizes of cells close to the building model from 2mm to 1mm, and increasing the number of cells in the vertical direction from 72 to 92. The simulated wind velocities corresponding to the building model of S2 under the influence of the profile of TWP13 at the lines cutting through the domain laterally \((X = −225\text{mm}, X = 75\text{mm}, X = 225\text{mm}, X = 525\text{mm})\) are plotted in Fig. 8 for the fine mesh and coarse mesh. The x coordinate of the lines are determined through specifying the building center located at \(X = 0\).

It is evident from the figure that the simulation results, yielded by both the OpenFOAM and FLUENT simulations, corresponding to the fine mesh is in excellent agreement with the simulation results of the coarse mesh, which substantiates that the simulations are grid independent.

In addition to the independency of the grid, the independency of the simulation results from the numerical schemes is also checked. In detail, the SFCD scheme [48], MUSCL scheme [49] and UMIST scheme [50], all in second order, are employed to discretize the momentum equations in the OpenFOAM simulation in addition to the QUICK scheme used to run the control case. The simulated wind velocities, corresponding to different numerical schemes under investigation, along the lines cutting through the computational domain laterally (Fig. 9(a)) and longitudinally (Fig. 9(b)) are plotted. The x and y coordinates of the lines are determined through specifying the building center located at \(X = 0\) and \(Y = 0\).

It is shown in the figure that the numerical schemes hardly influences the simulated wind field as the maximum difference among the simulated wind velocities at the same location is within the range of \(−0.3\text{ms}^{-1} \sim 0.3\text{ms}^{-1}\). Such differences are considered small in an independency check of the numerical scheme, and hence it is fair to conclude that the simulation results are independent from the numerical scheme used to discretize the momentum equation.

Fig. 10 shows the distribution of wind speeds at the pedestrian level around the building model S2 as observed from the wind tunnel experiment and extracted from the CFD simulation. The results shown in Fig. 10 corresponds to the case in which the profile TWP22 attacks the building model S2 at an incident angle of 0°. The white arrows shown in Fig. 10 indicate the direction of each approaching wind flow at the pedestrian level. As observed from Fig. 10, the wind tunnel measurements show noticeable modifications in the PLW field around the building model, including dissimilar corner streams, and the deviation of the far wake as reported by Tse et al. [15]. As it can be seen from Fig. 10(b) and (c) that both OpenFOAM and FLUENT acceptably replicate the flow modifications observed in wind tunnel experiments. There are, however, deviations in the PLW field when comparing Fig. 10(b) and (c) to Fig. 10(a). For example, on the leeward side of the building, CFD simulations tend to underestimate wind speeds. Such a discrepancy may be attributed to, first, the low accuracy of Irwin sensors in measuring wind speeds less than 2m/s [8], and, second, a well-known drawback of CFD simulations in predicting wind conditions in
the building wake [51].

In addition to the overall comparisons, Fig. 11 shows wind speed variations at the pedestrian level in the wind flows of CWP and TWP13 along four lines parallel to the Y-axis. One of the four lines is at a 0.225m distance upstream from the building center and the rest of the lines are at 0.075m, 0.225m and 0.525m distances downstream from the building center. As shown in Fig. 11(a) that both OpenFOAM and FLUENT predict similar variations in wind speeds as observed in the wind tunnel experiment despite underestimations of wind speeds in the wake of the building. Particularly, the mean wind speeds yielded from the CFD simulations are considerably smaller in an area bounded by the lines of $X = 0$, $X = 0.525m$ and the lines of $Y = -2W$, $Y = 2W$ downstream from the building ($W$ is the width of the building). Within this area, the mean wind speeds extracted form OpenFOAM and FLUENT simulation results are, on average, 37.4% and 24.8% smaller than the measurements from the wind tunnel test. Outside this area, the CFD simulation results are in a better agreement with the wind-tunnel measurements as the discrepancies reduced to, on average, 7.8% and 10.7% for the OpenFOAM and FLUENT simulations, respectively. Similarly, Fig. 11(b) shows that the wind speeds predicted by CFD simulations in the TWP13 flow deviate from the wind-tunnel measurements, particularly in the wake of the building S2. The differences in wind speeds between the CFD simulations of OpenFOAM and FLUENT and the wind tunnel experiment are about 27.5% and 11.2% in the TWP13 case. The largest difference in wind speeds is, however, observed away from the line $Y = 0$ for the TWP13 case, in contrast to the results corresponding to the profile of CWP, where the largest difference is always found exactly on the line $Y = 0$. The different locations of the largest
difference are related to the location of the Downstream Far-field Low Wind Speed (DFLWS) zone, which is deviated clockwise away from the line of \( Y = 0 \) in the TWP13 flow.

Similarly, the simulated mean wind speeds are extracted along the lines parallel to the \( X \)-axis and compared with the corresponding wind-tunnel measurements in Fig. 12. Fig. 12(a) reveals that the results of FLUENT simulation overestimate PLW speeds by about 20\% in the CWP wind flow compared to the wind tunnel measurements. The predictions of the OpenFOAM simulation, on the other hand, only moderately deviate from the wind-tunnel measurements. For instance, the deviation of OpenFOAM simulation results from wind-tunnel measurements is less than 10\% within the area bounded by the lines of \( X = 0 \) and \( X = 0.2 \) m. In the TWP13 flow, as shown in Fig. 12(b), FLUENT moderately overestimates the wind speeds, while the OpenFOAM simulation produces wind speeds comparable to the data of wind tunnel experiment along the line of \( Y = 0.2 \) m. The larger difference in wind speeds between the predictions of CFD simulations and the measurements of the wind tunnel experiment is related to the deviation of the DFLWS zone towards the line of \( Y = -0.2 \) m. Outside the DFLWS zone, such as along the two lines \( Y = -0.4 \) m and \( Y = 0.4 \) m, the results of CFD simulations are in a better agreement with the wind tunnel experiments. More specifically, the OpenFOAM simulation produces similar wind conditions.

![Fig. 12. The comparison of mean wind speeds at the pedestrian-level near building S2 from OpenFOAM (solid line), FLUENT (dashed line) and wind tunnel experiment (star dot) in (a) CWP, and (b) TWP13.](image-url)
speeds as measured from the wind tunnel experiment while the wind speeds extracted from the FLUENT simulation moderately deviate from the corresponding wind speeds measured in the wind tunnel test.

Fig. 13 shows the longitudinal variations in wind speeds corresponding to the building models of S1 and S3 in the TWP13 wind flow. Fig. 13 (a) implies that the simulation corresponding to a tall and slender building yields results that are in better agreement with wind tunnel measurements. The OpenFOAM simulation, in particular, captures the longitudinal variations in the PLW field observed in the wind tunnel test. The results of the FLUENT simulation are, however, not as accurate as those of the OpenFOAM simulation. More specifically, the FLUENT simulated PLW field shows small deviations from the wind-tunnel measurements along the line of $Y = 0.4m$ and has large deviations along the line $Y = 0.2m$. In contrast, Fig. 13(b) shows large deviations that are found for the CFD simulations. In particular, the CFD simulations present great deviations along the line of $Y = -0.2m$ when comparing to Fig. 11(a). It is believed that the large differences presented in Fig. 11(b) are in the connection with the DFLWS zone, which spreads over a large area downstream the building S3 (see Refs. [15,16]). In addition, the PLW field produced by the CFD simulations near the line of $X = 0$ are noticeably smaller than the measurements of the wind tunnel test. The strong near-wall turbulence that reduces the accuracy of wind-tunnel measurements on the leeward side of the building S3 may be the reason for such noticeable discrepancies.

The verifications presented in Figs. 10–13 substantiate that OpenFOAM outperforms FLUENT in simulating the PLW fields around isolated buildings, except in the DFLWS zone, where the reliability of both the CFD codes and the wind-tunnel test are low. Owing to its superior

![Fig. 13. The comparison of mean wind speeds at the pedestrian-level under TWP13 from OpenFOAM (solid line), Fluent (dashed line) and wind-tunnel tests (star dot) in the cases of (a) building S1, and (b) building S3.](image-url)
performance, the results of OpenFOAM are used for the further investigations on the PLW field around an isolated building.

4.2. Three-dimensional flow field in twisted winds

A PLW field in a twisted wind flow as shown in Fig. 10 is noticeably different from that in a conventional wind flow. The latter has a symmetrical distribution of wind speeds as demonstrated by several previous studies [8,15,22] in contrary to asymmetrical distributions of wind speeds in a twisted wind. Tse et al. [15,16] assumed that the asymmetric distribution of wind speeds is a result of flow modifications induced by (a) an oblique wind attack angle at the ground level and (b) the vertical variation in wind directions. The study of Tse et al. [15] provided a tentative explanation on the flow modifications observed. Using the detailed three-dimensional wind field produced by the OpenFOAM simulation, the present study proposes a more systematic and theoretically sound explanation for the deviation of the DFLWS zone, which is considered as the most important flow modification in the twisted flow.

In this study, the DFLWS zone is defined as the area in the far-field downstream the building with the wind speeds smaller than 80% of the approaching wind speed at the pedestrian height. Fig. 14 shows the location of the DFLWS zone in the simulation cases of the building S2 in the CWP flow and the two twisted wind flows, TWP13 and TWP22. The DFLWS zone is symmetrical about the building centerline in the CWP flow (Fig. 14(a)) but deviates clockwise about the centerline in the twisted wind flows (Fig. 14b and c). The deviation is dissimilar in the TWP13 flow and in the TWP22 flow. The deviation angle ($\alpha$), as proposed by Tse et al. [15], is employed to quantitatively assess the deviation of the DFLWS zone. The angle $\alpha$, as shown in Fig. 14, lies between the building centerline and a line connecting the building center and the center of the DFLWS zone.

In the study of Tse et al. [15], the deviation of the DFLWS zone is postulated as the displacement of the impinging location on the ground behind the building. Owing to the streamlines parallel to the building centerline, the CWP flow impinges the ground exactly on the building centerline and subsequently creates the symmetric DFLWS zone. With a vertical variation in wind directions, the streamlines of the twisted wind flow are no longer parallel to the building centerline even when the incident wind direction is 0°. The oblique streamlines in the twisted wind flow pass over the building and impinge on the ground at a location deviating from the building centerline. The displacement of the impinging location, which is assessed by $\alpha$, is therefore determined by the yaw angles of the twisted wind profile at the pedestrian level and at the ground level.
the top of the building as illustrated in Fig. 15. Fig. 15 shows the streamlines in the TWP22 flow at the pedestrian level and at the top of the building. It is evident that the direction of the streamlines at the top of the building S2, whose direction is shown by the first arrow appeared clockwise, is less deviated from the building centerline (shown as the black dashed line) than the direction of the approaching wind at the pedestrian level (shown as the arrow with dashed extensions). The deviation of the DFLWS zone, indicated by $\phi$ contained in Fig. 15, is, on the other hand, in between the lines indicating the directions of the flow at the building top and at the pedestrian level. In fact, owing to the interaction of wind circulations on the vertical and horizontal planes, the deviation of the DFLWS zone is neither equal to the yaw angles at the pedestrian level nor the angle at the top of the building but has a value in between the two yaw angles.

In order to further investigate the influence of the vertical variation in wind directions, the PLW fields around the building model S2 in the CWP wind flow striking the building model at $22^\circ$ is simulated and compared to the simulation case completed using TWP22. Given that the two wind flows strike the building from the same direction at the ground level, the comparison emphasizes the influence of the vertical variation in wind directions.

Fig. 16 shows the variation in the vertical wind velocity ($w$) along two sampling lines of $Y = 0$ (line 1) and $X = 0.275m$ (line 2). It should be noted that the computational domain is rotated by $22^\circ$ in the CWP case before the simulation results are extracted along the sampling lines. The variation of $w$ along the sampling line 1, as shown in Fig. 16(a) has two peaks for the CWP flow: a narrow peak around $X = 0.1m$ and a broad peak in the range between $X = 0.17m$ and $X = 0.225m$. These peaks correspond to the two strong vertical circulations that occur next to the leeward side of the building and in the DNLLWS zone, respectively. The TWP22 case has a comparable peak near the leeward side of the building but the peak at the DFLWS zone is considerably lower than that observed for the CWP flow. The low peak of the TWP22 case evidently indicates a weaker momentum exchange in the vertical direction compared to the CWP flow. The variations in $w$ along the sampling line 2 for the CWP and TWP22 flows, as shown in Fig. 16(b), have a phase difference of about $0.1m$ in the range between $Y = -0.35m$ and $Y = 0m$. The phase difference is gradually reduced to $0.05m$ when the $Y$ coordinate approaches $0$ and eventually diminishes towards the point of $Y = 0.4m$. The phase difference implies that the PLW field in the TWP22 flow striking at $0^\circ$ is not in exact agreement with the PLW field in the CWP flow striking the building at $22^\circ$. In fact, the phase difference presented in Fig. 16(b) suggests weak vertical advections in the DFLWS zone of the TWP22 flow. As a result, the properties of the DFLWS zone such as its wind speeds, area and the location are less affected by the flow properties in the TWP22 flow at higher altitudes, and hence the deviation angle of the DFLWS zone in the twisted flow is closer to the wind attack angle at the ground level.

Fig. 17 shows the top view of the streamlines passing the building in the CWP and TWP22 flows. The most distinctive difference is the absence of the organized vertical eddies from the right part of the wake (red elliptical area) in the twisted flow. In fact, TWP22 has sparse streamlines at the right side of the wake than in the CWP flow. Consequently, the wind velocity in the TWP22 flow, especially its component in the vertical direction, is not as large as in the CWP flow on the right side of the DFLWS zone. The small vertical velocities lead to weak vertical momentum exchanges, which likely affect the air pollution dispersion in the DFLWS zone. In contrast, the organized vertical eddies in the CWP flow are advantageous for removing pollution from the pedestrian level.

5. Concluding remarks

A new set of inflow boundary conditions is proposed to sustain twisted wind profiles in the CFD simulation that focuses on evaluating the PLW fields around isolated buildings. The new inflow boundary conditions are developed assuming the horizontal homogeneity of the wind flow for the standard $k\epsilon$ turbulence model. The proposed inflow boundary conditions have satisfactory consistency and the accuracy for
modeling the twisted wind profiles with the maximum yaw angles of 13° and 22° using an opensource CFD code, OpenFOAM, and a commercial CFD code, FLUENT.

The new boundary conditions are used to specify the inlet boundary conditions in a series of CFD simulations for assessing the PLW field around three isolated buildings, a tall and slender building (H/W = 4:1), a short and wide building (H/W = 0.5:1), and an intermediate building (H/W = 0.75:1). The results of OpenFOAM and FLUENT simulations were used for the assessment of the reliability of CFD simulations on the PLW fields around isolated buildings under twisted wind profiles, and investigating the driven flow mechanisms for flow modifications reported by Tse et al. [15]. Based on the three-dimensional flow field data extracted CFD simulations, following conclusions are drawn:

1. Both OpenFOAM and FLUENT can successfully replicate the PLW fields around isolated buildings, except in the DFLWS zone, in twisted wind flows. Both CFD codes show large differences in wind speeds in the DFLWS zone compared to those measured in the wind tunnel test. The differences may be related to the accuracy of measurements of wind speeds in the DFLWS zone from wind tunnel tests and/or CFD simulations.

2. Outside the DFLWS zone, OpenFOAM has superior performance in replicating wind speed and its variation compared with FLUENT. Particularly, OpenFOAM successfully predicts the variations in wind speed in the longitudinal direction as in the wind tunnel experiment while FLUENT regularly overestimates the wind speeds.

3. The three-dimensional flow field data of twisted wind profiles depict that the deviation of DFLWS zone is resulted from the combined effect of the vertical circulation that passes over the building and the horizontal circulation that wraps the building at the pedestrian level. The vertical circulation in a twisted wind is found to be weaker than that in the conventional wind. Thus, the deviation of the DFLWS zone is more affected by the yaw angle near the ground in a twisted flow.

4. The vertical wind velocity (w) in the DFLWS zone is found to be considerably smaller in the twisted flow than that in the conventional wind flow. In particular, the TWP22 flow has sparse horizontal streamlines in the DFLWS zone, indicating the absence of the organized vertical eddies to the right side of the wake, which negatively affects the removal of air pollutants at the ground level.

Acknowledgement

Authors of this paper would like to show their gratitude to the staff of the CLP Power Wind/Wave Tunnel Facility at the Hong Kong University of Science and Technology for their support for conducting the wind tunnel experiments. The work was also supported by National Natural Science Foundation of China (Grant No. 51608302, 51778336, 51579227).

References

[1] E. Ng, Policies and technical guidelines for urban planning of high-density cities – air ventilation assessment (AVA) of Hong Kong, Build. Environ. 44 (7) (2009) 1478–1488.
[2] W.H. Melbourne, P.N. Joubert, Problems of wind flow at the base of tall buildings, Proceedings Wind Effects on Buildings and Structures, 1971.
[3] S. Murakami, K. Uehara, H. Komine, Amplification of wind speed at ground level due to construction of high-rise building in urban area, J. Wind Eng. Ind. Aerod. 4 (3–4) (1979) 343–370.
[4] A. Penwarden, A. Wise, Wind Environment Around Buildings, Her Majesty’s Stationery Office, London, 1975.
[5] N. Isyumov, A.G. Davenport, The ground level wind environment in built-up area, Proc. 4th Int. Conf. On Wind Effects on Buildings and Structures, Heathrow, 1975, pp. 403–422.
[6] J. Gandermeer, Wind environment around buildings: aerodynamic concepts, Proc., 4th Int. Conf. Wind Effects on Buildings and Structures, Heathrow, 1975, pp. 423–432.
[39] J. O’Sullivan, R. Archer, R. Flay, Consistent boundary conditions for flows within the atmospheric boundary layer, J. Wind Eng. Ind. Aerod. 99 (1) (2011) 65–77.

[40] A. Parente, C. Gorlé, J. Beeck, C. Benocci, A comprehensive modelling approach for the neutral atmospheric boundary layer: consistent inflow conditions, wall function and turbulence model, Boundary-Layer Meteorol. 140 (3) (2011) 411–428.

[41] X. Cai, Q. Huo, L. Kang, Y. Song, Equilibrium atmospheric boundary-layer flow: computational fluid dynamics simulation with balanced forces, Boundary-Layer Meteorol. 152 (3) (2014) 349–366.

[42] W. Jones, B. Launder, The prediction of laminarization with a two-equation model of turbulence, Int. J. Heat Mass Tran. 15 (2) (1972) 301–314.

[43] M. Balogh, A. Parente, C. Benocci, RANS simulation of ABL flow over complex terrains applying an Enhanced k-ε model and wall function formulation: implementation and comparison for Fluent and OpenFOAM, J. Wind Eng. Ind. Aerod. 104-106 (2012) 360–368.

[44] D. Hargreaves, N. Wright, On the use of the k-ε model in commercial CFD software to model the neutral atmospheric boundary layer, J. Wind Eng. Ind. Aerod. 95 (5) (2007) 355–369.

[45] Y. Tominaga, A. Mochida, R. Yoshie, H. Kataoka, T. Nozu, M. Yoshikawa, T. Shirasawa, “AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, J. Wind Eng. Ind. Aerod. 96 (10–11) (2008) 1749–1761.

[46] B. Leonard, A stable and accurate convective modelling procedure based on quadratic upstream interpolation, Comput. Meth. Appl. Mech. Eng. 19 (1) (1979) 59–98.

[47] S. Patankar, D. Spalding, A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows, Int. J. Heat Mass Tran. 15 (10) (1972) 1787–1806.

[48] H. Ziman, A Computer Prediction for Chemically Reacting Flows in Stirred Tanks, PhD thesis Imperial College, University of London, 1990.

[49] B. van Leer, Towards the ultimate conservative difference scheme, V. A second order sequel to Godunov’s method, J. Comput. Phys. 32 (1979) 101–136.

[50] F. Lien, M. Leschziner, Upstream monotonic interpolation for scalar transport with application to complex turbulent flows, Int. J. Numer. Meth. Fluid. 19 (6) (1994) 527–548.

[51] R. Yoshie, A. Mochida, Y. Tominaga, H. Kataoka, K. Harimoto, T. Nozu, T. Shirasawa, Cooperative project for CFD prediction of pedestrian wind environment in the Architectural Institute of Japan, J. Wind Eng. Ind. Aerod. 95 (9–11) (2007) 1551–1578.