CFD evaluation of building geometry modifications to reduce pedestrian-level wind speed

Citation for published version (APA):

Document license:
CC BY-NC-ND

DOI:
10.1016/j.buildenv.2019.106293

Document status and date:
Published: 01/10/2019

Document Version:
Publisher’s PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:

• A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher’s website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.
Link to publication

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the “Taverne” license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.

Download date: 15. Sep. 2019
CFD evaluation of building geometry modifications to reduce pedestrian-level wind speed

T. van Druenen\textsuperscript{a,*}, T. van Hooff\textsuperscript{a,b}, H. Montazeri\textsuperscript{a,b}, B. Blocken\textsuperscript{a,b}

\textsuperscript{a} Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, PO Box 513, 5600, MB Eindhoven, the Netherlands
\textsuperscript{b} Building Physics Section, Department of Civil Engineering, KU Leuven, PO Box 5500 3000, Leuven, Belgium

\section{1. Introduction}

Increased pedestrian-level wind (PLW) speed can lead to uncomfortable and even dangerous conditions. Examples are given by Wise \cite{1}, who reported about untenantable shops due to a windy environment, and by Lawson and Penwarden \cite{2}, who mentioned the death of two elderly people due to a fall caused by high wind speeds at the base of a high-rise building. An event was noted where a pedestrian was crushed to death by a lorry blown over by increased wind speed near a tall building, which made the city council ban traffic and pedestrians during strong winds \cite{3}. PLW speed in the built environment is strongly affected by buildings or groups of buildings in general, and by the presence of high-rise buildings in particular. The approaching wind flow diverges over and around the building, resulting in a substantial amount of air flowing downwards in front (windward side) of the building. This results in a standing vortex at ground level, which sweeps around the building corners, where flow separation causes the creation of high-speed wind corner streams. Together with the standing vortex, these corner streams are the highest PLW speed zones for an isolated high-rise building \cite{4}. High PLW speeds can lead to wind discomfort or even wind danger \cite{5}. For this reason, many urban authorities nowadays only provide a permit for new high-rise building after a wind engineering study has indicated that their impact on the wind comfort and safety around the new and the surrounding buildings remains within specified limits.

In most of these studies, wind comfort refers to the mechanical effects of wind on people \cite{2,5}, although thermal and other effects can also be of importance \cite{6,7}. In the literature on outdoor comfort, a main distinction can be made between two categories of studies; PLW comfort studies on the one hand, and (outdoor) thermal comfort studies on the other hand. The former generally only focus on the mechanical effects of wind, while the latter include not only wind speed but also other parameters such as air temperature, radiant temperature, relative humidity, etc. For the assessment of mechanical wind discomfort, the mean wind speed $U$ is often used as an indicator, as in the Dutch Wind Nuisance Standard NEN8100 $^8$. Although gustiness is also important \cite{9-11}, it is more difficult to obtain accurate estimates of gustiness than of mean wind speed by wind-tunnel testing or numerical simulation with Computational Fluid Dynamics (CFD). In addition, sensors explicitly designed for robust and accurate PLW studies, such as Irwin probes are primarily suited for the accurate measurement of mean wind speed rather than the full range of turbulent fluctuations \cite{12}. Actually,
this problem of accurate measurement is the reason why many comfort criteria are only based on mean wind speed and not on the fluctuations. This was an explicit reason for only including mean wind speed in the Dutch wind nuisance Standard NEN8100 [8], which, at the time of writing this article, is still the only wind nuisance standard in the world. In other comfort criteria, gustiness is taken into account by an increased mean wind-speed threshold value [8–11].

In spite of the increasing awareness of the importance of wind comfort and wind safety [13], Willemens and Wisse stated that the relation between urban geometry and PLW comfort is often unclear for designers [5]. Moreover, alterations or additions in favor of PLW comfort and wind safety are sometimes found to be visually displeasing, impractical, expensive, and their effects are rarely investigated and often appear disappointingly small [5].

There are some well-established guidelines for avoiding wind comfort or wind danger problems in the design of high-rise buildings that originated from extensive wind-tunnel and CFD research in wind engineering in the past decades. These include locating building entrances, walkways and bicycle routes away from the building corners, avoiding these items to be located in narrow passages between buildings and avoiding passages through buildings as well as the creation of recreational areas in the close vicinity of high-rise buildings, unless specific attention is given to the design of these features and the mitigation of high PLW speeds (e.g. Refs. [4,14–17]).

Several studies investigated the effect of specific geometrical modifications of the building on the PLW environment in its immediate surroundings. Murakami et al. [18] conducted wind-tunnel experiments to demonstrate how the downflow along the windward building façade can be deflected by adding a podium, resulting in lower PLW speeds. Beramek [14] investigated building geometry modifications on single (isolated) high-rise buildings. Sand-erosion tests were performed for three canopy sizes (2.5, 5 and 7.5 m) and two wind directions (0° and 45°). In addition, two podium sizes (10 and 20 m) were tested for three wind directions (0°, 45° and 90°). The application of a canopy led to lower PLW speeds directly below the canopy, while the use of a podium actually resulted in adverse effects. Here, the PLW speeds did not decrease but instead were spread over a larger area [14]. Jamieson et al. [19] carried out wind-tunnel experiments of different building designs (e.g. the use of a canopy, a podium, balconies, and an octagonal building shape) placed in a representative city model. A beneficial effect of canopies on the wind conditions directly beneath the canopy was found, in front of and around the frontal corners of the building, reducing the maximum PLW speed by 10% [19]. Here, the downwind wind streams were deviated away from the building, moving the areas of wind discomfort further away from the corners. The use of a podium reduced the extreme speeds at the base of a large building down to 70% [19]. Tsang et al. [20] conducted wind-tunnel measurements on a podium located underneath a row of four high-rise buildings. Only one wind direction and one podium size were investigated. The results indicated that the application of a podium can result in large areas of lowered PLW speeds around the buildings. Lam [21] performed wind-tunnel tests to evaluate the effect of introducing a permeable floor at an intermediate level. This could provide a bleed path for the upper-level winds to pass through the building before they are deflected downwards to pedestrian level by the windward façade of the building. The measurements showed that the introduction of a permeable floor reduced the spatial extent where the high-rise building led to uncomfortable wind conditions in the corner streams at pedestrian level. However, it did not result in lower peak values of the effective wind speed [21]. Uematsu et al. [22] carried out a series of wind-tunnel experiments on the effects of the corner shape of high-rise buildings on the PLW environment. Especially for a wind direction perpendicular to the building, the PLW speeds could be reduced by small modifications of the corner shape [22].

Most previous studies investigated the effect of geometric modifications on an isolated high-rise building. However, Wise [1] and Murakami et al. [18] found that the PLW speeds may be twice that of the free PLW speeds when the high-rise building is built amidst a group of low-rise buildings. The taller building brings down higher wind speeds into the surrounding low-rise urban area, in which wind speeds could even increase to three or four times that commonly experienced in towns [1]. Comparable results were found by Yoshie et al. [23] and Du et al. [24]. On the contrary, PLW speeds dropped significantly with the presence of high-rise buildings in high-density cities. Peng et al. [25] investigated the evolution of the wind environment as a function of urbanization in parts of Hong Kong. Based on CFD simulations and weather station data, they concluded that the average PLW speed was reduced by 67% from 1964 to 2010, due to the construction of large groups of high-rise buildings. By combining wind-tunnel experiments with literature information, Stathopoulos and Wu [26] established generic models and empirical relations for the wind conditions over streets depending on building geometry. For a tall building, surrounded by uniform blocks, the maximum PLW-speed amplification depended on the height difference and the blockage ratio of the surrounding blocks. Kubota et al. [27] proposed guidelines for the relationship between building density and PLW speeds, as a result of wind-tunnel measurements on 22 actual Japanese urban areas. They found a strong relationship between the average PLW speeds and the gross building coverage ratio, where an increase of the latter could decrease PLW speeds.

In addition to wind-tunnel testing, numerical simulation with CFD has been used increasingly to study PLW conditions in urban areas, as outlined in several review papers (e.g. Refs. [6,28–35]). CFD has some specific advantages over wind-tunnel measurements. Generally, wind-tunnel measurements are only performed at a few selected points in the model, whereas CFD provides data on the relevant parameters in all points of the computational domain, enabling a more detailed analysis of the wind flow around the building(s). In addition, CFD simulations easily allow parametric studies to assess different design configurations, especially when the configurations are all embedded within the same computational model. Furthermore, geometrical scaling and dynamic similarity are not an issue since the simulations can be performed at full scale. However, CFD verification and validation are imperative towards accurate and reliable results.

The studies mentioned above confirmed that the application of a canopy, podium and permeable floor can significantly improve the PLW environment, and that the use of CFD simulations is a valuable tool for the assessment of mechanical wind discomfort. However, to the best knowledge of the authors, no systematic parametric studies into the effect of these building geometry modifications on the PLW environment using validated CFD simulations have been reported so far. Therefore, this paper presents a parametric study to systematically evaluate the effect of each of these three building geometry modifications on the PLW environment around an isolated high-rise building. The explicit goal and novelty of this study is to link the geometrical dimensions/configurations of the modification to the actual quantitative improvement in PLW conditions and this for a wide range of dimensions/configurations.

The simulations are based on the 3D steady Reynolds-averaged Navier-Stokes (RANS) equations. Earlier review papers on PLW comfort [23,28,35] have shown that for this particular application, steady RANS simulations can provide sufficient accuracy in the wind comfort assessment. This is due to the fact that steady RANS simulations can provide accurate results in areas of high amplification factors, which have the highest contribution to exceeding discomfort thresholds in PLW comfort criteria [23,28,35–38]. Therefore, the total result of the wind comfort assessment is not significantly influenced by the low accuracy of RANS in the regions of low amplification factors. This has
been demonstrated in Refs. [28,35]. It is undisputed that LES intrinsically provides higher accuracy than steady RANS. Nevertheless, this potential is not always realized because LES is also more complex and the lack of best-practice guidelines for LES can result in less accurate and less reliable results. Moreover, steady RANS simulations require a lower simulation complexity and a much lower computational demand.

The paper is structured as follows. The sub-configuration validation study using a range of turbulence models is presented in Section 2. Section 3 describes the parametric study. Limitations of this study are given in Section 4. The main conclusions are presented in Section 5.

2. Validation study

Several best-practice guidelines are available for the assessment of PLW conditions using CFD [13,30,36–38]. To validate the CFD simulations, these guidelines report that ideally, on-site measurements should be performed for the actual configuration, the results of which should be compared with CFD simulations performed with different turbulence models [13,30,36,38]. However, such measurements are generally not available or possible, as is the case for this study that is based on a generic building configuration, and therefore so-called “sub-configuration validation” is applied [13,36,39,40]. This implies that the validation study is performed for generic components (subconfigurations) of a given configuration that exhibit the same salient flow features as the actual configuration. When a given set of computational settings and parameters results in a sufficiently accurate outcome for the subconfigurations, it can reasonably be assumed that the same combination of computational settings and parameters will also yield accurate simulation results for the actual configuration. The actual building configuration used in the parametric study has dimensions 60 m × 60 m × 15 m (H × W × D = 4:4:1). For the subconfiguration validation a building with dimensions 0.2 m × 0.1 m × 0.1 m (H × W × D = 2:1:1) at reduced scale is used. In this validation study, four different turbulence models are assessed.

2.1. Experimental data

A data set by the Architectural Institute of Japan is used [41]. Note that all parameters available in this data set only the mean wind speed is used since this is the parameter of interest in this study. The experiments were conducted in the atmospheric boundary layer wind tunnel of Tokyo Polytechnic University [41]. The wind-tunnel cross section at the measurement location is W × H = 1.2 m × 1.0 m. Fig. 1a gives a schematic illustration of the wind tunnel model setup. The building model dimensions are 0.2 m × 0.1 m × 0.1 m (H × W × D = 2:1:1). The model was submerged in a turbulent neutrally stratified atmospheric boundary layer (ABL). The inflow condition is described by a power-law profile with exponent 0.25. The approach-flow reference wind speed $U_{net}$ was 4.2 m/s at building height H. The wind speed was measured using a split film probe. The sampling frequency was set at 1,000 Hz and the sampling time was 120 s, resulting in 120,000 data values [41]. Measurements in the empty wind tunnel were made along two lines at two positions: 2H in front of the position where the building would be located (approach-flow profile) and at the location of where the leeward façade of the building would be located (incident profile), which is a rare but very good practice of characterizing the flow conditions [42,43]. Fig. 1b shows an acceleration of the mean wind-speed profile near the bottom of the wind tunnel, while the turbulent kinetic energy decreases in the lower part of the domain (see Fig. 1c). As demonstrated by Blokken et al. [42,43], this is to be attributed to the absence of roughness elements on the turntable and the subsequent acceleration of the flow over the smooth turntable. It is important to record these changes in view of an accurate CFD validation process. Measurements with the building included were taken in four planes with number of measurement points n as shown in Fig. 2.

2.2. CFD simulations: computational settings and parameters

2.2.1. Computational domain and grid

The computational domain has the width and height of the wind-tunnel cross section as recommended by best-practice guidelines [36], i.e. a width of 1.2 m (6H) and a height of 1 m (5H). The distance from the model to the inlet of the domain was taken as 0.4 m (2H). This distance is short but it is equal to the distance between the measured approach-flow profiles and the building in the wind tunnel, and these approach-flow profiles will be used as inlet boundary condition. The extension of the computational model behind the building is 2 m (10H) [37]. The grid was constructed using the surface-grid extrusion technique [44] that allows a large degree of control over the quality of the grid and its individual cells. It consists of only hexahedral cells. The grid resolution is determined based on grid-sensitivity analysis (reported in section 2.3.1).

2.2.2. Boundary conditions

The inlet boundary conditions in the CFD simulation have to match the measured approach-flow profiles from the wind-tunnel experiments to avoid unintended streamwise gradients that can distort the outcome of the validation study [27,30,37,45–47]. The ABL inflow consists of the vertical profiles of mean wind speed, turbulent kinetic energy and turbulence dissipation rate or specific dissipation rate, depending on the selected turbulence model. A comparison of the measured approach-flow mean wind-speed profile with the logarithmic law (see Fig. 1b) is performed. The logarithmic law, with an aerodynamic roughness length equal to $y_0 = 0.005$ m, shows an acceptable
agreement with the approach-flow profile. Therefore, the mean wind-speed profile is prescribed by the logarithmic law:

\[ U(y) = \frac{u_{\text{ABL}}^*}{\kappa} \ln \left( \frac{y + y_0}{y_0} \right) \]

(1)

in which \( y \) is the height co-ordinate, \( y_0 \) the aerodynamic roughness length equal to \( y_0 = 0.005 \text{m} \) (1.5 m at full scale, assuming \( H = 60 \text{m} \)), \( \kappa \) the von Karman constant (0.42), and \( u_{\text{ABL}}^* \) the ABL friction velocity:

\[ u_{\text{ABL}}^* = \frac{U_{\text{ref}}}{\ln \left( \frac{y + y_0}{y_0} \right)} \]

(2)

The reference wind speed \( U_{\text{ref}} \) is 4.2 m/s at 0.2 m height \((y/H = 1)\) and \( u_{\text{ABL}}^* = 0.475 \text{m/s} \), based on the fit of the log law with the measurement data in Fig. 1b. The profile of turbulent kinetic energy \( k \) is obtained directly from the approach-flow measurement data (see Fig. 1c), for which the standard deviations of the turbulent fluctuations were measured in three directions. The inlet profiles of turbulence dissipation rate \( \varepsilon \) (for both the \( k-\varepsilon \) turbulence models and the Reynolds stress model (RSM)) and specific dissipation rate \( \omega \) (for the shear-stress transport (SST) \( k-\omega \) turbulence model) are calculated with the following equations:

\[ \varepsilon(y) = \frac{\nu^{3/2}}{\kappa(y + y_0)} \]

(3)

\[ \omega(y) = \frac{\varepsilon(y)}{C_{\omega} k(y)} \]

(4)

with \( C_{\omega} \) a model constant \((= 0.09)\). At the outlet, zero static gauge pressure is specified. At the walls, the standard wall functions [48] with the sand-grain roughness modification by Cebeci and Bradshaw [49] are used. In the wind-tunnel study, no roughness elements were present on the bottom of the wind-tunnel floor in the vicinity of the building. To match the acceleration of the measured incident mean wind-speed profile in the CFD simulations (see Fig. 1b), zero roughness is imposed at the bottom of the domain. The same conditions are applied at the building surfaces, and at the top and lateral boundaries. Because the computations are compared with wind-tunnel measurements, the top and lateral boundaries of the domain are treated as no-slip walls.

2.2.3. Solver settings

The CFD simulations are performed using the commercial CFD code ANSYS Fluent 15.0.7 [50]. The 3D steady Reynolds-averaged Navier-Stokes (RANS) equations are solved. The grid-sensitivity analysis is performed with the realizable \( k-\varepsilon \) turbulence model [51]. The validation study is performed with several turbulence models, including the realizable \( k-\varepsilon \) turbulence model, the renormalization (RNG) \( k-\varepsilon \) turbulence model [52], the SST \( k-\omega \) turbulence model [53] and the linear pressure-strain RSM model [54]. Pressure-velocity coupling is taken care of by the SIMPLE algorithm. Pressure interpolation is second order. Second-order discretization schemes are used for both the convection terms and viscous terms of the governing equations. The iterations were terminated when the scaled residuals did not show any further reduction with increasing number of iterations. For all simulations a minimum value of \( 10^{-4} \) was found for continuity. For the mean \( x- \), \( y- \), and \( z- \) velocity components, turbulent kinetic energy \( k \) and turbulence dissipation rate \( \varepsilon \) the minimum value was \( 10^{-6} \), while a value of \( 10^{-5} \) was found for the specific dissipation rate \( \omega \). The minimum value found for the Reynolds shear stresses was \( 10^{-6} \), while for the Reynolds normal stresses this was \( 10^{-7} \).

2.3. Results

2.3.1. Grid-sensitivity analysis

A grid-sensitivity analysis is performed for which five grids are constructed as shown in Fig. 3. Coarsening and refining is performed with an overall linear factor of \( \sqrt{2} \). A comparison is made based on wind-speed ratios, which are defined as the ratio of the local mean wind speed in the \( x \)-direction \((U_x)\) to the reference wind speed at building height \((U_{\text{ref}} = 4.2 \text{ m/s})\). The results are compared in four planes for the points displayed in Fig. 2 and the deviations between the results on different grids are expressed by the root-mean-square error (RMSE):

\[ \text{RMSE} = \frac{1}{n} \sum_{i=1}^{n} (y_1 - y_2)^2 \]

(5)

with \( y_1 \) and \( y_2 \), the wind-speed ratios at equal points in the two grids to be compared.

Fig. 4a shows discrepancies of more than 20% between the results.
Fig. 3. (a) Geometry for grid-sensitivity analysis and validation study. (b–f) Computational grids used in grid-sensitivity analysis; (b) coarsest grid: 25,540 cells, (c) coarse grid: 70,896 cells, (d) medium grid: 201,328 cells, (e) fine grid: 557,850 cells, and (f) finest grid: 1,610,624 cells.

Fig. 4. Scatterplot of the mean wind-speed ratio ($U_x/U_{ref}$) obtained on: (a) Coarsest vs. coarser grid; (b) coarse vs. medium grid; (c) medium vs. fine grid; and (d) fine vs. finest grid.
obtained on the coarser and coarse grid, especially in the xz-plane (= horizontal plane; see Fig. 2a). The total RMSE is 0.0599. Furthermore, some sampling points on the xy-plane also show large deviations between the two grids. The RMSE between the medium and the coarse grid is 0.0105 (Fig. 4b). The results on the coarse grid are in close agreement with the results on the medium grid. The deviations in the xz-plane are significantly reduced for these grids. However, there are differences in the xy-plane up to 10% even for high wind-speed ratios. Because the higher wind-speed values are most important in PLW studies, the coarse grid is not considered satisfactory. Fig. 4c shows the wind-speed ratios calculated using the medium and fine grid. Although there are no sampling points which show high wind-speed ratios ($U_x/U_{ref} > 1$) that deviate more than 3%, the total RMSE has slightly increased to 0.0135. The results of the simulation using the fine and finest grid are shown in Fig. 4d. The total RMSE is the lowest here (= 0.0102). Moreover, deviations concerning the higher wind-speed ratios ($U_x/U_{ref} > 1$) that were noticed before are within a 1.3% range now. This implies that the fine grid provides nearly grid-independent results, and this grid is therefore used for the remainder of the validation study.

2.3.2. Validation study

The wind-speed ratios ($U_x/U_{ref}$) of the wind-tunnel experiments are compared to those from the CFD simulations at the points displayed in Fig. 2. Fig. 5a shows the RMSE of the different simulation results with respect to the wind-tunnel experiments. The RSM model shows the lowest RMSE in the horizontal plane. However, the overall performance of the RSM model when considering all planes is slightly less compared to the performance of the realizable k-ε model. In addition, the RSM model often deals with convergence problems while using the required second-order discretization schemes. This could cause large iterative convergence errors, which is an additional reason to disregard this model for the parametric study. The RNG k-ε model performs better than the realizable k-ε model in front of the building and near the bottom of the domain, but yields a larger overall error due to inferior predictions besides and behind the building (not shown in figure). The results obtained using the realizable k-ε turbulence model show the best agreement with the wind-tunnel experiments in three of the four measurement planes. The k-ε model is well described in the literature and it is known that it can underestimate the separation and recirculation regions above and besides the building and may overestimate the speed and the size of the recirculation area behind the building (e.g. Refs. [55,56]). By breaking down the xz-plane at pedestrian level into three different sections, the results in Fig. 5b confirm this statement as the results in front of the building show a much better agreement than the results besides and behind the building. However, the higher wind-speed ratios show a good agreement with the experiments in every plane, which is important, since these are the values that are most relevant for wind discomfort and wind danger. This

![Fig. 5. (a) RMSE between CFD results and wind-tunnel measurements for the four measurement planes. (b) Scatterplot of the CFD results (realizable k-ε turbulence model) and wind-tunnel results in the xz-plane (y/H = 0.0625).](image-url)
corresponds with the main conclusions from the review by Blocken et al. [28]. For these reasons, the realizable k-ε turbulence model is retained for the parametric study.

3. Parametric study

3.1. Computational geometry and domain

The geometry modifications are applied to an explicitly modeled high-rise building. The dimensions of this target building are based on the study by Beranek and van Koten [4]. They stated that significant wind discomfort can be expected when the height of the building exceeds 50 m. In addition, Willemesen and Wisse [5] stated that buildings higher than 30 m always require a wind comfort assessment study based on CFD simulations or wind-tunnel tests. This results in a building of $H \times W \times D = 60 \times 60 \times 15 \, \text{m}^3$ selected for this study (see Fig. 6a). Because these dimensions differ from the ones used in the validation study, new grids are created and additional grid-sensitivity analyses are performed (see Section 3.4). Concerning the parametric study, CFD simulations are performed for three wind directions (ϕ):

Fig. 6. (a) Building geometry and simulated wind directions. Setup for the parametric study: (b) canopy; (c) podium; and (d) permeable floor.
ϕ = 0° (x direction), ϕ = 45° and ϕ = 90° (z direction), the results of which are representative of eight wind directions due to symmetry of the building. For the evaluation of PLW speeds, the CFD results are extracted from a sampling plane at a height y = 1.75 m, indicated by the dotted line in Fig. 6a.

The canopy (see Fig. 6b) is attached at a height of 4 m (1/15 H). The podium (see Fig. 6c) has a height of 4 m. The parametric study investigates the following depths (S₀) of the canopy and podium: 3, 6, 12, 18, 24 and 30 m (S₀/H = 0.05, 0.1, 0.2, 0.3, 0.4 and 0.5). The canopy is implemented as a zero-thickness smooth wall in the computational geometry and grid. The configurations with permeable floor (FP) are obtained by omitting floor 0, 1, 2, 3, 4, 5, 7, 9, 10, 11 and 13, respectively (see Fig. 6d). The height of each floor is 4 m.

The dimensions of the computational domain are based on the best-practice guidelines for CFD simulations in wind engineering [36–38,57–60], i.e. distances between the target building and the domain faces of 5H in the vertical, lateral and upstream direction and 15H in the downstream direction, with H = 60 m. The simulation of different wind directions ϕ is taken into account by varying the inlet faces as shown in Fig. 7a.

The surroundings of the target building are not modeled explicitly (i.e. with their actual shape and size) but implicitly by means of certain values for the aerodynamic roughness length y₀. These values are drawn from the updated Davenport-Wieringa roughness classification [61]. Fig. 7c depicts how the bottom of the domain is split up into two parts: an outer part and an inner part. For the outer part, y₀ = 0.5 m is imposed, which corresponds to an “old cultivated landscape with many rather large obstacle groups separated by open space of about 10 obstacle
heights” [61]. This type of terrain is chosen because a high-rise building, as investigated in this study, is rarely built in wide open spaces. The area closer to the building however is described by a lower aerodynamic roughness length of \( y_0 = 0.03 \text{m} \), which implies open terrain with low vegetation, e.g. grass [61].

### 3.2. Computational grid

To allow a large degree of control over the quality of the grid, it is constructed using the surface-gridded extrusion technique by van Hooff and Blocken [44]. The different geometrical variations (canopy and podium depths) required for the parametric study are included in one geometry and grid, which allows easy production of a range of different geometries and grids without having to rebuild them all from scratch [44]. In the area of interest, at least 18 cells per cubic root of the building volume are used and 10 cells per building opening [30,37]. In addition, as the focus is on the pedestrian level, at least three cells are provided below pedestrian height (1.75 m) [30,37]. The necessary grid resolution is determined by a grid-sensitivity study (see Section 3.4) resulting in grids of 1,006,080 hexahedral cells for the canopy, grids ranging from 1,003,704 to 1,066,080 hexahedral cells for the podium and grids ranging from 2,043,948 to 2,049,348 hexahedral cells for the permeable floor.

Strictly, a set of four requirements needs to be satisfied in the CFD simulations, as outlined in Ref. [45]:

1. ANSYS Fluent uses \( k_S \)-type wall functions, where \( k_S \) is the equivalent sandgrain-roughness height;
2. The relationship between \( k_S \) and the aerodynamic roughness length \( y_0 \) in ANSYS Fluent is: \( k_S = 9.793y_0/C_D \) [45], where \( C_D \) is the roughness constant (default = 0.5);
3. At least three cells need to be provided below pedestrian height (= 1.75 m);
4. In ANSYS Fluent, the near-wall grid size should be such that \( y_P > k_S \) with \( y_P \) the center point of the wall-adjacent cell.

For high values of \( y_0 \), such as 0.5 m, these four requirements cannot be satisfied simultaneously. Therefore, various remedial measures were proposed by Blocken et al. [45]. The measure applied in this study is the use of a non-conformal grid (e.g. Refs. [30,45]). Here, different cell heights are used for the ground-adjacent cells in different areas of the domain, together with adjusted values for \( C_D \) to satisfy \( y_0 = 9.793y_0/C_D \) [45] and \( y_P > k_S \) at every position in the domain. This involves using higher cells in the outer area around the target buildings (where \( y_0 = 0.5 \text{m} \) and strictly 3 grid layers below 1.75 m are not needed) while lower cells can be used in the inner area (where \( y_0 = 0.03 \text{m} \), as shown in Fig. 7c,d,e. An additional intermediate grid is created to smooth the transition between these two separate grids. To compute the flow across the non-conformal boundary, ANSYS Fluent computes the intersection between the interface zones that comprise the boundary and interpolates results across these interfaces [50]. This allows using \( y_0 = 0.5 \text{m} \) without introducing unintended streamwise gradients [45] while keeping sufficient grid resolution in the area of interest.

### 3.3. Boundary conditions and solver settings

The inlet mean wind-speed profile is prescribed by the logarithmic law (see Eq. (1)) with \( y_0 = 0.5 \text{m} \). In order to achieve horizontal homogeneity of the ABL profiles [45] for a different value for \( y_0 \), the boundary conditions were slightly adapted compared to those used in the validation study. The reference wind speed \( U_{10} \) is 5 m/s at 10 m height. The turbulent kinetic energy \( k \) is calculated using the equation by Richards and Hoxey [62]:

\[
k(y) = \frac{u_*^2 ABL^2}{\sqrt{C_D}}
\]

with \( C_D = 0.09 \). The turbulence dissipation rate \( \varepsilon \) is given by Eq. (3). At the walls, the standard wall functions [48] with the sand-grain roughness modification by Cebeci and Bradshaw [49] are used. Roughness is specified for the bottom of the domain while the building walls are considered smooth. Zero static gauge pressure is specified at the outlet. Zero normal velocity and zero normal gradients of all variables are imposed on the lateral and top boundaries.

The CFD simulations are performed by solving the 3D steady Reynolds-averaged Navier-Stokes (RANS) equations using the commercial CFD code ANSYS Fluent 15.0.7 [50]. The realizable \( k-\varepsilon \) turbulence model [51] is adopted for closure. Second-order discretization schemes are used for the governing equations and the equations of the turbulence model. The SIMPLE algorithm is used for the pressure-velocity coupling. Pressure interpolation is second order. The iterations were terminated when the scaled residuals did not show any further convergence. The following minimum values were obtained: \( 10^{-7} \) for the mean x-, y- and z-velocity components, \( 10^{-6} \) for the turbulent kinetic energy \( k \), \( 10^{-5} \) for the turbulence dissipation rate \( \varepsilon \) and \( 10^{-4} \) for continuity.

First, a simulation is performed in an empty domain to analyze the acceleration of the flow at pedestrian level due to potential unintended streamwise gradients [45]. As expected, the incident mean wind speed increases substantially (+46% on pedestrian level) near the bottom of the domain. This increase is comparable to the validation study (see Fig. 1b). Here, an increase in wind speed of 49% was found at equivalent dimensionless height 0.03H. This acceleration is inevitably due to the different roughness values imposed on the bottom of the domain. Note however that in reality such accelerations are also observed in the direct proximity of high-rise buildings situated in relatively open terrains, such as streets, grass, or parks.

### 3.4. Grid-sensitivity analysis

The grid-sensitivity analysis is performed by comparing the mean wind speeds at pedestrian level (\( h = 1.75 \text{m} \)). The analysis focuses on the 9 m (S/H = 0.15) canopy configuration (Fig. 8a and c) and on the building configuration with the third floor (fourth building layer, \( F_p = 3 \)) removed (Fig. 8b and d). The grid refinement is performed with an overall linear refinement factor of \( 2 \). Simulations are performed for \( \phi = 0^\circ \). The results are normalized by the incident PLW speed (\( h = 1.75 \text{m} \)) \( U_0 = 3.66 \text{m/s} \).

Fig. 8e and f shows the results obtained on the medium grid versus those on the fine grid for the canopy/podium and permeable-floor solution, respectively. In both cases, the largest deviations are found in the recirculation areas bounded by the corner streams in which the mean wind speed is relatively low. Maximum differences in \( U/U_0 \) of 0.123 are observed for the canopy/podium grid and of 0.100 for the permeable-floor grid. A RMSE of 0.012 is calculated within the sampling area with an offset of 50 m (5/6H) around the building at a height of \( h = 1.75 \text{m} \). For the permeable-floor grid, a RMSE of 0.009 is calculated. Both medium grids show a good compromise between computational accuracy and computational costs and are therefore selected for further analysis. In addition, it is chosen not to coarsen the grids any further since this would conflict with the best-practice guidelines about the minimum number of cells in building openings and along building edges.
3.5. Results

The results are analyzed in a sampling plane within an area covering the building circumference until an offset of 50 m (5/6H) around the building at a height of h = 1.75 m. In this particular study, an offset size of 5/6 H is deemed sufficient as it can capture the high-speed corner streams for every configuration with added canopy or podium. The results are normalized by the incident PLW speed value $U_0$ at pedestrian level ($h = 1.75$ m) (i.e. in the empty domain) for the particular wind direction $\phi$: $U_0 = 3.66$ m/s ($\phi = 0^\circ$), $U_{45} = 3.87$ m/s ($\phi = 45^\circ$) and $U_{90} = 3.91$ m/s ($\phi = 90^\circ$). Area-averaged and maximum mean wind speeds are calculated for the sampling plane. With respect to evaluating the change of the overall PLW speeds, it is debatable whether the averaging plane should include or exclude the solid part. This is especially true for the podium solution, in which the solid part grows with increasing podium depth. In this study it is chosen to use the solid-

Fig. 8. (a,b) Medium grid for: (a) canopy (1,066,080 cells); (b) permeable floor (2,049,348 cells). (c,d) Fine grid for: (c) canopy (2,130,664 cells); (d) permeable floor (3,539,910 cells). (e,f) Scatterplot of the fine versus the medium grid regarding the dimensionless wind speed ($U/U_0$) in a horizontal grid at a height of 1.75 m with an offset of 50 m (5/6H) around the building and an equal grid spacing of 3 m (0.05H) m for; (e) canopy (f) permeable floor.
exclusive average, which is widely used in comparable studies with similar problems (e.g. Refs. [25,27]). For nearly all cases, the maximum mean wind speed on pedestrian level is found in the corner streams. Therefore, the maximum mean wind speed indicator allows a fair comparison between the different configurations. Exceptions are the $F_p = 0$ cases, in which the maximum mean wind speed is present in the passage below the building. For the sake of brevity, only several cases are shown in Figs. 9, 11 and 13. An overview of all cases can be found in the appendix to this paper.

3.5.1. Canopy

The canopy depth is varied between 3 m and 30 m, representing $S_0/H = 0.05$ to 0.5.

Fig. 9 presents contours of the dimensionless mean wind speed ($U/U_\phi$) and the superimposed mean velocity vector fields at $y = 1.75$ m and $z/H = 0$ for $S_0/H = 0$ (reference case, left column) and $S_0/H = 0.3$ (right column), for three wind directions: $\phi = 0^\circ$, $45^\circ$ and $90^\circ$. In the reference case (Fig. 9a), in front of the building, the wind flows in upstream direction (negative x-direction). This is due to the standing vortex created by the downflow in front of the building, as shown in Fig. 9c. However, when a canopy is attached, the wind flows in downstream direction (positive x-direction) at this location (Fig. 9b). This implies the absence of a standing vortex at pedestrian level, which subsequently reduces the mean wind speed in the corner streams. The wind flowing downwards along the windward building facade does not longer reach pedestrian level (Fig. 9d) since it is partly evacuated over the canopy. Fig. 9a and b shows that the shape of the corner streams (red-yellow area) is altered, moreover, it shows that the maximum value in these areas is reduced. Indeed, the maximum mean wind speed ($U_{\text{max}}/U_0$) in the corner streams on pedestrian level declines from 1.82 for the reference case to 1.62 for the case with a canopy attached $S_0/H = 0.3$.

Also in the $45^\circ$ situation (Fig. 9e), the corner streams are responsible for the highest mean wind speeds, especially near the corner at the end of the long windward facade. Without canopy, the approaching wind flow is directed partly downwards along the windward building facade, increasing the mean wind speed of the far end corner stream. By adding the canopy, a large part of this downflow does not reach the pedestrian level anymore and as a result, the maximum mean wind speed decreases significantly. Indeed, $U_{\text{max}}/U_0$ decreases from 1.77 for the reference case to 1.47 for the canopy with $S_0/H = 0.3$ (see Fig. 9f). Differences in local wind directions can be observed in front of the long windward facade: without canopy, the wind flow is directed away from the building, as a result of the wind flowing downwards along the windward facade. In the situation with canopy however, the wind is guided along the windward building facade at pedestrian level.

The largest reduction of maximum mean wind speed is found for $\phi = 90^\circ$. A canopy with $S_0/H = 0.3$ reduces this wind speed by 21% (Fig. 9b). The maximum mean speed in the corner streams decreases from $U_{\text{max}}/U_0 = 1.52$ to $U_{\text{max}}/U_0 = 1.13$ (see Fig. 9g vs. Fig. 9h). The large aspect ratio of the cross section below the canopy, together with its relatively large length in streamwise direction, results in a flow configuration similar to flow between two parallel flat plates. Due to the flow resistance in the space between the canopy and the ground, the separated corner streams are deviated to the lateral edges of the canopy, instead of reattaching on the long edges of the building. As a result, the mean wind speed on the leeward side of the building reduces substantially.

Fig. 10 summarizes the results for all canopy depths tested in terms of the area-averaged mean wind speed and the maximum mean wind speed. For $\phi = 0^\circ$, the values do not show strong changes as a function of canopy size: a decrease of 2% is noted for $S_0/H = 0.5$. For $\phi = 45^\circ$, a
decrease of 7% is obtained for $S_0/H = 0.5$ while for $\phi = 90^\circ$ the area-averaged mean wind speed is reduced by 29%. For the maximum mean wind speed however, the decrease is more pronounced (see Fig. 10b). By increasing the canopy size, the maximum mean wind speed in the corner streams reduces substantially. A maximum reduction of 12% is noted for $\phi = 0^\circ$, while this is 20% for $\phi = 45^\circ$ and 36% for $\phi = 90^\circ$.

3.5.2. Podium

The podium depth is varied between 3 m and 30 m, representing $S_0/H = 0.05$ to 0.5. Fig. 11 presents contours of $U/U_0$ and the mean velocity vector fields for two cases: $S_0/H = 0.05$ and $S_0/H = 0.3$, for three wind directions: $\phi = 0^\circ$, $45^\circ$ and $90^\circ$. Adverse effects are found for $\phi = 0^\circ$ and $S_0/H = 0.05$, where high mean wind speeds ($U_{max}/U_0 = 1.86$) are visible in the corner streams in Fig. 11a. The depth is too small to catch the downflow on top of the podium and the standing vortex in front of the building persists at the pedestrian level (Fig. 11c). The increased frontal area of the building actually increases the maximum mean wind speed in the corner streams by 2% compared to the reference case (Fig. 9b). If the size of the podium is increased to $S_0/H = 0.3$ (Fig. 11b), the corner streams are significantly reduced in terms of maximum mean wind speed ($U_{max}/U_0 = 1.51$) and the higher wind speeds are located further away from the original building geometry. In this case, the standing vortex is retained on top of the podium and effectively held away from pedestrian level (see Fig. 11d).

For $\phi = 45^\circ$, comparing Fig. 11e with Fig. 9e shows that the application of a podium with $S_0/H = 0.05$ results in a negligible reduction of the size and strength of the corner streams from $U/U_{45} = 1.77$ to 1.76. A very significant reduction ($U/U_{45} = 1.52$) however is observed when the podium size is increased to $S_0/H = 0.3$ (Fig. 11f).

The largest reduction of the maximum mean wind speed is present for $\phi = 90^\circ$, from $U/U_{90} = 1.45$ for $S_0/H = 0.05$ to $U/U_{90} = 1.17$ for $S_0/H = 0.3$. Fig. 11g and h shows a decrease of the high-speed wind area with increasing podium size. In the area behind the wide podium, the mean wind speed reduces to almost zero.

Fig. 12 summarizes the results for all podium depths tested. Overall and except for the smallest podium depths and $\phi = 0^\circ$, the area-averaged (Fig. 12a) and the maximum (Fig. 12b) mean wind speed in the sampling plane decrease with increasing podium depth for all wind directions. The largest reduction of the area-averaged and maximum mean wind speed is found with the application of the deepest podium ($S_0/H = 0.5$). Concerning the area-averaged mean wind speed a maximum reduction of 12% is found for $\phi = 0^\circ$, 22% for $\phi = 45^\circ$ and 26% for $\phi = 90^\circ$. For the maximum mean wind speed the largest reduction is 34% for $\phi = 90^\circ$, while for wind directions $\phi = 0^\circ$ and $\phi = 45^\circ$ a reduction of approximately 28% and 26% is achieved, respectively.

3.5.3. Permeable floor

The permeable floor is varied between the ground floor and the 13th floor, representing $F_P = 0$ to 13. Fig. 13 presents contours of the dimensionless mean wind speed ($U/U_0$) and the mean velocity vector fields for four cases: $F_P = 0$, $F_P = 1$, $F_P = 2$ and $F_P = 3$, for three wind directions: $\phi = 0^\circ$, $\phi = 45^\circ$ and $\phi = 90^\circ$. The removal of a building floor can reduce the maximum and area-averaged mean wind speed around a building at pedestrian level, but as shown in Fig. 13, the reductions are smaller than for the canopy and podium solution (see Section 3.5.1 and 3.5.2). For $\phi = 0^\circ$, contours of the dimensionless mean wind speed ($U/U_0$) and the mean velocity vector fields in the vertical cross section $z/H = 0$ are given in Fig. 14. The wind flowing downwards in front of the building (partly) flows through the permeable floor and thus does not participate in the creation of a standing vortex at ground level.

However, caution should be taken in removing lower building floors. In this case, the high wind speeds within the permeable floor can affect the PLW speeds. For $F_P = 0$ (see Fig. 13a,e), a large area of increased mean wind speed is noted at pedestrian level. Furthermore, the removal of the first floor ($F_P = 1$) causes an increase in corner stream mean wind speed from $U_{max}/U_0 = 1.82$ to $U_{max}/U_0 = 1.86$. In addition, Fig. 14c shows that the high-speed wind flow through the permeable floor is directed downwards on the leeward side of the building, thereby increasing the PLW speed in the lee of the building.

For $F_P = 2$, the wind flow through the permeable floor barely causes an increase in mean wind speed in the wake region of the building (see Fig. 13c). In addition, the mean wind speed directly in front of the building at pedestrian level is reduced due to the fact that downward directed high-speed winds in front of the building flow through the permeable floor (see Fig. 14d), and thus do not strongly participate in the creation of a standing vortex at ground level. By removing higher-located floors, the mean wind speed in the corner streams approach levels observed in the reference case (see Fig. 13d, h and 13l vs. Fig. 9a,
e and 9g), while the area-averaged mean wind speed is reduced compared to the reference case.

Fig. 15 summarizes the results for all levels of permeable floors tested. It confirms that it is not advisable to convert the ground floor ($F_p = 0$) of the building into a permeable floor, as widely discussed in previous studies [1,16,17,63]. The area-averaged mean wind speed is 20.4% higher when the ground floor is open for $\phi = 0^\circ$ and 15.4% higher for $\phi = 45^\circ$ (Fig. 15a). Removing the first floor ($F_p = 1$) of the building increases the area-averaged mean wind speed around the building by 8.0% compared to the reference case for $\phi = 0^\circ$ and by 3.9% for $\phi = 90^\circ$. By removing higher-located floors, the area-averaged mean wind speed is reduced further with reductions of approximately 6%, 6% and 1% by the removal of the seventh floor (Fig. 15a) for $\phi = 0^\circ$, 45° and 90°, respectively.

In general, the maximum mean wind speed decreases with the removal of a lower building layer. However, when a too-low building layer is removed, adverse effects on the PLW environment are observed (Fig. 15b). In contrast to the area-averaged mean wind speeds, for $FP = 0$ the maximum mean wind speeds show an increase of only 3.3% for $\phi = 45^\circ$, and even a decrease of 3.0% for $\phi = 0^\circ$ and 9.7% for $\phi = 90^\circ$ (Fig. 15b). This effect is also noticeable for $\phi = 90^\circ$. The maximum mean wind speed is reduced by 9.7%, while on the other hand the area-averaged mean wind speed is 3.0% higher than the reference case (see Fig. 15a vs. Fig. 15b). The removal of the second floor ($FP = 2$) of the building leads to the largest reduction of the maximum mean wind speed at pedestrian level for $\phi = 0^\circ$ and $\phi = 45^\circ$, with a reduction of 6.4% and 6.6% (Fig. 15b). For $\phi = 90^\circ$, a maximum reduction of 10.1% is found for $F_p = 1$.

4. Limitations and future work

This study provides systematic insights of the potential reduction in PLW speed due to the addition of geometric features: canopies, podiums and permeable floors. The study is subjected to a number of limitations, which can be addressed in future research:

- This study focused on mean wind speed as the target parameter. Nevertheless, it is well-known that in addition to the mean wind speed, wind gusts also contribute to PLW discomfort and especially danger (e.g. Refs. [28,64,65]). Further research should focus on the impact of turbulent fluctuations on PLW comfort and of the impact of building geometry modification on the turbulent fluctuations. However, it is well-known that obtaining accurate estimates of the standard deviation of the turbulent fluctuations of the wind speed is much more difficult than for mean wind speed. This holds for both wind-tunnel testing and for CFD. This is at least one of the reasons why many comfort criteria are only based on mean wind speed and not on a combination of mean wind speed and turbulent fluctuations. An example are the criteria in the only PLW comfort standard in the world today, the Dutch Wind Nuisance Standard NEN8100 [8].

- The impact of the building geometry modifications was quantitatively assessed by means of average and maximum mean PLW speeds. The use of other quantitative methods or statistical analyses could provide additional insights in the effectiveness of the applied measures.

- A simplified isolated high-rise building was considered in the current study. Surrounding buildings will alter the wind flow pattern and may both increase [1,18,23,24] or decrease [25,27] PLW speeds. Evidently, this will also affect the effectiveness of the applied measures. The impact of building details, different building dimensions, and the performance of the investigated solutions for a

---

**Fig. 11.** Contours of the dimensionless velocity magnitude ($U/U_\phi$) and velocity vector field in (a,b,c,f,g,h) the sampling plane at a height of 1.75 m and (c,d) the vertical cross section at $z/H = 0$: (a,c,e,g) $S_p/H = 0.05$. (b,d,f,h) $S_p/H = 0.3$. (a,b,c,d) $\phi = 0^\circ$. (e,f) $\phi = 45^\circ$. (g,h) $\phi = 90^\circ$.
Fig. 12. Effect of adding a podium with $S_0/H$ on (a) area-averaged wind speed and (b) maximum wind speed.

Fig. 13. Contours of the dimensionless velocity magnitude ($U/U_\omega$) and velocity vector field in the sampling plane at a height of 1.75 m: (a,e,i) $F_P = 0$. (b,f,j) $F_P = 1$. (c,g,k) $F_P = 2$. (d,h,l) $F_P = 3$. (a,b,c,d) $\phi = 0^\circ$. (e,f,g,h) $\phi = 45^\circ$. (i,j,k,l) $\phi = 90^\circ$. 

high-rise building in a given urban environment should be taken into account in future research.

- Further research may focus on a permeable floor above a canopy/podium, i.e., a combination of two of the measures studied in this paper. Also, a study on the influence of the thickness of the canopy could be relevant. The application of a vertical edge on top of the circumference of the canopy could further reduce the wind flowing downwards to pedestrian level and yield further improvements.

- Also an investigation on varying heights of the canopy/podium can be a valuable extension of the present study.

- The results with respect to the permeable floor may be improved by the application of Venturi-shaped openings (e.g., Refs. [66,67]) to guide the wind flow to a greater extent through the permeable floor. In addition, the empty space created by the permeable floor might be used for the placement of wind turbines for the generation of electricity [68].

Fig. 14. Contours of dimensionless velocity magnitude \( (U/U_\phi) \) and velocity vector field in vertical cross section at \( z/H = 0 \) for \( \phi = 0° \): (a) Reference case. (b) \( F_P = 0 \). (c) \( F_P = 1 \). (d) \( F_P = 2 \). (e) \( F_P = 3 \). (f) \( F_P = 7 \).
5. Conclusions

Using computational fluid dynamics (CFD), a parametric study is carried out to investigate the effect of different building geometry modifications on the PLW speed around a generic high-rise building. These modifications include: (i) a canopy around the building at height 4 m above the ground; (ii) a podium attached to the base of the building; and (iii) the introduction of a permeable floor in the target building. The geometrical dimensions of the canopy and podium, and the position of the permeable floor, are linked to the quantitative reduction in mean wind speed at pedestrian level. It is shown that in general the canopy and podium solutions have a much larger effect on the PLW speed than the introduction of a permeable floor. For the specific building geometry and geometrical modifications analyzed in this study, the following conclusions are provided:

5.1. Canopy
- The application of a canopy can lead to a decrease in the area-averaged mean wind speed, with a maximum decrease of 29% for a depth of \( S_0/H = 0.5 \) and \( \phi = 90^\circ \). In general, the decrease in area-averaged mean speed becomes larger with increasing canopy size.
- The maximum decrease in the maximum occurring mean wind speed is 36% for a depth of \( S_0/H = 0.5 \) and \( \phi = 90^\circ \). The decrease in maximum speed generally becomes larger with increasing canopy size.
- The application of a canopy generally moves the higher-speed wind streams further away from the building.
- The PLW speeds in front of a long building façade can decrease significantly by the application of a canopy.

5.2. Podium
- The application of a podium leads to a maximum decrease of the maximum (34%) and area-averaged (26%) PLW speeds around the building. This decrease is larger with increasing podium size.
- The mean wind speeds in the corner streams can be significantly reduced. However, caution should be taken as a small podium may increase the PLW speeds in the corner streams.

5.3. Permeable floor
- One should avoid the use of the ground floor as a permeable floor in the building. As known from literature, and as shown again in this study, this can lead to large high-speed wind areas at pedestrian level.
- The lowest area-averaged PLW speed is obtained when a floor approximately halfway the building is removed; reductions of up to 6% compared to reference case.
- A permeable floor at lower heights can decrease the maximum mean wind speed at pedestrian level with up to 10.1%.
- If a too-low building layer is removed from the building, the area-averaged PLW speed increases and the low-speed wake region behind the building is reduced in size.

It should be noted that the intention of this study was to provide insights into the effect of three types of building geometry modifications. Its intention was not to derive generally valid design guidelines for practical use. This is not possible, since the effect of a certain modification depends not only on the building geometry under study and its surroundings but also on the specific geometry of the modification, which can be very different from the simple geometries in the present study. Although the present study provides insights into what measures can be effective, for practical applications, a dedicated wind comfort and wind danger study will have to be performed in which the specific geometry of the building, its surroundings and its potential geometrical modification should be taken into account.

Acknowledgements

Twan van Hooff and Hamid Montazeri are currently postdoctoral fellow of the Research Foundation Flanders and acknowledge its financial support (project FWO 12R9718N and project FWO 12M5319N, respectively). The authors also gratefully acknowledge the partnership with ANSYS CFD.
Appendix A

Fig. A1. Canopy: Contours of the dimensionless velocity magnitude ($U/U_p$) and velocity vector field in the sampling plane at a height of 1.75 m for the reference case, $S_c/H = 0.05$, $S_c/H = 0.1$ and $S_c/H = 0.2$, for three wind directions $\phi = 0^\circ$, $\phi = 45^\circ$ and $\phi = 90^\circ$.  


Fig. A2. Canopy: Contours of the dimensionless velocity magnitude \((U/U_\phi)\) and velocity vector field in the sampling plane at a height of 1.75 m for \(S_\phi/H = 0.3\), \(S_\phi/H = 0.4\) and \(S_\phi/H = 0.5\), for three wind directions \(\phi = 0^\circ\), \(\phi = 45^\circ\) and \(\phi = 90^\circ\).
Fig. A3. Podium: Contours of the dimensionless velocity magnitude \( \frac{U}{U_0} \) and velocity vector field in the sampling plane at a height of 1.75 m for the reference case, \( S_0/H = 0.05 \), \( S_0/H = 0.1 \) and \( S_0/H = 0.2 \), for three wind directions \( \phi = 0^\circ \), \( \phi = 45^\circ \) and \( \phi = 90^\circ \).
Fig. A4. Podium: Contours of the dimensionless velocity magnitude ($U/U_4$) and velocity vector field in the sampling plane at a height of 1.75 m for $S_0/H = 0.3$, $S_0/H = 0.4$ and $S_0/H = 0.5$, for three wind directions $\phi = 0^\circ$, $\phi = 45^\circ$ and $\phi = 90^\circ$. 
Fig. A5. Permeable floor: Contours of the dimensionless velocity magnitude ($U/U_\phi$) and velocity vector field in the sampling plane at a height of 1.75 m for the reference case, $F_P = 0$, $F_P = 1$ and $F_P = 2$, for three wind directions $\phi = 0^\circ$, $\phi = 45^\circ$ and $\phi = 90^\circ$. 
Fig. A6. Permeable floor: Contours of the dimensionless velocity magnitude ($U/U_\phi$) and velocity vector field in the sampling plane at a height of 1.75 m for $F_p = 3$, $F_p = 4$, $F_p = 5$ and $F_p = 7$, for three wind directions $\phi = 0^\circ$, $\phi = 45^\circ$ and $\phi = 90^\circ$. 

Fig. A7. Permeable floor: Contours of the dimensionless velocity magnitude \((U/U_\phi)\) and velocity vector field in the sampling plane at a height of 1.75 m for \(F_p = 9, F_p = 10, F_p = 11\) and \(F_p = 13\), for three wind directions \(\phi = 0^\circ, \phi = 45^\circ\) and \(\phi = 90^\circ\).

References
