Impact of volume distribution on pedestrian wind environment in high-rise urban districts: a CFD study

Milad Sadeghfar¹, Sadra Sahebzadeh²

¹ University of Tehran, Tehran, Iran
² Eindhoven University of Technology, Eindhoven, the Netherlands

sadeghfar.milad@ut.ac.ir

Abstract. Pedestrian wind environment assessment is becoming an essential part of the urban design process especially in dense urban areas due to its ability to address the wind comfort/safety/health concerns in an early phase. In this paper, high-fidelity computational fluid dynamics (CFD) simulations, validated with experimental data, are performed on eight different designs in a generic urban layout to study the impact of volume distribution on pedestrian wind environment in high-rise urban districts. The results show that the blockage effects of the high-rise buildings decelerates the wind in the streets parallel to the flow while accelerating the flow in the streets perpendicular to the flow. This effect is evident up to a two block distance upstream of the high-rises. Furthermore, it is shown that consequent rows of high-rises in the downstream of the first row facing the wind flow have little effect on the upstream pedestrian wind; however, they have a significant role in the extent of affected areas downstream. The findings of this study provide further understanding about the impact of different volume distributions on pedestrian wind environment in high-rise urban districts and clarify their effect on wind safety and comfort.

1. Introduction
Pedestrian wind environment assessment in dense urban areas is receiving increasing interest and becoming an essential part of the urban design since it can address the pollution dispersion concerns and ensure the health, safety and comfort of the population [1–3].

Reduced-scale wind-tunnel measurements and computational fluid dynamic (CFD) simulations are gaining more momentum in active wind utilization studies [4,5] as well as passive [6–9] including pedestrian wind environment assessment. This is due to the fact that on-site full-scale measurements are not an option for urban environments in the design stage. Wind-tunnel measurements while essential for CFD validation purposes, can be time-consuming and expensive for studying a large number of different designs. CFD simulations on the other hand, can be performed on a large number of designs in a fairly short amount of time and provide reliable whole-field high-quality results.

Even though some studies have been performed on pedestrian level wind comfort around isolated high-rise buildings and different urban morphologies, to the best of our knowledge, the impact of volume distribution on pedestrian wind environment in high-rise urban districts has not yet been comprehensively investigated. This study aims to provide further understanding about the impact of different volume distributions on pedestrian wind environment in high-rise urban districts. In this regard, different volume distribution scenarios are compared based on wind safety/comfort criteria.

The rest of the paper is as follows; CFD simulations are presented in Section 2 with studied geometries detailed in Section 2.1. Results and conclusions follow in Section 3 and 4 respectively.
2. CFD simulations

2.1. Studied geometry

Pedestrian wind environment is studied for eight different volume distributions (Figure 1) in the same layout (Figure 2). The low-rise urban blocks are assumed to be (width × length × height) 40×40×10 m³. High-rise buildings are assumed to be 25×25×100 m³; distributed in eight different configurations in the center of the low-rise blocks. For easier comprehension, all the eight studied distributions are compared against a reference case in which no high-rise is included (Figure 1a). All studied cases have the same plan area density $\lambda_p = \sum_{i=1}^{n} A_i / A_{total}$, in which $A_i$ is the area of the buildings in the center of the studied district (indicated with orange in Figure 1) and $A_{total}$ is the total constructed area. The dimensionless frontal area index of the studied cases is calculated as $\lambda_f = \sum_{i=1}^{n} A_{i,front} / A_{ref}$ in which the $A_{i,front}$ and $A_{ref}$ are the area of the buildings’ walls facing the wind flow in studied configurations and the reference case, respectively. The $\lambda_f$ of the cases is indicated in Figure 1.

Figure 2 depicts the studied urban layout. The dimensions, 1/400 scale and distributions of the blocks are selected with respect to the wind-tunnel measurements by Architectural institute of Japan (AIJ) [10], which is employed for CFD validation (see Section 2.4). In the figure, the orange blocks indicate the high-rise district surrounded by grey blocks indicating the low-rise surrounding buildings.

![Figure 1. Studied geometries: (a) reference case and (b-l) volume distributions](image)

2.2. Computational domain and grid

Simulations are carried out in a 1/400 reduced-scale 3D computational domain. The domain’s dimensions are selected based on the best practice guidelines for CFD simulations in urban areas [10,11] (Figure 3). The scale is selected with respect to the validation study (see Section 2.4).

A computational grid with 4.7 million cells is generated (Figure 4). Polyhedral cells are used to discretize the space. The grid resolution resulted from a grid-sensitivity analysis that will be presented in Section 2.4.
The minimum and maximum cell volumes in the domain are in the order of $10^{-5}$ and $10^{-4}$ m$^3$, respectively, resulting in a $y^+$ value of $\approx 290$. This $y^+$ value is well within the required range for the use of wall-functions, as is the case in the current study.

Figure 2. Studied urban layout. Scale = 1/400. Dimensions are in mm.

Figure 3. Computational domain for sample case D1 (Figure 1b).

Figure 4. Computational grid for sample case D1 (Figure 1b).

2.3. Boundary conditions and solver settings
Table 1 details the boundary conditions and solver settings of the CFD simulations. The settings are selected based on the best practice guidelines for CFD simulations in urban areas[10,11]. The turbulence model is selected based on a detailed validation study (see Section 2.4).

2.4. Solution verification and validation
A grid sensitivity analysis is performed to reduce the discretization errors and computational time. Three consecutively refined grids with a 2 refinement factor are generated (Figure 5). Simulations are performed for the D1 case as a sample (Figure 1b). Table 2 details the three grids’ number of cells and corresponding $y^+$ values.

Figure 6 shows the dimensionless velocity magnitude at $y = 0.005$ m height in 1/400 scale (corresponding to 2 m pedestrian level in full scale) along a horizontal line passing through the middle of a street parallel to the wind direction, normalized by the freestream velocity ($U_\infty = 5.2$ m/s) at surrounding blocks height (0.025 m in 1/400 scale corresponding to 10 m in full scale). A marginal difference is observed between the results of the fine and medium grids. Therefore the medium grid is selected for the rest of the study.
Table 1. Boundary conditions and solver settings of the CFD simulations

| Boundary conditions                        |
|-------------------------------------------|
| Inlet (#1 in Figure 3): ABL profiles of mean wind speed \( U \) (m/s), turbulent kinetic energy \( k \) (m\(^2\)/s\(^2\)) and turbulence dissipation rate \( \varepsilon \) (m\(^2\)/s\(^3\)) are imposed; |
| Outlet (#2 in Figure 3): Zero-gauge static pressure; |
| Top and sides (#3 in Figure 3): symmetry; |
| Ground (#4 in Figure 3), building walls: no-slip wall; |

| CFD approach                      |
|----------------------------------|
| steady Reynolds-averaged Navier-Stokes (RANS) |

| Turbulence model                  |
|----------------------------------|
| renormalization group (RNG) k-\( \varepsilon \) with standard wall-functions |

| Pressure-velocity coupling scheme |
|-----------------------------------|
| SIMPLE                            |

| Discretization order          |
|--------------------------------|
| 2\(^{nd}\) order               |

| Scaled residuals convergence criteria [-] |
|------------------------------------------|
| \( x, y, z \) velocity = \( 10^{-8} \), continuity, \( k = 10^{-7} \), \( \varepsilon = 10^{-6} \) |

| Aerodynamic roughness length, \( z_0 \) [m] |
|---------------------------------------------|
| 0.03                                         |

Figure 5. Computational grids for grid-sensitivity analysis. Detailed view of the center of the layout for (a) coarse, (b) medium and (c) fine grids.

Table 2. Grid sensitivity analysis details

| Grid     | Number of cells [-] | \( \bar{y} \) [-] |
|----------|----------------------|-------------------|
| Coarse   | 2,337,042            | 313               |
| Medium   | 4,674,084            | 290               |
| Fine     | 10,879,839           | 267               |

Figure 6. Grid sensitivity analysis: normalized velocity magnitude along the measurement line (indicated with red) in sample case D1 (Figure 1b) for coarse, medium and fine grids.

CFD model is validated against experimental wind tunnel measurements of the flow field near a high-rise building in a regular urban block performed by Architectural Institute of Japan (AIJ). The measurements were carried out in the atmospheric boundary layer wind tunnel of Niigata Institute of Technology with a cross section of 1.2 m \( \times \) 1.0 m (width \( \times \) height). The measurements were performed on a 1/400 reduced-scale model at a measuring height of 0.005 m above the wind tunnel floor (2 m above ground in real scale). Thermistor anemometer and split film probe were used to measure wind speed in 74 points around the high-rise building.
Based on the validation study, renormalization group (RNG) k-ε turbulence model with standard wall-functions results in the least deviation from the wind tunnel measurements in high-speed flow regions which are the focus of the current study; therefore this setting is selected for the simulations. A good agreement is achieved between the CFD and the experimental measurements and therefore the CFD model is deemed validated. For further details on the validation study please refer to Ref. [12].

3. Results

Figure 7 depicts the dimensionless velocity magnitude at y = 0.005 m height in 1/400 scale (corresponding to 2 m pedestrian level in full scale) for the studied cases.

![Normalized velocity magnitude at y = 0.005 m height in 1/400 scale](image)

**Figure 7.** Normalized velocity magnitude at y = 0.005 m height in 1/400 scale (corresponding to 2 m pedestrian level in full scale) for the studied cases.

According to Figure 7, in the near-field of the high-rise center, the blockage effects decelerate the wind in streets parallel to the flow. This is more evident in cases D4-6 (Figure 1d-f). The wind accelerates in streets perpendicular to the flow up to one block upstream, i.e. cases D2, D4 and D6 (Figure 1b,d and f). Highest wind accelerations occur in the streets next to the high-
rises and parallel to the flow. In these areas, wind safety measurements must be taken into account to avoid possible complications caused by the extreme wind velocities. These measurements include considering a canopy, podium or a permeable floor in the high-rise. Comparison of D2, D7 and D8 cases (Figure 1c,h and i) shows that consequent rows of high-rises in the downstream of the first row facing the wind have little effect on the amplitude of this acceleration in the near field.

As for the far-field, high-rise buildings cause wind acceleration in the streets perpendicular to the flow. This effect is evident up to a two block distance upstream of the high-rises. Blockage effects of the high-rise center drastically decreases the wind flow velocity in the streets leading to the center parallel to the flow. This decrease can be as high as 75% of the reference case. Consequent rows of high-rises in the downstream of the first row facing the wind flow have little effect on the upstream pedestrian wind; however, they have a significant role in the extent of affected areas downstream.

The studied cases cover two different $\lambda_f$ values of 1.6 (D2 in Figure 1b) and 2.8 (D3-8 in Figure 1c-i). More cases with different frontal areas need to be investigated to understand the effect of frontal area on the pedestrian wind environment.

4. Conclusions
High-fidelity computational fluid dynamics (CFD) simulations, validated with experimental data, are performed on a generic urban layout with eight different volume distributions in the centre to study the impact of volume distribution on pedestrian wind environment in high-rise urban districts. The following conclusions are made:

- The blockage effects of the high-rise buildings decelerates the wind in the streets parallel to the flow in the near-field and leading to the center in the far-field.
- The blockage effects of the high-rise buildings accelerate the flow in the streets perpendicular to the flow. This effect is evident up to a two block distance upstream of the high-rises.
- Consequent rows of high-rises in the downstream of the first row facing the wind flow have little effect on the upstream pedestrian wind; however, they have a significant role in the extent of affected areas downstream.

The findings of this study provide further understanding on the impact of volume distributions on pedestrian wind environment in high-rise urban districts, clarifying its effect on wind safety/comfort.

5. References
[1] Zheng X, Montazeri H, Blocken B. CFD analysis of the impact of geometrical characteristics of building balconies on near-façade wind flow and surface pressure. Build Environ 2021:107904.
[2] Zheng X, Montazeri H, Blocken B. Large-eddy simulation of pollutant dispersion in generic urban street canyons: Guidelines for domain size. J Wind Eng Ind Aerodyn 2021;211:104527.
[3] Blocken B, Carmeliet J. Pedestrian wind environment around buildings: Literature review and practical examples. J Therm Envel Build Sci 2004;28:107–59.
[4] Sahebzadeh S, Rezaeinha A, Montazeri H. Towards optimal layout design of vertical-axis wind-turbine farms: Double rotor arrangements. Energy Convers Manag 2020;226:113527.
[5] Sahebzadeh S, Rezaeinha A, Montazeri H. Impact of relative spacing of two adjacent vertical axis wind turbines on their aerodynamics. J. Phys. Conf. Ser., vol. 1618, 2020.
[6] Sahebzadeh S, Heidari A, Kamelnia H, Baghbani A. Sustainability features of Iran’s vernacular architecture: A comparative study between the architecture of hot-arid and hot-arid-windy regions. Sustain 2017:9.
[7] Sahebzadeh S, Dalvand Z, Sadeghfar M, Heidari A. Vernacular architecture of Iran’s hot regions; elements and strategies for a comfortable living environment. Smart Sustain Built Environ 2018.
[8] Heidari A, Sahebzadeh S, Dalvand Z. Natural ventilation in vernacular architecture of Sistan, Iran; Classification and CFD study of compound rooms. Sustain 2017:9.
[9] Sahebzadeh S, Rezaeinha A, Montazeri H. Vertical-axis wind-turbine farms: Impact of relative spacing on aerodynamic performance of double rotor arrangements 2019.
[10] Tominaga Y, Mochida A, Yoshide R, Kataoka H, Nozu T, Yoshikawa M, et al. AJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. J Wind Eng Ind Aerodyn 2008;96:1749–61.
[11] Franke J, Hellsten A, Schlunzen KH, Carissimo B. The COST 732 Best Practice Guideline for CFD simulation of flows in the urban environment: a summary. Int J Environ Pollut 2011;44:419–27.
[12] Sadeghfar M, Sahebzadeh S. Pedestrian wind environment surrogate modeling. 2021. Under review.