CFD Simulations of Wind Flow in an Urban Area with a Full-scale Geometrical Model

S. Liu 1, W. Pan 1, X. Cheng 1, H. Zhang 1, Z. Long 1, and Q. Chen 1,2*

1School of Environmental Science and Engineering, Tianjin University, Tianjin 300072, China
2School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA

SUMMARY
A CFD study explored the use of wind information from a meteorological station to simulate wind distribution in an urban community. The study constructed a full-scale urban model with building details from the meteorological station to the community. The wind distribution computed by the detailed full-scale model was compared with that computed by a simplified full-scale model, where the computational domain included only building structures in the proximity of the community and other regions were simplified with roughness according to the building information in the region. The mean wind velocity computed by the simplified full-scale model was 6.4% higher than the detailed full-scale model in the community area. The simplified full-scale model used only a half of the grid number compared with the detailed full-scale model. Thus, it can reduce computing time at least by a half. It was much easier to construct the geometrical model for the simplified full-scale model than that for the detailed full-scale model.

INTRODUCTION
The investigation of wind around buildings in an urban community in the lower part of the atmospheric boundary layer (0-200 m) is crucial for designing natural ventilation in buildings, pedestrian comfort, and air pollutant dispersion (Blocken 2015; Razak et al. 2016). The designs require wind flow information in and around the community.

With the development of computing resources and grid generation techniques, an increasing number of researchers have used computational fluid dynamics (CFD) in obtaining the information on the urban wind environment (Chen 2006). The traditional CFD models for urban flow simulation can be classified as meso-scale models (10 to 200 km in the horizontal direction) and micro-scale model models (100 m to 2 km in the horizontal direction) (Hang and Li 2010; Britter and Hanna 2003). The meso-scale models did not give detail flow information below the urban canopy layer because it uses some kinds of parametrization methods for the building details so the results are not useful for designing natural ventilation in buildings, pedestrian comfort, and air pollutant dispersion. The micro-scale models use meteorological data several kilometres away as inflow boundary and did not explicitly consider the effect from the meteorological station to the community, which implies the wind environment determined may not be accurate. Liu et al. (2017) proposed a full-scale model (typically 2 to 20 km long), which was between the micro-scale and the meso-scale model. The full-scale model is the same as a micro-scale model with explicitly constructed building details, but the computational domain extends from the community site to a meteorological station several kilometres away. They found that the full-scale model was able to produce the wind velocity distribution in the community by using wind information from a meteorological station that was 10 km away. It overestimated the wind velocity by only 20% compared with the measured data obtained on the rooftop of a building in the community site.

Although the full-scale model could calculate wind distribution at a community site with reasonable accuracy, it is very time consuming to construct all the building details in the computational domain and the computation requires a very significant computing resource. In fact, we only care about the flow distribution inside the community of interest. Therefore, it would be more practical to only construct building details inside the community area while treat the terrain from the meteorological station to the community as some kinds of resistance. The question now is how to manipulate the urban architectures between the meteorological station and the community. The meso-scale models generally use parametrization methods for the building details to emphasize the impact of geographical and meteorological conditions on urban wind environments (Liu et al. 2001). The parametrization methods can be classified into three categories, drag force parametrization, porosity concepts and roughness length approaches. Drag force parametrization and porosity concepts used wind tunnel experimental data in which roughness elements are simple cubes with certain distributions, whereas actual urban constructions are highly spatially inhomogeneous so the methods may not be suitable for actual urban environment. The roughness length method is a very simple approach to include the effects of urban terrain on wind speed and turbulence by using a prescribed aerodynamic roughness $z_0$ in contrast to the urban terrain, which is of great important in building and wind engineering applications. But its performance of representing the building structures in actual urban area for urban flow simulation has not been evaluated.

Previous studies have not determined how to correctly manipulate the urban architectures between the meteorological station and the community. This investigation examined a method of simulating intermediate terrain with roughness length in order to find a suitable simulating method for accurately calculating wind distribution in a community with limited computing resource.

METHODS
This section describes how a geometrical model could be constructed to present the region between a meteorological station and an urban community, the numerical model used to compute the wind distribution in the region, the mesh grid construction for the computational domain, and boundary conditions used, as well as how experiment was conducted to obtain wind information to validate the numerical results.
Geometrical Model
This investigation used a full-scale model that extends from an urban community (Tianjin University) to the closest meteorological station 10 km away in Tianjin, China. The computational domain was 12.6 km long, 5.4 km wide and 0.351 km high, as shown in Fig. 1(b). This detailed full-scale model was used as a baseline case. For comparison, this study further constructed a simplified full-scale model as shown in Fig. 1(c). The computational domain of this model was the same as the detailed full-scale model but only the building structures in the community site concerned and its immediate surroundings was explicitly constructed while other regions were simplified with roughness length. The immediate surroundings should be at least one additional street block in each adjacent direction around the community (Tominaga et al. 2008). The partitions to different roughness lengths in the computational domain were according to the building density and building height by using a satellite map.

Numerical Model
Our simulations used a commercial CFD program, ANSYS Fluent 14.0 (ANSYS 2011). The CFD program used the Reynolds-averaged Navier-Stokes (RANS) equations with the re-normalization group (RNG) $k-\varepsilon$ turbulence model (Yakhot and Orszag 1986) to solve the turbulent wind flow in the computational domain. The model performed well in simulating urban wind flows (Ferreira 2002). The governing transport equations were solved by using the finite volume method. The SIMPLE algorithm was used for pressure and velocity coupling of the transport equations, and the second-order discretization schemes were used for solving all the independent variables. For more detailed information about the CFD numerical technique, please refer to the program manual (ANSYS 2011).

Grid Arrangement and Mesh Generation
Gambit 2.4.6 was used to generate a discrete grid for discretizing the governing transport equations. Because of the complexity of the geometrical model, this study used a hybrid grid scheme with a tetrahedral grid, which can adapt to the geometric structures very well. There were 3,637 buildings scattered throughout the inner region of the detailed full-scale model, and the building heights ranged from 3 m to 117 m. As shown in Fig. 2(a), this study divided the inner region into 69 sub-regions according to building height and density. Streets and community fences were usually used as partition lines. The mesh was first generated for each sub-region and then combined for the whole computational domain. Because of our limited computing capacity and the possible impact of grid size on accuracy, the maximum grid size was 20 m near the meteorological station and was gradually reduced to 7 m at the community site in the lengthwise direction. The maximum grid size in the vertical direction was 10 m. The grid resolution along the

---

Figure 1. Geometrical models used in this study: (a) building details between an urban community (Tianjin University as shown in the square) and a meteorological station as shown by the star, (b) computational domain and roughness length used for the detailed full-scale model, and (c) computational domain and roughness length used for the simplified full-scale model.
perimeter without buildings was 20 m. This led to a total grid number of 8.9 million for the detailed full-scale model of Fig. 1(b). Fig. 2(b) shows the grid cells generated on the building surfaces and in the area without building details for the top right corner in Fig. 2(a) with triangular and quadrilateral mesh. The mesh type for the simplified full-scale model was similar to that for the detailed full-scale model. Because roughness length was used to replace the detailed building structure between the community and the meteorological station, the total grid number was reduced to 4.9 million for the simplified full-scale model.

Figure 2. Grid generation for the full-scale model: (a) sub-regions and (b) grid cells on the building surfaces and in the perimeter zone without building details.

Boundary Conditions

To simulate the wind flow, we should specify appropriate boundary conditions. The simulations used the wind data from the meteorological station in the upwind direction as the inflow boundary conditions. The meteorological station was located in 10 km southwest of the community and recorded wind information data hourly. This investigation selected the wind direction from the southwest for more than three hours as inlet boundary conditions to avoid the influence of wind direction mutation. The selected wind data was from 11:00 am to 2:00 pm on May 19, 2017 with a wind angle of 234°, 223°, 224°, respectively, as the boundary conditions. The wind angle means the direction where the wind blows in. The north wind is 0° (360°). The east wind is 90°. The south wind is 180° and the west wind is 270°. The corresponding wind speed during the period was 2.6 m/s. Thus, the western wind is 180° and the west wind is 270°. The corresponding wind speed and turbulence by using a prescribed aerodynamic roughness $z_0$ in contrast to the urban terrain. In ANSYS Fluent, wall functions with a roughness modification based on the equivalent sand-grain roughness height $k_s$ and the roughness constant $C_r$ can be used to reflect the influence of roughness elements on the urban wind flow field (ANSYS 2011). Therefore, the prescribed aerodynamic roughness $z_0$ can be defined in ANSYS Fluent after correct conversion to the corresponding $k_s$ and $C_r$ value. Blocken et al. (2007) proposed the conversion equation between the equivalent sand-grain roughness height and aerodynamic roughness $z_0$ as

$$k_s C_r = 9.793 z_0$$ (2)

Therefore, this equation can be used to describe different aerodynamic roughness lengths $z_0$ to take into account the influence of roughness elements on the wind flow field by using different $k_s$ and $C_r$ in ANSYS Fluent. Table 1 shows the $z_0$ value for different terrains (Wieringa 1992) and the corresponding $k_s$ and $C_r$ value converted through Eq. (2) for this investigation. Figure 1 shows the different roughness lengths used for the computational domain according to the building density and building height by using a satellite map. The surfaces of the buildings in the computational domain were assumed to be non-slip conditions for the wind.

Table 1. Roughness and case setup for different terrains (Wieringa 1992)

<table>
<thead>
<tr>
<th>Type</th>
<th>$z_0$ (m)</th>
<th>$k_s$ (m)</th>
<th>$C_r$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grass land</td>
<td>0.03</td>
<td>0.5</td>
<td>0.59</td>
</tr>
<tr>
<td>Few isolated obstacles</td>
<td>0.05</td>
<td>1.0</td>
<td>0.5</td>
</tr>
<tr>
<td>Low crops / Occasional large obstacles</td>
<td>0.1</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>Parkland / Shrubs / Numerous obstacles</td>
<td>0.5</td>
<td>1.0</td>
<td>4.897</td>
</tr>
<tr>
<td>Densely distributed mid-rise and high-rise buildings</td>
<td>1.0</td>
<td>1.0</td>
<td>9.793</td>
</tr>
</tbody>
</table>

Field Measurements for CFD Validation

In order to validate the CFD simulation results, this investigation used seven HOBO micro weather stations to measure the wind speed at the community from November 06, 2016 to May 31, 2017. The measurements were conducted at 2 m above the roofs of four buildings in the community. The height of the four buildings ranged from 9 to 20 m. The measuring frequency of the wind speed was every minute. The measured data was averaged hourly to be consistent with hourly data acquired from the meteorological station. The micro weather stations had a measuring accuracy of ±0.4% for wind speed if it was greater than 0.5 m/s. Fig. 3 shows the seven measurement positions inside Tianjin University.
RESULTS

Model Validation
This investigation first validated the CFD results by comparing our simulated wind speed with the data from a wind tunnel from the literature. For the wind tunnel case, the two CFD models discussed become the same one. Therefore, the experimental data from the wind tunnel can only validate the turbulence model, numerical algorithm, etc. used in the full-scale and the simplified full-scale models.

The wind tunnel data was collected at an urban complex in Niigata, Japan, with a scale of 1:250 (Tominaga et al. 2005). Scalar wind velocities were measured at a height of 8 mm above the wind tunnel floor (corresponding to a height of 2 m above the ground in actual scale) by multi-point thermister anemometers. Fig. 4(a) shows the building model of the urban area in the actual scale of 500 m long, 500 m wide, and 300 m high. A hybrid grid type was used in this study, with a tetrahedral grid in the inner area with building structures and a hexahedral grid in the outer regions. The total grid number was 5.6 million, as shown in Fig. 4(b). Table 2 lists the boundary conditions used in this investigation.

Table 2. Boundary conditions used in this study.

<table>
<thead>
<tr>
<th>Inlet flow boundary</th>
<th>Interpolated values of velocity and turbulence kinetic energy from experimental data for the inflow, ( \varepsilon = C_{\varepsilon} \cdot k \cdot dU/dz ) (( \varepsilon = P_k ))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outflow boundary</td>
<td>Outflow with zero pressure</td>
</tr>
<tr>
<td>Top surface</td>
<td>Solid mirror wall</td>
</tr>
<tr>
<td>Ground surface</td>
<td>Logarithmic law with roughness length ( z_0 = 0.024 ) m</td>
</tr>
<tr>
<td>Building surfaces</td>
<td>Logarithmic law for smooth walls</td>
</tr>
</tbody>
</table>

Fig. 4. (a) Geometric configuration of the urban area used for the CFD validation and (b) the grid built for the CFD simulations.

Fig. 5(a) compares the wind speed ratio computed in selected locations at a height of 8 mm above the wind tunnel floor with the corresponding experimental data. The wind speed ratio is defined as the ratio of the wind speed magnitude at each point of interest and the wind speed at the same height at the inflow boundary. The mean relative error of our calculation was 36.4% that seems high. As a comparison, Fig. 5(b) shows the computed results from Yoshihide et al. (2005) with the same experimental data (Exp: Experimental data; Code O: Overlapping structured grid; Code M: Structured grid; Code T: Unstructured grid). Although Yoshihide et al. did not show the quantitative error, the two computed results were similar. The comparisons indicate that the RNG k-\( \varepsilon \) model and numerical algorithm can be used to reasonably predict the airflow in an urban area, considering the complexity of the geometry and wind flow.

Comparison of Different Geometric Models
This study has further compared the performance of the detailed full-scale model with that of the simplified full-scale model as shown in Fig. 1 for predicting the wind speed on the university campus. The two models used the same wind velocity profiles from the meteorological station as the inlet boundary conditions and the same roughness length on the ground surface of the two sides. The simplified full-scale model used roughness length to represent the terrain of urban architectures between the meteorological station and the community.

Fig. 6 depicts the wind flow fields on the university campus computed by the two models at a height of 11 m above the...
The average velocity magnitude of the detailed full-scale model and simplified full-scale model was 1.10 m/s and 1.17 m/s, respectively. The simplified full-scale model calculated a wind velocity 6.4% higher than the detailed full-scale model. The difference may imply that the roughness length method cannot completely represent the building structures. The wind fields obtained by the two models were very similar as shown in Fig. 6. This is because the immediate surroundings of one additional street block in each adjacent direction around the community were retained. Therefore, it can be concluded that flow field inside the community was affected mostly by its immediate surrounding building structures rather than urban architectures between the meteorological station and the community, and the two models perform similarly.

Fig. 7 compares the simulated wind speed with that measured at the rooftop locations in Fig. 3. Our measurements found that although the hourly wind velocity and wind direction from the meteorological station was the same, measured data inside the urban community were varying because wind was inherently time-dependent. Therefore, the error bars of the experimental data represent the variation range of the wind magnitude at the seven measuring locations when the wind information from the meteorological station was the same as that set for the inflow boundary condition. The squares represent the mean wind speed of the experimental data. The results show that the computed wind speeds from the detailed and simplified full-scale model were within the actual wind speed variations at most of the positions. The average relative error compared with the mean experimental data for the detailed full-scale model and the simplified full-scale model was 45.4% and 47.9%, respectively. Both the models did not consider trees around the buildings, and only took into account the influence of the trees by roughness length of 0.03 m on the community, as shown in Fig. 1(b) and 1(c). In addition, there were differences between the geometrical model and the actual building structures even on the community. Therefore, we considered the computed results acceptable for such a complex case. The results obtained by the simplified full-scale model were very close to that of the detailed full-scale model so that it could be used to calculate wind distribution at an urban community.

This study also compared the computing time required by the two CFD models. All the simulations were performed on a work station with 24 cores and 128 Gb memory. The computing time for the detailed and simplified full-scale models was 41.8 h and 22.6 h, respectively. The simplified full-scale model used only a half of the grid number compared with the detailed full-scale model. Thus, it can reduce computing time at least by a half. In addition, the effort in constructing the geometrical model was much smaller for the simplified model, because it did not require detailed building information outside the community concerned. Therefore, it was a better choice to use the simplified full-scale model to conduct numerical simulating for urban wind environment.

DISCUSSION

Although the two full-scale models performed quite well for such a complex case study, they had some limitations:

- The detailed full-scale geometrical model was established with the aid of a satellite image because no public geometrical model was available. The roughness partition of the simplified full-scale model was based on the image. The model used approximations that may have affected the wind
profiles. We were unable to estimate the errors that may have resulted.

• This study used the RNG k-ε model to simulate the urban wind flow under steady-state conditions. However, wind constantly changes its direction and magnitude. Should a transient simulation method be used, it could improve the accuracy but would significantly increase the computing costs.

CONCLUSIONS
This investigation conducted CFD simulations of wind environment in an urban community to identify a suitable method in simulating intermediate terrain by using roughness length. The study led to the following conclusions:

• The velocity flow fields obtained with the two models were very similar. The flow field inside the community was affected mostly by its immediate surrounding building structures.

• The mean wind velocity computed by the simplified full-scale model was 6.4% higher than that of the detailed full-scale model in the community area. This may implies that the roughness length method cannot completely represent the building structures.

• The simplified full-scale model used only a half of the grid number compared with that used by the detailed full-scale model. Thus, the simplified model can reduce the computing time at least in half and the effort in constructing the geometrical model.

ACKNOWLEDGEMENT
The research presented in this paper was partially supported by the National Key R&D Program from the Ministry of Science and Technology, China, on “Green Buildings and Building Industrialization” through Grant No. 2016YFC0700500 and by the National Natural Science Foundation of China through Grant No. 51678395.

REFERENCES
ANSYS Inc. 2011. ANSYS Fluent 14.0 user’s guide. ANSYS Inc. Southpointe.