The effect of urban obstacles on the flow distribution at pedestrian area

M A Zainol¹, A A Razak²*, M F Mohamad²*, H F Pahroraji¹, A Sobri², H A Kasim¹ and A H Khalid¹

¹Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM), Pasir Gudang Campus, Johor Darul Takzim, Malaysia.
²Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM), Shah Alam, Selangor Darul Ehsan, Malaysia.

*azlirazak@salam.uitm.edu.my, faizal3744@salam.uitm.edu.my

Abstract: The pedestrian level is an important area where most of the human activities occur in this area such as walking, reading, and cycling. Along with the pedestrian activities, there are small street shops and presences of the vehicle at the street canyon where people park their cars in this area. This study aims to investigate the effect of the presence of obstacles such as vehicles at the street canyon on wind flow structure and ventilation performance. A computational simulation study is carried out to analyze the contribution of obstacles to pedestrian ventilation performance. A steady Reynolds-averaged Navier-Stokes equation with the realizable standard k-ε model is used as a turbulence solver. The result showed presences of obstacle at a pedestrian level will effect on wind flow structure where the flow becomes more chaotic compared with no presences of obstacles at a pedestrian level. Furthermore, the air change rate (ACH) is reducing as the number of obstacles increase thus will affect the ventilation performance at the pedestrian area. The presences of an obstacle such as vehicle and street shops will contribute to the wind flow structure and ventilation performable at pedestrians' level.

1. Introduction
In modern city development, modern building design becoming a good choice for developer in order to attract customer or buyer. However, modern city development needs to consider the wind condition or air ventilation entering the city to provide good city breathability especially at pedestrian level. Due to increasing of population and market properties, high-rise building is designed and build inside the city to overcome the issue. The development of high building will effect on the wind condition inside the city, therefore wind condition and pedestrians safety become important requirements in urban planning and design[1]. At designing stage, the developer needs to get an information about wind condition inside the city such as wind speed and air flow around the buildings. This information is important to architects and urban planners to design the layout and to determine the suitable high of the buildings. Unplanning city layout or building height might be effect on the city breathability, air pollution and high wind speeds are regularly presented at a pedestrian level that can be experienced as uncomfortable, or infrequently even perilous [2]. The objective of this study is to investigate the contribution of urban obstacles such as car effects and small shops on wind flow structure at the pedestrian level. The condition of wind inside the pedestrian area are depending on the building shape, size and other obstacles that might blocking the movement of wind at pedestrian area [3].
The high wind speed at a high level may be deflected down to ground level by tall buildings causing unpleasant and even dangerous conditions for pedestrians. The formation of vortex around buildings, between buildings or along avenues might accelerate wind speeds at the pedestrian level and giving rise to pedestrian discomfort. Besides that, the poor ventilation performance unable to disperse contaminated air from the urban area and lead to the accumulation of traffic fumes and other pollutants. [4]

2. Methodology

2.1 Numerical method
Wind tunnel experiment is commonly used to investigate wind flow and ventilation performance [5], [6]. The result generated by the wind tunnel experiment is generally acknowledged in the scientific and engineering application. However, due to the time consuming and difficulties of constructing and modified the model, the Computational Fluid Dynamics (CFD) method becomes a choice for researchers instead of wind tunnel experiments. CFD method will solve fluid flow problems through numerical approximation and algorithm using the computer. For example, W.D. Jannsen et al. performed a CFD analysis to study the pedestrian wind comfort around the buildings [7]. The 3D Reynold-averaged Navier-Stokes (RANS) equation [8] with the renormalization group (RNG) $k$-$\varepsilon$ turbulence closure model will solve the simulation alongside the continuity equation. The open source CFD software OpenFOAM 2.3 [9][10] is used to perform the CFD simulation [11] with the presumption of steady states, incompressible flow and isothermal condition during the simulation. The RNG $k$-$\varepsilon$ turbulence model is a two-equation turbulence model where similar to the standard $k$-$\varepsilon$ model but has been derived by using the renormalization Group method. This turbulence model varies from the standard $k$-$\varepsilon$ model just because of the alteration of $\varepsilon$ to the equation. The time-averaged continuity of governing equation and momentum in RANS equation can be derived as follows [12][13]:

$$\frac{\partial u_i}{\partial x_i} = 0$$

$$\frac{\partial}{\partial x_j}(u_i u_j) = \frac{\partial}{\partial x_j}[(v + v_t) \frac{\partial u_i}{\partial x_i}] - \frac{1}{\rho} \frac{\partial p}{\partial x_i}$$

The transport equation for turbulent kinetic energy (TKE) $k$ and dissipation rate of turbulent kinetic energy $\varepsilon$ can be written as below:

$$\frac{\partial}{\partial x_i}(k u_j) = \frac{\partial}{\partial x_j} \left[ (v + v_t) \frac{\partial k}{\partial x_j} \right] + v_t S_{ij} \frac{\partial u_i}{\partial x_j}$$

$$\frac{\partial}{\partial x_j}(\varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ (v + v_t) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon} \frac{\varepsilon}{k} v_t S_{ij} \frac{\partial u_i}{\partial x_j} - C_{\varepsilon} 2 \frac{\varepsilon^2}{k} - \frac{C_{\mu} k^2 (1 - \frac{\nu}{\eta})}{1 - \beta \eta^2}$$

Where $u_i$ and $u_j$ are mean and fluctuating velocity components in the $x_i$ and $x_j$ direction respectively. $p$ is the mean pressure and $\rho$ is the fluid density. An isotropic eddy viscosity and mean strain tensor can be present as:

$$v_t = \frac{C_{\mu} k^2}{\varepsilon}$$

$$S_{ij} = \frac{[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}]}{2}$$

2.2. Simulation Setup
The validation study was performed to ensure the accuracy of the simulation setup and the reliability of the simulation result. The result obtained from the simulation was compared with the wind tunnel
experiment done by Cheng et al. [14] and CFD simulation was done by Coceal et al. [5]. The dimensions of the domain for validation study is 0.1 m x 0.1 m x 0.1 m (L x W x H) where the height of the building is 0.025 m represent as h. The cubical obstacles were arranged in a square layout with a 25% packing density ($\lambda_p = 25\%$) as shown in Figure 1, packing density is defined as the ratio of building roof area over ground surface area. The streamwise wind velocity was set to be uniform at 2 m/s and cyclic condition was applied in a streamwise and spanwise direction to yield an infinity repeating model as shown in Figure 1. A slip condition was applied at top boundaries while wall function is imposed on the floor and building surface. Table 1 presents the boundary condition and Figure 2 illustrates the boundary condition set up in this study.

**Table 1. Boundary Conditions**

| Domain                | Boundary Conditions                           |
|-----------------------|-----------------------------------------------|
| Inlet, Outlet         | Cyclic                                        |
| Left, Right Sides     | Cyclic                                        |
| Floor                 | No-slip condition and wall function           |
| Building Surfaces     | No-slip condition and wall function           |
| Top                   | Free slip                                     |

**Figure 1. Boundary Conditions**

3. **Cases Study**

In order to investigate the effect of building on flow structure at the pedestrian level, a cubical building with dimension h x h x h where h is 0.025 m is arranged in square arrangement and an obstacle in cubical shape are placed at pedestrian level in order to study the contribution on obstacle at pedestrian level on pedestrian flow structure. The detail position obstacles (case study) are illustrated in Figure 2.
4. Result and Discussion

4.1 Validation
The mean wind velocity is normalized with $u_{ref}$, where $u_{ref}$ is taken at 4h from ground level and compared with simulation analysis performed by Coceal et al [5]. The point location of mean wind velocity is above the building surface as shown in Figure 3. Apart from the velocity profile, the flow structure has also been compared to ensure the accuracy of the simulation setup capturing the flow structure around the building as shown in Figure 4. The result shows the mean wind velocity has good agreement with mean wind velocity performed by Coceal et al [5]. Therefore, a similar simulation setup is used to investigate the contribution of obstacles at the pedestrian level.

![Figure 3. Mean wind velocity profile](image-url)
4.2 Simulation Result

4.2.1 Mean wind Velocity
The mean wind velocity profile was taken from the floor of the domain until the top of the domain for all three cases. Figure 5 shows the spatial average of normalized mean wind velocity, the mean wind velocity in the x-direction is normalized with \( u \) reference, where \( u \) reference is at 4h. There is no big difference between all three cases due to the addition of a car is slightly not affect the flow velocity in the domain.

![Figure 5](image)

**Figure 5.** Spatially average profile of mean streamwise velocity \( u_x \) normalized by reference velocity, \( u_{ref} \).

4.2.2 Mean Flow Structure
The airflow structure is an important especially at the pedestrian level because it determines pressure drag and the dispersive stress. It also shows the accumulation of the pollutant that been trap at the vortex. Based on the result obtained in Figure 6, the airflow speed has low velocity inside the street canyon compared at the outside street canyon where blue region shows low velocity speed meanwhile red region shows high velocity speed. However, the airflow structure for all cases was different due to different obstacles orientations; the presence of an obstacle at the pedestrian level will affect the formation of airflow structure at the pedestrian level. From Figure 4 (case 1), the winds enter the urban area without any obstacles at the pedestrian level, the airflow through smoothly at the street canyon and the vortex only formed at the back of the buildings. However when an obstacles places behind the upfront building, at the street canyon and both places, the vortex formation occurred at the second row of the buildings, there are small and big vortex formed at this area (case 2), meanwhile as the obstacle position changed to street canyon, the vortex formation occurred at the first row of buildings (case 3). As the number of obstacles added to the pedestrian level as shown in case 4, there is no vortex big formed at this area as formed in case 2 and case 3, however, the wind flow movement for this case are not smooth compared to case 1, case 2 and case 3. Based on the result, the number of obstacles and its positions will influence
the flow structure formed at the pedestrian area, this might be contributing to the effects of pollutant disperse from the pedestrian area.

**Figure 6.** Flow structure of mean field \((u,v)\) in a horizontal \(x-z\) plane close to the bottom surface at \(z = 0.05h\) for case 2, case 3 and case 4.

4.2.3 Air Change Rate (ACH)

**Figure 7.** Average Ventilation Rate.

The pedestrian level is an important area in the urban area where all the people activities occurring in this area such as walking, reading, and resting. Therefore, in this area, the ventilation performance must be good to provide sufficient fresh air toward pedestrians. The ventilation at the pedestrian level will contribute to ventilation performance. Based on Figure 7 case 1 has a higher ventilation rate compared to other cases due to no obstacle at the pedestrian level. Meanwhile, as the number of obstacles increases at the pedestrian level the ventilation rate will be decreased. Based on the result, the presence of obstacles such as cars and street shops at the pedestrian's level will contribute to the ventilation performance.
5. Conclusion
The presence of an obstacle such as street shops and vehicles inside the street canyon will be contributed to the formation of wind flow structure formed and effect on the capability of wind to disperse pollutants from the pedestrian area. Besides that, as the number of obstacles inside the pedestrian area increase the flow structure formed becomes more chaotic and the air change rate (ACH) is reduced. These problems might affect breathability and ventilation performance inside the pedestrian level. The architect and town planner need to consider the contribution of obstacle at pedestrians’ area at the design phase to avoid unpredictable ventilation problem after the development phase and at the same time will be an effect on the pedestrian's health.

Acknowledgement
The authors would like to express their sincere gratitude to Wind Engineering and Building Physics Center, Universiti Teknologi MARA (UiTM) and i-Kohza Wind Engineering for (Urban, Artificial, Man-made) Environment, Malaysia Japan International Institute of Technology (MJIIT) as the Research Institutes facilitating this research.

References
[1] I. Panagiotou, M. K. A. Neophytou, D. Hamlyn, and R. E. Britter, “City breathability as quantified by the exchange velocity and its spatial variation in real inhomogeneous urban geometries,” *Science of the Total Environment*, 2013.
[2] N. Z. Saddok Houda, R. Belardi, “A CFD Consol model for simulating complex urban flow,” *Energy Procedia*, 2017.
[3] S. H. R. E. Britter, "Flow and Dispersion in Urban Area," *Annual Review Fluid Mechanics*, 2003.
[4] Y. Huang, C. Long, J. Deng, and C.-N. Kim, “Impacts of Upstream Building Width and Upwind Building Arrangements on Airflow and Pollutant Dispersion in a Street Canyon,” *Environment. Forensics*, 2014.
[5] O. Coceal, T. G. Thomas, I. P. Castro, and S. E. Belcher, “Mean flow and turbulence statistics over groups of urban-like cubical obstacles,” *Boundary-Layer Meteorology*, 2006.
[6] O. Coceal and S. E. Belcher, “Mean winds through an inhomogeneous urban canopy,” *Boundary-Layer Meteorology*, 2005.
[7] W. D. Janssen, B. Blocken, and T. van Hooff, “Pedestrian wind comfort around buildings: Comparison of wind comfort criteria based on whole-flow field data for a complex case study,” *Building Environment*, 2013.
[8] Y. Jiang, C. Allocca, and Q. Chen, “Validation of CFD simulations for natural ventilation,” *International Journal of Ventilation*, 2004.
[9] M. A. Zainol, A. A. Razak, N. M. Ali, and Q. J. Kwong, "Effect of Upstream Building Configurations on Mean Wind Speed Ratio at Urban Pedestrian Level Using LES," *Science and Technology*, 2017.
[10] H.K Versteeg and W.Malalasekera, *An Introduction to Computational Fluid Dynamics*. 2007.
[11] C. J. Greenshields, “OpenFOAM,” , 2015.
[12] Y. Tominaga and T. Stathopoulos, "CFD simulations of near-field pollutant dispersion with different plume buoyancies," *Building and Environment*, 2018.
[13] B. Chen, L. Angeles, Y. Miao, and S. Wang, “Construction and Validation of an Urban Area Flow and Dispersion Model on Building Scales,” *Journal of Meteorological Research*, 2013.
[14] H. Cheng and I. P. Castro, "Near-wall flow over urban-like roughness," *Boundary-Layer Meteorology*, 2002.
[15] A. Abd Razak, A. Hagishima, N. Ikegaya, and J. Tanimoto, “Analysis of airflow over building arrays for assessment of urban wind environment,” *Building. Environment*, 2013.