Computational fluid dynamics analysis on the effect of flow distribution on pedestrian in urban area

Haszeme Bin Abu Kasim¹, Mohd Faizal Bin Mohamad²*, Siti Khadijah Alias¹, Ahmad Hilmi Khalid¹, Muhd Azhar Bin Zainol¹ and Nur Asyikin Binti Ahmad Tajudin²

¹Faculty of Mechanical Engineering Universiti Teknologi MARA (UiTM) Cawangan Johor, Pasir Gudang Campus 81750 Masai, Johor Darul Takzim, Malaysia.
²Faculty of Mechanical Engineering Universiti Teknologi MARA (UiTM), Shah Alam Campus 40450 Shah Alam, Selangor, Malaysia.

*Corresponding author E-mail: mfaizal01@gmail.com

Abstract. Buildings in urban area consist of various heights and building arrangements. Meanwhile, the pedestrians at the area must be considered for their wind comfort and wind safety. This is because, near high-rise buildings, high wind velocities are often introduced at pedestrian level that can be experienced as uncomfortable or even dangerous. Nowadays feasibility Computational Fluid Dynamics (CFD) for the assessment in wind engineering such as wind comfort and safety of pedestrian are very demanding. Hence, this study is conducted using CFD to analyses the effect of different building heights on the wind flow distribution towards pedestrian. The study was done on two different cases based on two different buildings (uniform height and non-uniform height) with squared and staggered arrangements. The turbulent wind over the pedestrian was acquired by solving the 3D steady state Reynold’s-Average Navier-Stokes (RANS) equations using RNG k-ε turbulence closure model. The results show that changes on the height of the buildings varies from the center of the urban canyon based on the vorticity of the flow distribution.

1. Introduction

In urban city centers, wind nuisance has become a more obvious problem as favored high-rise buildings are developed. Urban geometrical varieties do affect the wind flow whereby it alters the overall wind velocities [1]. This affects the strolling activity in the city. Hence, it is recommended to conduct a study on wind comfort. Responsible authorities like architects and urban planners should consider the wind flow pattern around high-rise buildings to create more pleasant and safer buildings. The environment nearby high-rise buildings can exert uncomfortable high wind velocities on pedestrian. Despite, due to difference in generally accepted comfort criteria in different regions, there is no commonly international standard for wind comfort criteria [2].

When a wind gust strikes a high-rise building, the wind is prone to be deflected towards the ground resulting to high wind velocities on the windward side and at the corners of the building. Occurrences on pedestrian area often associated with three types of flow which are vortex flow between buildings near to the ground level, descending airflows passing around leeward building corners and airflows passing through openings (passages) at ground level connecting the windward and leeward sides of buildings [3][4][5].

Generally, the friction and barrier produced by the urban canopy can lower the urban wind speed. The variations of building heights form an irregular surface area whereby these buildings apply a
strong frictional drag of air moving over them leading to turbulence with quick sudden change in wind speed and direction [6][7][8].

It is inferable that the airflow within urban area is different from rural areas corresponding to the diversity of dispersion and concentrations of air pollutants for different wind directions and building shapes [9][10].

Based on past studies on urban wind, CFD is preferable to simulate the flow distribution due to its advantages and overwhelming demand to assess the wind comfort and safety of pedestrian [11][12]. To investigate the effect of flow distribution, it is examined throughout a standard condition whereby the flow is incompressible, turbulent, steady-state and isothermal where the temperature remains constant. Despite the Large-Eddy Simulation (LES) which based on a local filtering, the equations are unsteady to be solved and RANS is more preferred as less computational effort is required that gives time averaged mean value for velocity field [13][14].

In conclusion, CFD tool can be used to simulate the flow distribution in urban area. This study is intended to evaluate the distribution flow pattern on pedestrian within urban street canyon and to analyze the effect of high rise buildings in urban area using ANSYS Fluent software. The study is done on two different cases which are uniform height and non-uniform height based on two different building arrangements of squared and staggered. Table 1 shows for the case conditions and Figure 1 shows for the Schematics of the building geometry. Validation is done to ensure the reliability of the simulation.

2. Methodology
The numerical approach of CFD was used to simulate the flow distribution using ANSYS Fluent software. This simulation starts with pre-processing, solving and post-processing.

2.1 Pre-Processing Stage
In this stage, the geometries of the buildings were designed using CATIA V5R20 based on urban-like cubicles. The domain was then generated in ANSYS Design Modeler with domain size of 4H where H was the building height of 25mm. The boundary conditions were then assigned on the inlet, outlet, lateral sides, bottom, top and buildings.

In order to do meshing for the building geometries, tetrahedral mesh was selected due its appropriate for unstructured geometry. Refer to Figure 2 for the unstructured tetrahedral meshing. Meanwhile for the sizing of meshing, it was set up to fine mesh to get the convergence and accurate results such as Figure 3. The parameters of minimum orthogonal quality must be between 0.15 to 1.00 and maximum skewness between 0 to 0.94 had to be achieved. Refer to Figure 4 for the minimum orthogonal quality and Figure 5 for the max-inum skewness respectively.

| Case | Condition                                      |
|------|-----------------------------------------------|
| 1    | Uniform height with staggered arrangement     |
| 2    | Uniform height with squared arrangement       |
| 3    | Non-uniform height with staggered arrangement |
| 4    | Non-uniform height with squared arrangement   |
2.2 Solver (Fluent)
In this study, the solver used was Fluent. This stage involves the inputs required to simulate the flow distribution. The viscous model and boundary conditions were applied here.

Figure 1. Schematics of the building geometry and their imposed boundary condition. (Upleft for case 1, upright for case 2, downleft for case 3 and downright for case 4)
2.2.1 Viscous Model
The Reynold’s Averaged Navier-Stokes (RANS) equation was used to solve the exact unknown solution because it was already sufficient for general engineering application (quick and reasonable) compared to Large Eddy Simulation (LES) which requires more time, memory and computing power. The turbulence closure model of RNG k-ε was then selected as it is suitable for complex flow of moderate swirl, vortices and locally transitional flow.

2.2.2 Boundary Conditions
For boundary conditions, the flow was maintained by streamwise pressure gradient of -0.874. At the top, free-slip boundary condition was applied. Meanwhile, at the wall of the buildings and floor, no-slip boundary conditions were applied. Interface meshing was then used to match the non-conformal walls and to impose the translational periodic on streamwise and lateral faces.

Figure 2. Unstructured tetrahedral meshing on case 1.

Figure 3. Sizing of the meshing.
2.2.3 Calculation Cycle

To acquire accurate result, the iterations done must be converged. For this simulation, the number of iterations was set to 20,000. This was to monitor and ensure the stability of the simulation.

2.3 Post-Processing

All the data were obtained after the iterations were done. The results of streamwise velocity and height (vertical direction) were obtained and exported to Excel Spreadsheet to get the velocity profile from the top of the building until top of the domain. The wind velocity streamline plot was also obtained to observe the wind flow pattern at the pedestrian height of 0.05H.

3. Result and Discussion

3.1 Results Case 1 (Validation)

To ensure the reliability and accuracy of the simulation, a validation was done on Case 1. This case was done based on the direct numerical simulation of turbulent flow over regular arrays of urban-like, cubicle obstacles [15]. The validation of velocity profile was compared as per Figure 6.
Figure 6. Validation of profile on mean streamwise velocity, $U_x$ normalised by reference velocity, $U_{ref}$.

The velocity profile is taken from the top of the building until to the top of the domain. It shows that the velocity profile is in good agreement with the current study. Hence, the simulation was considered reliable.

3.2 Results Case Study (Case 2, Case 3 & Case 4)

Urban obstacles such as buildings can exert a relatively large drag force on the atmosphere. In the urban canopy sublayer, the flow at a specific point can be directly affected by local obstacles and in the level of roughness sublayer, the flow is still adjusting itself because of the previous obstacles. In the case of non-uniform of heights, as the flow is closer to the buildings, individual velocity profiles deviate as they directly respond to the real surface due to the spatial irregularity.

For shear stress, at position of uniform height as shown in Figure 7, the maximum shear stress occurs approximately at the height of the obstacles. Lower than this height, the shear stress carried by the wind decreases as the buildings take up part of stress through the drag forces on them. Meanwhile for non-uniform height as shown in Figure 9 and Figure 11, the maximum shear stress occurs at the approximate-ly at the height of the highest building and de-creases to zero shear stress like uniform-height buildings.

It shows that in Figures 8, 10 and 12, at windward side of the building, when the ambient wind strikes the bluff body of the building, a stagnation point is formed in front of the building with a separation lines creating divergence alongside strong uplift and conver-gence at the leeward side.
Figure 7. Velocity streamline of case 2 from side view (y-axis).
Figure 8. Velocity streamline of case 2 from top view (z-axis).

Figure 9. Velocity streamline of case 3 from side view (y-axis).
Figure 10. Velocity streamline of case 3 from top view (z-axis).

Figure 11. Velocity streamline of case 4 from side view (y-axis).
The horseshoe vortex is then developed down the bluff body for all cases. It is formed when producing a wake region which is characterized by periodic vortex formation with a circular cavity. In these areas, the pressure is lower than the stagnation point pressure. This cavity does not mix efficiently with other air and tends to circulate within the area. The effect of wake weakens as its distance from the building increases.

4. Conclusion & Recommendation

A compact city with geometrical inhomogeneity such as irregular building arrangements and heights means higher building density, higher population density, higher anthropogenic heat and reducing the wind flow. Despite this kind of buildings can be aesthetically pleasing, the reduced wind flow can lead to urban warming (2-3°C higher), poor air quality with pollutants accumulation, poor visibility due to dust, smokes and fogs trapped in urban canyon and thus reduced in productivity. In contrast to the rural area, wind cannot penetrate into a high-rise compact city due to large canopy drag [16]. The buoyancy driven flows along building walls (wall slope flows) will become dominant due to building heights and large wall areas.

To enhance the natural ventilation performance, a wind catcher or wind tower can be used to direct the atmospheric wind into urban street canyon [17]. Besides, through-building gap can be proposed to provide better ventilation thus making pedestrian to feel better and more comfortable.

Figure 12. Velocity streamline of case 4 from top view (z-axis).
Acknowledgement
The authors acknowledge technical and financial support from UiTM Shah Alam to publish this paper. Besides that, special thanks go to authors group members of Paper to Publish (P2P) from UiTM Pasir Gudang because successfully make the journal paper become reality.

References

[1] K. M. Al-Obaidi, M. Ismail, and A. M. Abdul Rahman, “A study of the impact of environmental loads that penetrate a passive skylight roofing system in Malaysian buildings,” Frontiers of Architectural Research, vol. 3, no. 2, pp. 178–191, 2014.

[2] H. Montazeri, B. Blocken, W. D. Janssen, and T. van Hooff, “CFD evaluation of new second skin facade concept for wind comfort on building balconies: Case study for the Park Tower in Antwerp,” Build. Environ., vol. 68, pp. 179–192, 2013.

[3] J. Li and M. Donn, “The influence of building height variability on natural ventilation and neighbor buildings in dense urban areas,” in Proc. Building Simulation 2017, 2017, pp. 2805–2813.

[4] M. Carpentieri, A. G. Robins, and S. Baldi, “Three-dimensional mapping of air flow at an urban canyon intersection,” Boundary-Layer Meteorol., vol. 133, no. 2, pp. 277–296, 2009.

[5] M. S. Fadl and J. Karadelis, “CFD Simulation for Wind Comfort and Safety in Urban Area: A Case Study of Coventry University Central Campus,” Int. J. Archit. Eng. Constr., vol. 2, no. 2, pp. 131–143, 2013.

[6] S. L. Ceccio, “Friction Drag Reduction of External Flows with Bubble and Gas Injection,” Annu. Rev. Fluid Mech., vol. 42, no. 1, pp. 183–203, Jan. 2010.

[7] P. Sachs, “Wind Forces in Engineering,” Wind Forces Eng., pp. 306–353, 1978.

[8] J. F. Manwell, J. McGowan, and A. Rogers, Wind Energy Explained: Theory, Design, and Application. 2009.

[9] M. A. Hart and D. J. Sailor, “Quantifying the influence of land-use and surface characteristics on spatial variability in the urban heat island,” Theor. Appl. Climatol., vol. 95, no. 3–4, pp. 397–406, Mar. 2009.

[10] I. D. Stewart and T. R. Oke, “Local Climate Zones for Urban Temperature Studies,” Bull. Am. Meteorol. Soc., vol. 93, no. 12, pp. 1879–1900, Dec. 2012.

[11] H. Liu, P. Li, and J. Van Lew, “CFD study on flow distribution uniformity in fuel distributors having multiple structural bifurcations of flow channels,” Int. J. Hydrogen Energy, vol. 35, no. 17, pp. 9186–9198, 2010.

[12] M. Goutham, M. Shanmugam, M. Mihai, K. Maninder, K. Sid, and G. Ephraim, “Validation of computational fluid dynamics methodology used for human upper airway flow simulations,” J. Biomech., vol. 42, no. 10, pp. 1553–1559, 2009.

[13] Y. Tominaga, T. S.-J. of W. E. and Industrial, and U. 2011, “Tominaga Y, Stathopoulos T. CFD modeling of pollution dispersion in a street canyon: comparison between LES and RANS. J Wind Eng Ind Aerodyn 2011; 99:340e8.,” Elsevier, no. 99, pp. 340–348, 2011.

[14] M. L. Shur, P. R. Spalart, M. K. Strelets, and A. K. Travin, “A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities,” Int. J. Heat Fluid Flow, vol. 29, no. 6, pp. 1638–1649, 2008.

[15] C. Yan, W.-X. Huang, S.-G. Miao, G.-X. Cui, and Z.-S. Zhang, “Large-Eddy Simulation of Flow Over a Vegetation-Like Canopy Modelled as Arrays of Bluff-Body Elements,” Boundary-Layer Meteorol., vol. 165, no. 2, pp. 233–249, Nov. 2017.

[16] J. Barlow, “Urban climate,” 2014.

[17] K. J. K. Ishugah T F, Li Y, Wang R Z, “Advances in Wind Energy Resource Exploitation In Urban Environment,” Elsevier, vol. 47, pp. 1–5, 2017.