Investigation of pollutant dispersion in urban and spatial variation in concentrations on building model

H.M. Heravi [a], M.Kahrom [b] , M.A. Moharreri [a], F.S. Abbasi [a]

[a] Department of Mechanical Engineering, Islamic Azad University, Mashhad Branch, Iran

[b] Department of Mechanical Engineering, Ferdowsi University, Mashhad, Iran

heravihm@mshdiau.ac.ir

mohsen.kahrom@yahoo.ir.uk

Amir168113@yahoo.com

Fakhri_abasi@yahoo.com

Abstract, *in crowded cities the air environment can be very different from that rural area. The wind transports clean external air into urban street network. Ingress of outdoor pollution is dependent on wind-induced and contaminant concentration distributions on the building surfaces. This study considers models and investigates natural ventilation by variation on height and area density. The objective of this study is to investigate the air ventilation impacts of the high-rise buildings in complex building clusters and area density (area building divide to total area). The models specifications are [area density, height].The research method employs the numerical algorithm of computational fluid dynamics (CFD-code). This is based on finite volume discretization of equations of motion, an unstructured grid volume made of prisms or tetrahedral cells various matrix- inverting routing sand. The realized k-ε turbulence model is used to solve the steady state wind field in urban setting and investigating pollutant dispersion inside the street canyon. The model sittings of validation study accomplishes by comparing the simulation wind field around building blocks with another research. This paper assumes a uniformly distributed pollutant release at model level to simulate a source generated by vehicles. The results show that the sensitivity of this region to the area density of array and height. Also indicates that at 2 m above ground, concentration pollutants increase inside the street canyon when height of the high-rise buildings increases. While this reduction of air ventilation was anticipated. These results clearly have important suggestions for urban planning.*

Key words: natural ventilation, area density, pollutant dispersion, realized k-ε

1 INTRODUCTION

Land prices in a commercial city are a material proof for the subject under discussion that increases rapidly year to year [1]. To solve this problem developers often increase their building height. This fuels the increasing trend towards taller and taller buildings. The buildings can block the ‘‘fresh air’’ which ventilate the urban and is important for pollutant dispersion in the street canyon. People are exposed in pedestrian precinct everyday and this affects their health. In 2003 Sotiris and Bernard et al. [2] provided an overview of modeling techniques and available software for assessing air quality in street canyons. Within that framework, relevant experimental and theoretical studies were also briefly discussed.

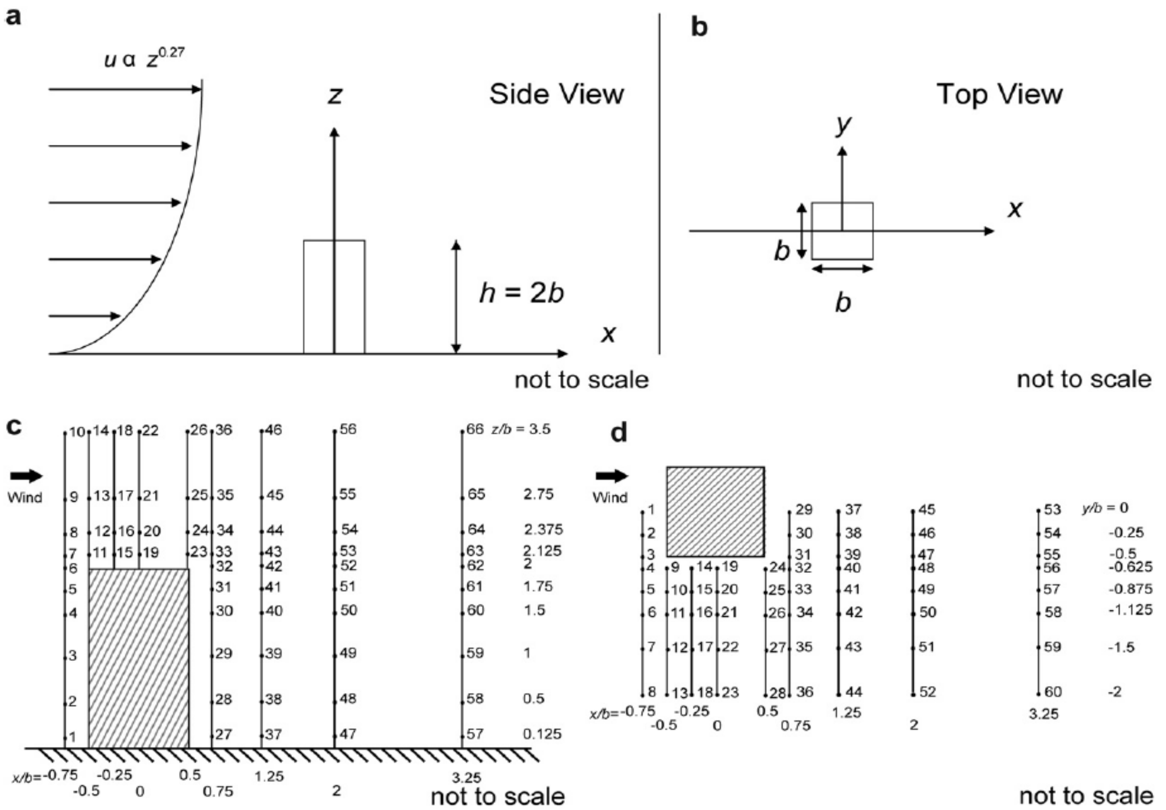
Using numerical CFD codes have many advantages for optimizing wind tunnel experiments [3]. Some studies use FLUENT to simulate the wind flow and pollutant dispersion within an isolated street canyon and Find that wider streets and lower buildings are favorable to pollutant dilution within canyons. In 2004 Kim and Baik [4] developed Three dimensional CFD model with the RNG k_ε turbulence used to investigate the effects of ambient wind direction on flow and dispersion around a group of buildings, The results are capable. This is founded that exclude for the case where the ambient wind direction is perpendicular to the buildings; pollutants are trapped in the entrance vortex, thus showing high concentrations. In 2010 Dipankar and Gautam et al. [5] offer to investigate in general, the effect of spacing between the cylinders on vortex dynamics and the resulting wake structures for the separation ratio of 1.2, 2, 3, and 4. The existence of quasi-periodic regime and low-dimensional chaotic behavior has been revealed with two/three-cylinders and they understood the scenario when the number of cylinders is five. They have used a finite volume method to simulate and analyze this important problem.

Several simplified simulations were carried out to study the effect of emission rate and source design on flow structures and pollutant dispersion within the canyon. In 2008 Tetsuro Tamura [6] studied strongly on the hill’s surface condition, such as roughness effects, as well as curvature effects. A rough surface was simulated by arraying roughness blocks on the ground. In 2007 Wang and McNamara [7] studied both CFD and wind-tunnel investigations of the variations of flow and dispersion patterns at or near an isolated street intersection when the angle between the upwind and crosswind street is subjected to a change from an orthogonal street intersection. In 2009 Jie and Zhiguo et al. [8] studied the effects of various factors influencing pollutant dispersion simulation. The simulation schemes involved cases reflecting the effects of different factors influencing pollutant dispersion. In 2010 Parham and Mirzaei et al. [9] investigated the pedestrian ventilation system (PVS), they proposed ventilate building canopy under various atmospheric stability conditions: stable, neutral, and unstable. In 1998 Ishihara and Hibi et al. [10] placed a square building model with scale ratio of 2:1:1 (height: width: depth) within a turbulent boundary layer to study the steady-state wind field around a cube. In 2000 Huang and Akutsu et al [11] studied a two dimensional air quality numerical model in an urban street canyon based on the k_ε turbulent model and atmospheric convection diffusion equation was developed. In 2008 Yoshihide and Akashi et al. [12] described guidelines proposed by the Working Group of the Architectural Institute of Japan (AIJ). This study investigates the air ventilation and pollutant dispersion rate in a street canyon when high-rise buildings are situated upstream of the flow and effect of area density.

2 Methodology

2.1 Model validation

To validate the model, the simulation results are compared with wind tunnel data. The wind tunnel experiment is described by Ishihara and Hibi (1998). In the experiment, a square building model with scale ratio of 2:1:1 (height: width: depth) is placed within a turbulent boundary layer and the wind profiles approaches a power law profile with index 0.27 from the upstream. The dimensional of the model building was 0.08 m in width and length (b) and 0.16 m in height (h). The turbulence statistics were measured at the centerline of a vertical cross-section and on horizontal planes at 1/16 (z/b=0.125) and 10/16 (z/b =1.25) of the building height. 66 and 60 measurement points in vertical and in horizontal directions are set to measure the average wind velocity in each direction of 3-dimensional space and the standard deviation of fluctuating wind velocities. The locations of the measurement points in the wind tunnel test are illustrated in Fig1. The measurement point data is compared with the simulation results. Figure (3) compares the experimental and calculated scalar wind velocity in vertical cross-section along the centerline.
The x-axis represents the measured scalar wind velocity in the wind tunnel experiment while y-axis represents the simulated scalar wind velocity in the CFD model. The wind speed is normalized according to the wind speed at the same height when there is no building. If it is limited to the region where the wind speed has increased, which is important in the evaluation of the pedestrian wind environment. It is predicted within an accuracy of approximately $\pm$ 10%. However, in the weak wind region behind the building, the wind speed ratio is evaluated lower in the calculation than in the experiment. The distribution of average wind velocity u on the vertical cross-section along the centerline of the building is illustrated in Figure (4). Wind velocities are plotted transversely using this as the origin. (Positive values are plotted on the right side of the measuring line, and negative values on the left side.) The calculated values agree fairly well with the experimental values. The simulated values and the experimental values agree relatively.



F

Figure1: Configuration of experimental model and approaching wind speed profile from (a) side view and (b) top view. Measurement points in wind tunnel test at (c) centerline on a vertical cross-section and (d) on horizontal planes at 1/16 (z/b = 0.125) and 10/16 (z/b = 1.25) of building height.

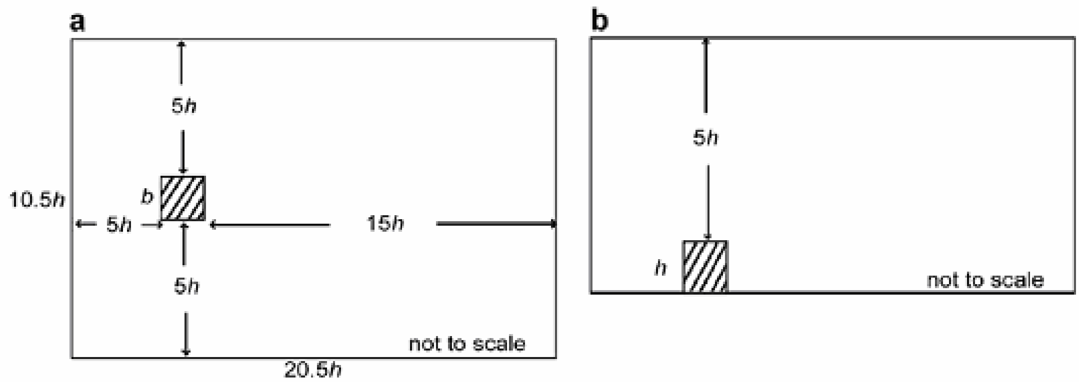


Figure 2: Validation case geometry from (a) top view and (b) side view.

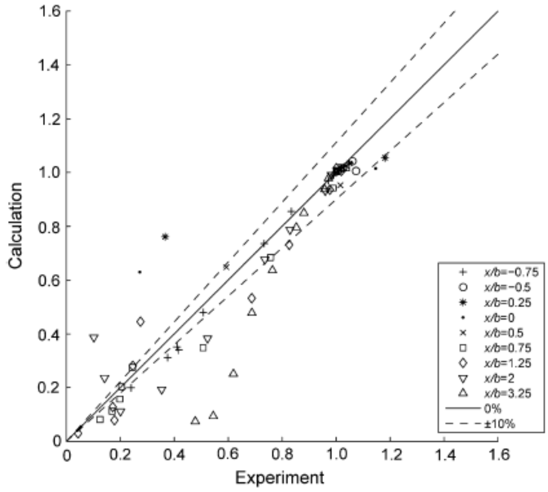


Figure 3: Comparison between wind tunnel data and simulation results (Vertical cross section along centerline of building).

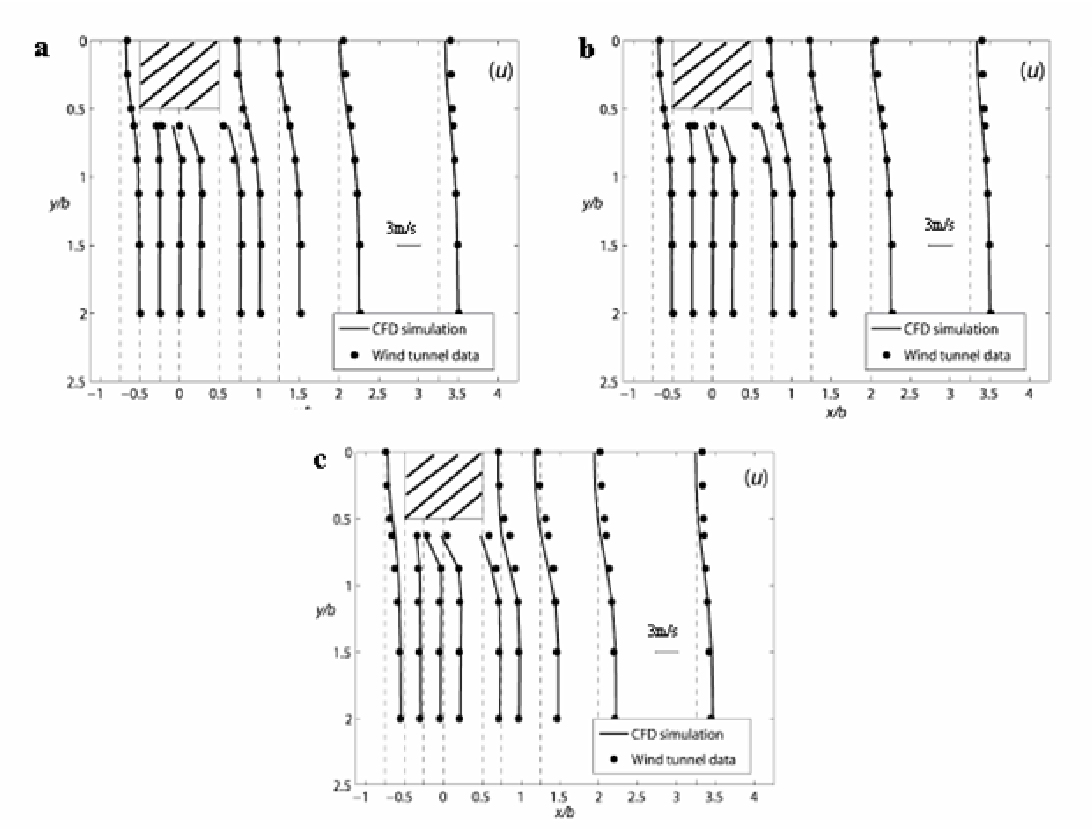


Figure 4: Comparison between wind tunnel data with simulation results by RKE. Vertical cross section along centerline of building on (a), on horizontal plane at z = 1.25b on (c) and on horizontal plane at z =0.125b on (e)

2.2 CFD model configuration

This paper employs computational fluid dynamics numerical algorithms (CFD – FLUENT version 6.3.26) to simulate the steady state wind field in an urban setting and to investigate pollutant dispersion inside the street canyon. The validation of model settings was carried out by comparing the simulation wind field around a single building block to wind tunnel data. In the validation simulation, a square shaped building with scale ratio of 2:1:1 (height (h): width (b): depth (b)) was located in the computational domain in the CFD model. The domain size is designed according to the CFD guidelines (Franke and Tominaga et al. [12,13]). where H is the height of the target building. Based on the recommendations, the building in this validation case is located 5h away from the inflow boundary and 15h away from the outflow boundary. It is also 5h away from the lateral and top boundaries. The geometry is illustrated in Fig. 2(a), (b). A second order discretization scheme for the pressure calculation and second order upwind discretization schemes for the momentum, turbulent kinetic energy and turbulent dissipation rate calculations are utilized. The smooth surface wall condition is adopted for all boundaries. The inflow boundary condition is interpolated from the experimental approaching flow with ε =($(c_\mu)^{1/2}$) k.dU/dz.. The zero gradient condition is at the outflow boundary. The realizable k _ ε turbulent model is utilized.

3. Governing equations and turbulence models

Continuity equations and the averaged Navier–Stokes equation can be used as the basis for describing the motion of turbulent airflow:

$$\frac{\partial \overline{v_i}}{\partial x_i} = 0 \qquad (1)$$

$$\bar{v}_j \frac{\partial \overline{v_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(v\frac{\partial \overline{v_i}}{\partial x_j} - \overline{u_i uj}\right). \qquad (2)$$

Where $\bar{v}$ and $\overline{p}$ represent fluid mean velocity and pressure, respectively; ρ is the fluid density and ν is the kinematics viscosity.

The standard k _ ε turbulence model [13] is used here in the form;

$$\overline{v}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{v_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + v_t\left(\frac{\partial \overline{v_i}}{\partial x_j} + \frac{\partial \overline{v_j}}{\partial x_i}\right)\frac{\partial \overline{v_i}}{\partial x_j} - \varepsilon. \quad (3)$$

$$\overline{v}_j \frac{\partial \varepsilon}{\partial x_j} = -\left(c_2 \frac{\varepsilon^2}{k + \sqrt{v\varepsilon}}\right) + \frac{\varepsilon}{k} + c_1 s_\varepsilon + Diff(\varepsilon). \quad (4)$$

In Esq. (3) and (4), k is the turbulence kinetic energy and ε is the turbulence dissipation rate.

4 CFD model configurations

In order to investigate the air ventilation impacts [14] as affected by the alignment of high-rise buildings to a downstream street canyon, 2000*1500*480 computational domain is set in the CFD simulation where the height of the low-rise buildings is 20m. The inflow boundary and outflow boundaries are set 32H and 100H away from the buildings, respectively, and the lateral size of the computational domain is extended 32H from the outer edges of the buildings. The top boundary is set 31H above the buildings. A 6*3 block of low-rise buildings with height H form a parallel street canyon with a constant aspect ratio of building height to street width equal to about 0.8 is defined. 6 high rise buildings with height H_{var} are located 5H away in the upstream direction, where H_{var} varies in different cases. The values of H_{var} are 0H, H, 2H and 4H in reference case, case H, case 2H and case 4H respectively. We consider a uniform source for pollution inlet. The building geometry is illustrated in Figure 5. Wind profiles approaches a power law profile with index 0.27 from the upstream. We use UDF in fluent to consider this profile for air inlet velocity that has illustrated in figure 6.

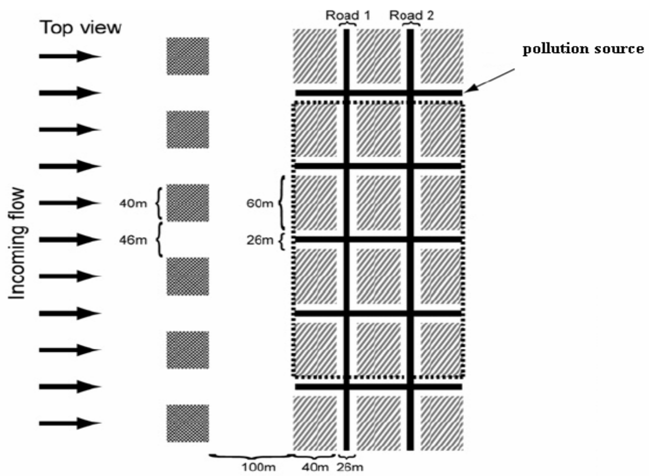


Figure5: Top view of building geometry. Black arrows represent wind flow direction.

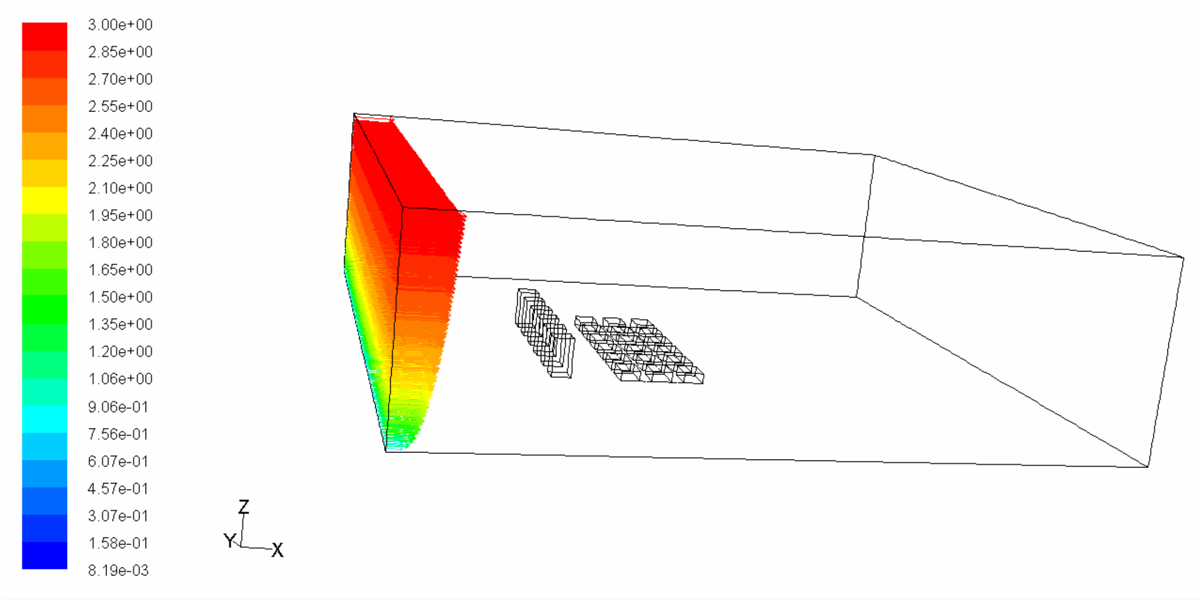


Figure 6: wind speed profile for inflow boundary.

5 Air ventilation

The key purpose of this study is to investigate in what form, building alignments are best for design and planning so as to achieve better wind penetration into, and hence air ventilation of, the city, especially at the pedestrian Levels. The focus of this methodology is to introduce the best case for height of rise buildings in the complex buildings and width of the streets to indicate and to allow better layout design of developments and planning within the urban fabric.

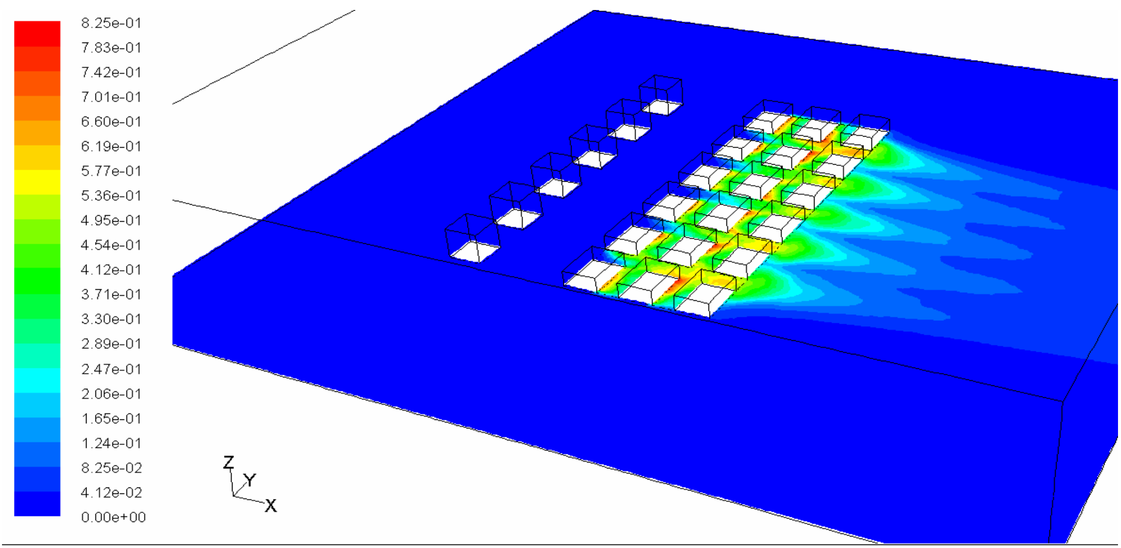


Figure7: NO2 concentration at pedestrian level (2 m above ground)

6 Results and discussion

Figure 7, illustrates the NO2 concentration at the pedestrian level (2 m above ground), figure8 shows the NO2 mol fraction at the pedestrian level in cases: reference, H, 2H, and 4H. Obviously, in the downstream area it is lower in reference case and higher than other three cases. Without the upstream buildings blocking the incoming flow, the approaching wind can directly flow into the street canyon, and hence the velocity in the downstream area is higher in reference case.

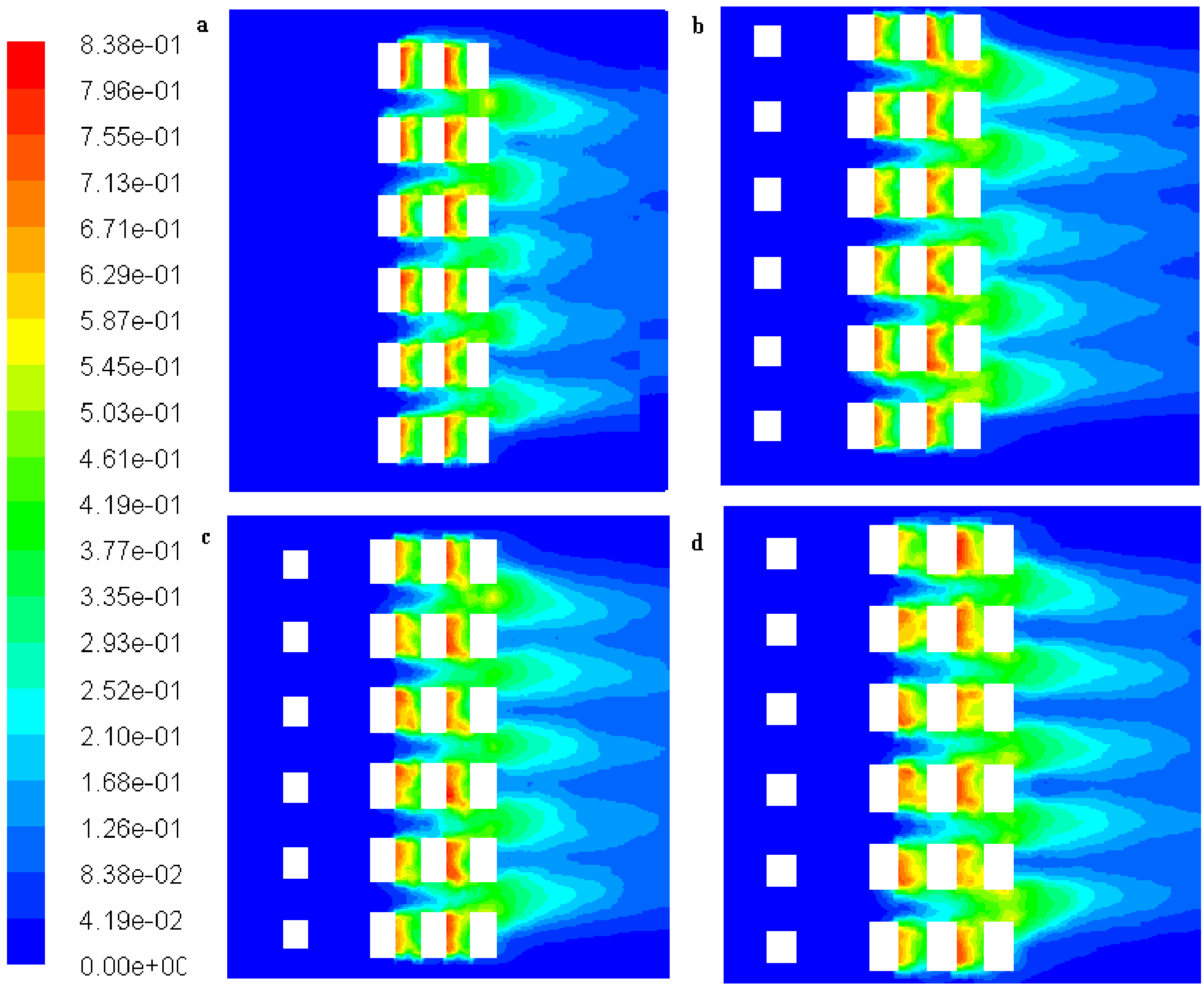


Figure8: NO2 concentration at pedestrian level (2 m above ground) in case: reference (a), H (b), 2H(c), 4H (d).

Figure 9, illustrates the velocity vectors at the pedestrian level when comparing the reference case with (a) case H; (b) case 2H; (c) case 4H. As shown in the figures, the velocity at the leeward side of the buildings is more than the windward. Velocity reduces when the frontal buildings are built to the same height as the street canyon. However, when the height of the frontal buildings is increased, the velocity reduces this is because, not only, is the lower level wind flow blocked, but the upper layer wind flow is not able to penetrate into the building cluster directly. The situation becomes more serious when the height of frontal buildings changes to 4 times that of the building cluster. A large portion of the prevailing wind is blocked, causing poor ventilation inside the street canyon and an unacceptable level of pedestrian discomfort.

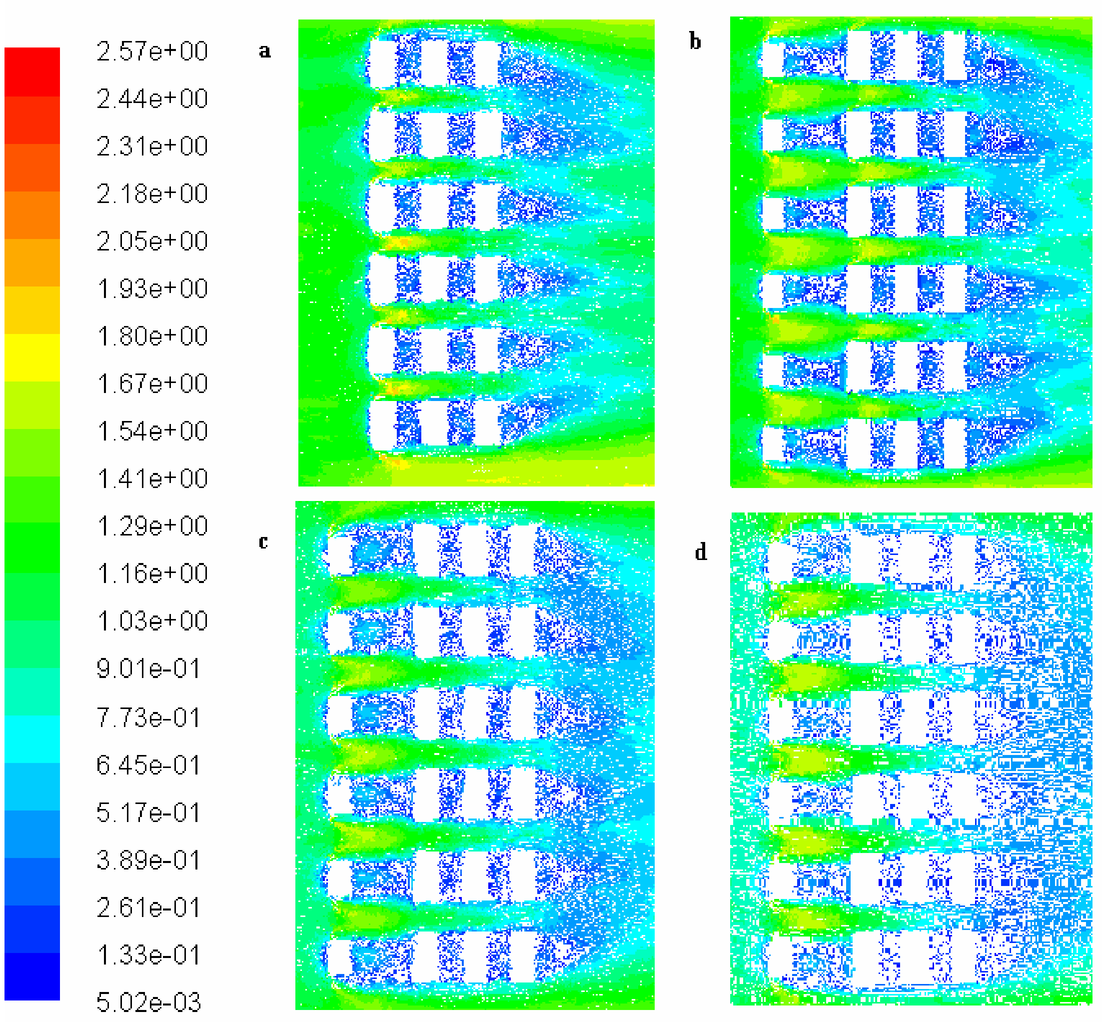


Figure 9: velocity vectors at the pedestrian level at pedestrian level (2 m above ground) in case: reference (a), H (b), 2H(c), 4H (d).

When designers are forced to increase the height of buildings, there is also a channeling effect. In this study we change the width of streets in worse case 4H; we consider 3 cases 26m, 32.5m, and 39m. Figure10 illustrates the NO2 mol fraction at the pedestrian level these cases. In the downstream area it is lower in the width of streets 39m and higher in 26m than other cases.

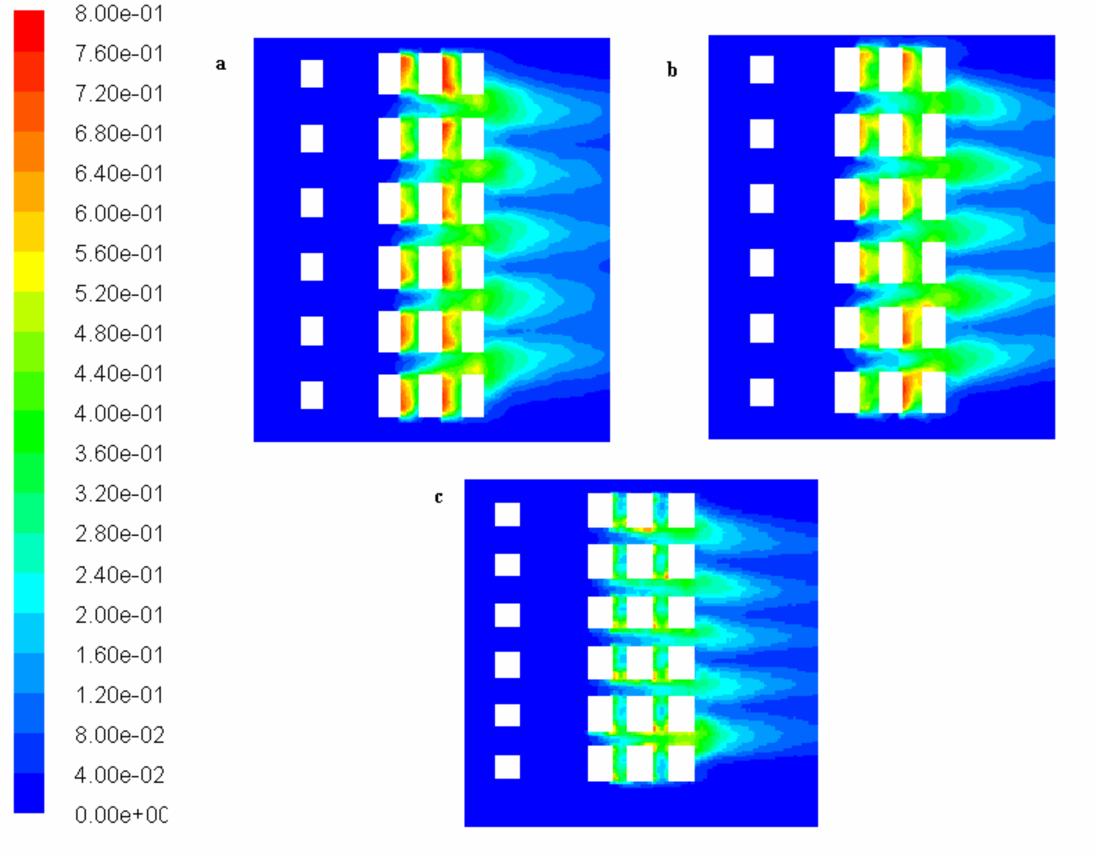


Figure 10: NO2 mol fraction at the pedestrian level (2 m above ground) in case: 26m (a), 32.5(b), 39m(c).

4 Summary

In this study, the influence of different heights of frontal blocking Buildings and width of streets are considered from the perspective of effective air Ventilation. The velocity at 2 m above ground (pedestrian level), NO2 mol fraction are investigated to quantify the air ventilation impact caused by the alignment urban. It is illustrated that the velocity reduces by increasing of the high-rise buildings that are located in the upstream direction of a street canyon and width of the streets. This can also have serious impacts on pollutant dispersion. It is concluded that the alignment of high-rise buildings located to the upstream of the city, blocks the fresh air into the city. this effect is not only on the air ventilation but also on the pollutant dispersion rate. Therefore, the designers and urban planners should pay careful attention to this subject.

REFERENCES

[1] Lam, J.S.L, Lau, A.K.H, Fung, J.C.H, Application of refined land-use categories for high resolution meso-scale atmospheric modeling, Boundary Layer Meteorology, 119, 263–288, 2006.

[2] Sotiris.V, Bernard E.A. Fisher, Koulis. P, Norbert Gonzalez-Flesca, Modeling air quality in street canyons: a review, Atmospheric Environment 37, 155–182, (2003).

[3] Chan. T.L., Dong. G., Leung. C.W., Cheung. C.S., Hung.W.T., Validation of a two-dimensional pollutant dispersion model in an isolated street canyon, Journal of Atmospheric Environment 36, 861–872, (2002).

[4] Kim. J.J., Baik. J.J., A numerical study of the effects of ambient wind direction on flow and dispersion in urban street canyons using the RNG k _ 3 turbulence model, Journal of Atmospheric Environment 38, 3039–3048,(2004).

[5] Dipankar.C, Gautam.B, Sakir.A, Numerical simulation of flow past row of square cylinders for various separation ratios, Computers & Fluids 39 , 49–59,(2010).

[6] Tetsuro Tamura, Towards practical use of LES in wind engineering, Journal of Wind Engineering and Industrial Aerodynamics 96 ,1451–1471,(2008).

[7] X. Wanga, K.F. McNamarab, Effects of street orientation on dispersion at or near urban street intersections, Journal of Wind Engineering and Industrial Aerodynamics 95, 1526–1540, (2007).

[8]Jie He, Zhiguo Qi, Chihang .Z, Xiangtai .B, Simulations of pollutant dispersion at toll plazas using three-dimensional CFD models, Transportation Research Part D 14, 557–566, (2009).

[9]Parham A. Mirzaei, Fariborz Haghighat, A novel approach to enhance outdoor air quality: Pedestrian ventilation system, Building and Environment xxx, 1–12, (2010).

[10] Ishihara, T., Hibi, K., Turbulent measurements of the flow field around a high-rise building. Journal of Wind Engineering, Japan (76), 55–64, 1998.

[11] Hong Huang, Yoshiaki Akutsu, Mitsuru Arai, Masamitsu Tamura, A two-dimensional air quality model in an urban street canyon: evaluation and sensitivity analysis, Atmospheric Environment 34, 689-698, (2000),

[12] Yoshihide.T, Akashi, AIJ guidelines for practical applications of CFD to Pedestrian wind environment around buildings, Journal of Wind Engineering and Industrial Aerodynamics 96, 1749–1761, (2008).

[13] Franke. J, Recommendations of the COST action C14 on the use of CFD in predicting pedestrian wind environment. In: The Fourth International Symposium on Computational Wind Engineering, Yokohama, Japan. 2006.

[14] Huang, H., Ooka, R., Chen, H., Kato, S., Hong, C., Takeo, T., Takeaki, W., CFD analysis on traffic induced air pollutant dispersion under non-isothermal condition in a complex urban area in winter. Journal of Wind Engineering and Industrial Aerodynamics 96, 1774–1788, 2008.