Numerical prediction of air flow within street canyons based on different two-equation k- models

Yazid, Afiq Witri Muhammad; Sidik, Nor Azwadi Che; Salim, Mohamed; Yusoff, Nur Hamizah Mohamad

Published in:
IOP Conference Series: Materials Science and Engineering

DOI:
10.1088/1757-899X/50/1/012012

Publication date:
2013

Document Version
Publisher's PDF, also known as Version of record

Numerical prediction of air flow within street canyons based on different two-equation *k*-models

2013 IOP Conf. Ser.: Mater. Sci. Eng. 50 012012

(http://iopscience.iop.org/1757-899X/50/1/012012)

Numerical prediction of air flow within street canyons based on different two-equation *k*-ε models

Afiq Witri Muhammad Yazid[1], Nor Azwadi Che Sidik[1,a], Salim Mohamed Salim[2] and Nur Hamizah Mohamad Yusoff[1]

[1]Faculty of Mechanical Engineering,
Universiti Teknologi Malaysia, 81310, Skudai, Johor Bahru, Malaysia
[2]School of Engineering, Taylor's University, 47500, Subang Jaya, Selangor, Malaysia

E-mail: [a]azwadi@fkm.utm.my

Abstract. Numerical simulations on airflow within street canyons were performed to investigate the effect of the street aspect ratio and wind speed on velocity profiles inside a street canyon. Three-dimensional Standard, Renormalization Group (RNG) and Realizable k-ε turbulence model are employed using the commercial CFD code FLUENT to solve the Reynolds-averaged Navier-Stokes (RANS) equations. A comparison of the results from the presently adopted models with those previously published demonstrated that the k-ε model is most reliable when simulating wind flow. The model is then employed to predict the flow structures in a street canyon for a range of aspect ratios (building height to street width ratio) between 0.5 - 2 at Reynolds number of 9000, 19200 and 30700 corresponding to the ambient wind speeds of 0.68m/s, 1.46m/s and 2.32m/s respectively. It is observed that the flow structure in the street canyon is influenced by the buildings aspect ratios and prevailing wind speeds. As the street aspect ratio increases, the air ventilation within the canyon reduces.

1. Introduction

Street canyons are one of the most prominent features found in city environments where the flow is dominated by the interaction of prevailing wind and building layout. Rapid urbanization demands the constructions of high rise buildings resulting in low air ventilation hence affecting inhabitants comfort. Besides buildings configuration/layout, the local meteorological conditions and trees plantings are amongst the many factors affecting wind flow within street canyons. The formation of a centrally located vortex is one of the characteristic features of street canyon flow when prevailing winds are perpendicular to the street canyon. While oblique or parallel approach flow provides the best wind flow condition for the purpose of ventilation, perpendicular approach flow is the worst configuration due to existence of vortex that has a very low wind speed at pedestrian level[1]. Parameterization of buildings layout to represent canyon geometry has been widely used in terms of the ratio of the building height (H) to the width (W) between buildings. The flow regimes of isolated roughness flow, wake interference flow and skimming flow at certain range of buildings aspect ratio has been well described [2-4].
The introduction of two-equation turbulence closure schemes more than two decades ago has benefited most Computational Fluid Dynamics (CFD) researchers as it is computationally efficient [5-8]. The wind engineering community has long adopted the two-equation turbulence models,

specifically the Standard *k*-ε turbulence model, to represent successfully the wind flow field [5,9-11], although it is not so accurate for flows involving impingement and separation regions [12] which may lead to the poor prediction of flow structures at the wake of buildings. Therefore, another alternative turbulence model should be adopted to ensure reliable results. Up till now, the authors are only aware of one numerical study available on the assessment of flow in street canyons using two-equation turbulence models by Chan [13]. They found out that the RNG *k*-ε turbulence model provide the best results on the pollutant concentration distribution in a street canyon when compared to Standard *k*-ε and Realizable *k*-ε model. However, there is missing information regarding the wind flow field within the street canyon. Furthermore, there are only a few studies on the flow induced in a street canyon for different ambient wind speeds by Baik and Kim [14] at H/W = 2.0 and by Nazridoust and Ahmadi [15] at H/W = 0.5 and 1.0.

Therefore, the objective of this study is to analyze the performance of *k*-ε turbulence models and its variants to simulate three-dimensional (3-D) flow inside a street canyon. This is important as there is no study that has been carried out to assess the performance of various *k*-ε turbulence models (Standard, RNG and Realizable *k*-ε) on the flow structures inside a 3-D street canyon. The comparison of a wide range of ambient wind speeds against experimental data also helps to increase confidence in our results. The most accurate *k*-ε turbulent model is then adopted to simulate different buildings aspect ratio (H/W) coupled with a wide range of ambient wind speeds by analyzing the flow structure and air ventilation inside the street canyon. This enhances our understanding on the capacity of buildings aspect ratio parameter towards air ventilation in urban street canyon.

2. Numerical Model

In the current CFD model, 1:200 scaled model of a street canyon aligned perpendicular to wind direction is constructed in a reduced 3-D spatial domain formed by two buildings of equal height H and separated by a street width W based on the experimental setup by Allegrini [16]. The distance of the street canyon from the inlet and top boundary is based on the provided boundary conditions of experiment. The outlet distance is at a distance H from the cavity and outflow condition is imposed. Only the central 1/3rd of the total lateral distance from the actual experiment was modelled while imposing symmetry boundary condition at both sides as illustrated in figure 1.

Incompressible, 3-D steady turbulence flows are solved using two-equation turbulence models of RANS equations. The models considered for the purpose of validation are Standard *k*-ε, RNG *k*-ε and Realizable k-ε combined with enhanced wall treatment for the near wall treatment [17]. Similar inlet profiles of velocity and turbulent kinetic energy from the experiment were imposed at the inlet of the numerical domain ranging from Re = 9000, 19200 and 30700 which corresponds to 0.68, 1.46 and 2.32 m/s of freestream velocity. The dissipation rate can be approximated by ε = (Cμ3/4k3/2)/l. The Reynolds number is based on the free stream velocity, U at buildings height, H. The governing equations are discretized using second order upwind for all the advection term and second order for pressure interpolation while the SIMPLE algorithm is used to solve the equations. The convergence criteria are set at 1.0e-6 to ensure the convergence of the results. The numerical simulation is performed using the commercial CFD software Fluent 6.3.

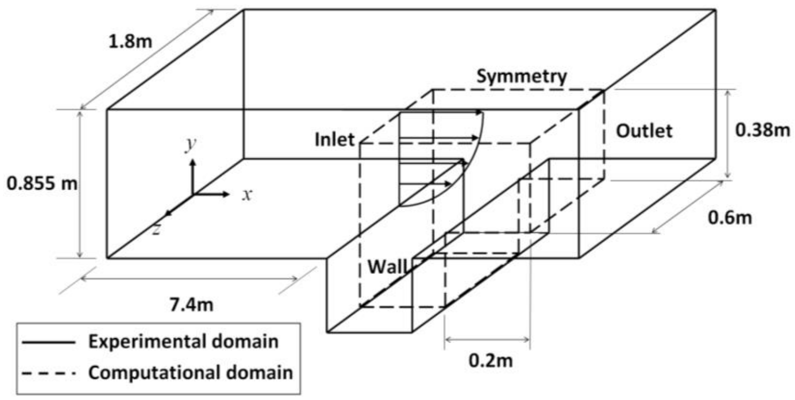


Figure 1. Experimental and computational domain (not to scale).

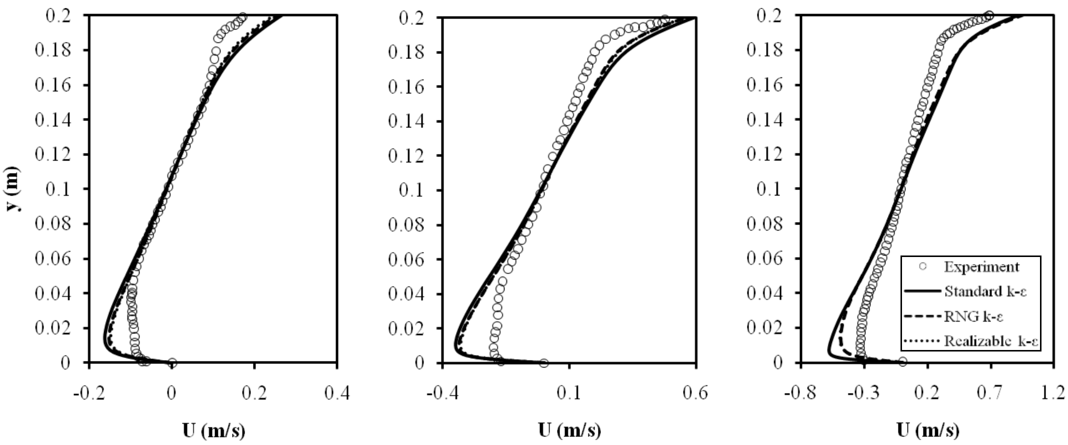
3. Model Validation

The accuracy of the CFD model is evaluated using extensive wind-tunnel experimental results of Allegrini [4]. The data consists of isothermal and thermal case for buildings aspect ratio H/W = 1 at a wide range of Reynolds number. For the purpose of validation, only the benchmark of isothermal case is compared. Mesh analysis is done through grid independence study with different mesh configuration of a structured hexahedral mesh. Since the enhanced wall treatment is used, the dimensionless parameter of wall y^+ is kept below 5 to ensure the variable gradient near the wall is adequately resolved [18]. In addition to that, cell stretching away from the buildings wall and ground floor with 1.2 is used to reduce the computational time while maintaining its accuracy.

After an appropriate mesh is identified for each turbulence models at each Reynolds number, the performances of the turbulence models are assessed using mesh independence. The mesh independence study of each flow case and turbulence models were analyzed based on the vertical profile of wind x-velocity generated inside the street canyon as illustrated in figure 2 while the percentage of average error between the numerical and experimental results are then calculated based on the vertical x-velocity profile inside street canyon with vertical interval of 0.01m as tabulated in table 1.

Table 1. Percentage of average error.

Turbulence models	Percentage of average error (%)		
	Re = 9000	Re = 19200	Re = 30700
Standard *k*-*ε*	28.18	51.67	59.58
RNG *k*-*ε*	22.02	41.43	53.85
Realizable *k*-*ε*	19.94	41.62	49.99



(a) (b) (c)

Figure 2. Vertical x-velocity profile: (a) Re = 9000, (b) Re = 19200, (c) Re = 30700.

It can be seen that the results of all two-equation turbulence models have over predicted the vertical profile particularly at 0 m < y < 0.02 m and y > 0.18 m. Nevertheless, the Realizable k-ε yields the least error for Re = 9000 and 30700 whilst for Re = 19200, the error is slightly higher than RNG k-ε where the deviation is less than 0.2%. With the good flow profile achieved by the Realizable k-ε, the model is then chosen for the investigation of the remaining objectives of this paper.

4. Results And Discussion

This section presents the numerical solutions of turbulence flow at different buildings aspect ratio and ambient wind speeds. The height of the building is fixed at H while only the width of the canyon is varied to get different buildings aspect ratio. This allows the Reynolds numbers to be maintained throughout the investigations. Figure 3 shows the flow patterns in the vertical cross-section for different Reynolds numbers. In the case of low wind speed (Re = 9000) with H/W = 0.5, the ambient flow at roof level penetrates into the canyon between buildings creating 'wake interference flow' described by Oke [2]. In the case of higher H/W (1.0, 2.0), the flow above roof is separated from the canyon flow and thus is categorized as 'skimming flow'. The secondary vortex located at the bottom of leeward wall with H/W = 0.5 and at the bottom of street canyon with H/W = 2.0 is disappeared or reduced in size as the wind speed aloft increased to Re = 30700. Looking at H/W = 2.0, the strong ambient speed aloft (Re = 30700) enhanced the momentum transfer deep into the lower region of the street canyon and in turn elongated the primary vortex further down and at the same time reduced the size of secondary vortex.

Figure 4 shows the x-velocity profile along the middle height of the street canyon at different buildings aspect ratio and Reynolds number. Comparison between different H/W on the vertical velocity shows that the air ventilation inside street canyon is reduced by the increment of H/W. This is most probably due to low air penetration from the above causing reduction of momentum transfer at deep street canyon.

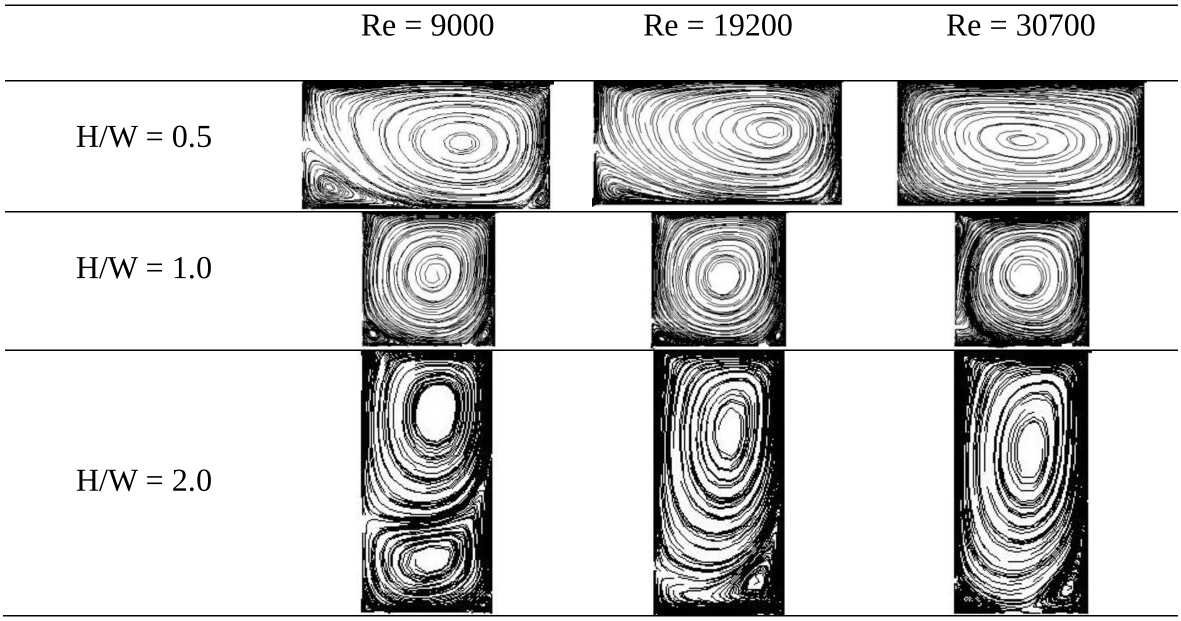
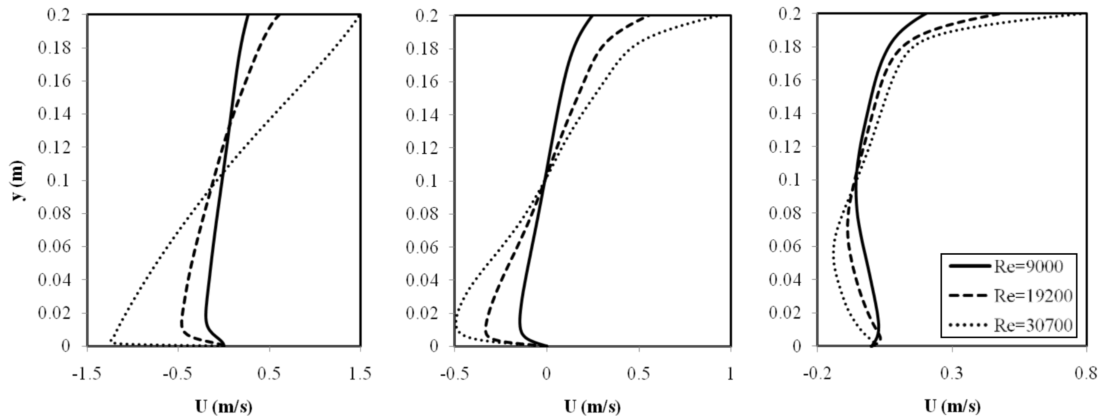


Figure 3. Streamlines inside street canyon (model scale) with different Reynolds numbers and buildings aspect ratios.



(a) (b) (c)

Figure 4. Vertical x-velocity profile: (a) H/W = 0.5, (b) H/W = 1.0, (c) H/W = 2.0.

5. Conclusions

The numerical results of the vertical velocity profile and streamline on air flow are investigated. It is observed that:

- The Realizable k-ε is the most accurate amongst other *k*-ε turbulence models to predict the wind flow profile inside street canyon.
- A secondary vortex is observed only at low Reynolds number.
- The air ventilation inside street canyon reduces with the increase of H/W.

6. Acknowledgements
The authors would like to thanks the Universiti Teknologi Malaysia and Ministry of Higher Education, Malaysia for sponsoring this research. This research is supported by research grant Vot. 4F114 and 4L074.

References
[1] Soulhac L, Perkins R J and Salizzoni P 2008 *Bound-Lay Meteorol.* **126** 365.
[2] Oke T R 1998 *Energ. Buildings* **11** 103.
[3] Kovar P A, Louka P, Sini J F, Savory E, Czech M, Abdelqari A, Mestayer P G and Toy N 2002 *Water Air Soil Poll.* **2** 237.
[4] Afiq W M Y, Azwadi C S N and Saqr K M 2012 *International Journal of Mechanical and Materials Engineering* **7** 158.
[5] Mochida A, Tominaga Y, Murakami S, Yoshi R, Ishihara T and Ooka R 2002 *Wind Struct.* **5** 227.
[6] Tahseen T A, Ishak M and Rahman M M 2013 *J Mech Eng Sci* **4** 418-430.
[7] Tahseen T A, Ishak M and Rahman M M 2012 *Inter J Automot Mech Eng* **6** 753-765.
[8] Najiha M S, Rahman M M, Yusoff A R and Kadirgama K 2012 *Inter J Automot Mech Eng* **6** 766-774.
[9] Vardoulakis S, Dimitrova R, Richards K, Hamlyn D, Camilleri G, Weeks M, Sini J F, Britter R, Borrego C, Schatzmann M and Moussiopoulos N 2011 *Environ Model Assess.* **16** 169.
[10] Idris M S, Irwan M A M and Ammar N MM 2012 *J Mech Eng Sci* **2** 206-216.
[11] Mohd Adib M A H, Mohd Hasni N H, Osman K, Maskon O and Kadirgama K 2012. *J Mech Eng Sci* **2** 217-225.
[12] Meroney R N, Leitl B M, Rafailidis S and Schatzmann M 1999 *J. Wind Eng. Ind. Aerod.* **81** 333.
[13] Chan T L, Dong G, Leung C W, Cheung C S and Hung W T 2002 *Atmos. Environ.* **36** 861.
[14] Baik J J and Kim J 1999 *Electron. J. Appl. Meteorol.* **38** 1576.
[15] Nazridoust K and Ahmadi G 2006 *J. Wind Eng. Ind. Aerod.* **94** 491.
[16] Allegrini J, Dorer V and van Carmeliet J 2013 *Build Environ.* **59** 315.
[17] FLUENT Documentation: User guide, ANSYS Inc, 2005
[18] Ariff M, Salim S M and Cheah S C 2009 7th International Conference on CFD in the Minerals and Process Industries CSIRO, Melbourne, Australia. December