ELSEVIER
CrossMark
Available online at www.sciencedirect.com
ScienceDirect
Procedia Engineering 121 (2015) 2209 – 2216
Procedia Engineering
www.elsevier.com/locate/procedia

9th International Symposium on Heating, Ventilation and Air Conditioning (ISHVAC) and the 3rd International Conference on Building Energy and Environment (COBEE)

Evaluation of CFD Simulation Using Various Turbulence Models for Wind Pressure on Buildings Based on Wind Tunnel Experiments

Biao Li[a], Jing Liu[a,b,*], Feifei Luo[a], Xiaoxin Man[c]

[a]*School of Environmental and Municipal Engineering, Harbin Institute of Technology, 150090, Harbin, China*
[b]*State Key Laboratory of Urban Water Resource and Environment, Harbin Institute of Tech-nology, 150090, Harbin, China*
[c] *China Construction Engineering Design Group Corporation Limited (Headquarter), 100037, Beijing, China*

Abstract

Various Reynolds Average Numerical Simulation (RANS) turbulent models of Computational Fluid Dynamics (CFD) are widely used in numerical simulation of wind environment with buildings areas, but none of them is the most accurate or applicable one. In order to evaluate different numerical simulation models, wind tunnel experiments were designed to validate the results of numerical simulations. Rigidity models surface pressure measurements were employed to get pressure values on several building surfaces among buildings, which accurately assessed different simulating results of wind field effects on buildings. By comparing and validating turbulent models with measurements of wind tunnel experiments, useful guidelines and references were given to choose appropriate turbulent model for numerical wind environment researches and practical application. This study presents the RNG k- model including the DVM treatment has a relative good predicting for both the plan view of velocity contour and the surface pressure. In addition, the Spalart-Allmaras models can be more appropriate predicting results for surface pressure measurements among all models, but its reliability of buildings wind environment prediction is not cleared yet. The future work will be needed.

Peer-review under responsibility of the organizing committee of ISHVAC-COBEE 2015

Keywords: CFD, wind tunnel experiment, wind pressure, buildings

* Corresponding author. Tel.: +86-13104517951; fax: +86-0451-86282123.
E-mail address: liujinghit0@163.com

Peer-review under responsibility of the organizing committee of ISHVAC-COBEE 2015
doi:10.1016/j.proeng.2015.09.094

1. Introduction

Computational fluid dynamics (CFD) is having an increasing impact on the assessments of buildings wind environment and aerodynamic effects [1-4], and a large number of codes employing a variety of solution strategies are now available. Thus, evaluation of the performance among various turbulence models predicting the wind environment around buildings can be more concern, which benefits the selection of turbulence models and improves accuracy of buildings wind environments simulations [5-9]. However, how to describe or estimate the wind environments of buildings may be a serious question, because the effects of wind incidence, building shape and buildings layout are still unclear. Therefore, based on the wind tunnel tests with inhomogeneous buildings, in which surface pressure measurements were deployed, the present study investigated the performance of several RANS models, mainly including one-equation and two-equation turbulence models that historically the most widely used turbulence models in industrial CFD. In general, total 11 turbulence models and combinations of different correction methods were comprehensively conducted both in the wind environment around buildings and aerodynamic effect of buildings. The work reported here focused on an inhomogeneous buildings configuration rather than a regular or stagger layout.

2. Methods

2.1. Test model

The test model considered in this paper is an inhomogeneous asymmetrical staggered layout model (Fig. 1a), which comprised of 23 rectangular blocks of equal height as 105 mm, the totally blocking ratio was 2.69%, and reported in detail in Li et al. [10]. As shown in Fig. 1a, solid lines in the plane view referred to roads network. Moreover, there were three modules, placed in upstream, midstream, and downstream areas (black blocks in Fig. 1a), for the measurement of wind pressure on buildings surfaces. Hereafter, those were referred to as the upstream pressure module (UPM), midstream pressure module (MPM), and downstream pressure module (DPM), respectively. In addition, as shown in Fig. 1b, each pressure module was totally distributed with 40 pressure-measuring taps both in the windward face and leeward face. Furthermore, to ensure that no vibration occurred when testing, the modules for wind pressure tests were made of 10 mm-thickness Plexiglas. Before the wind tunnel tests, all taps had been validated by strict airtight tests. All the other blocks (white blocks in Fig. 1a) were made of 2 mm-thickness ABS (Acrylonitrile butadiene styrene) to ensure that no destruction occurred at the test velocity.

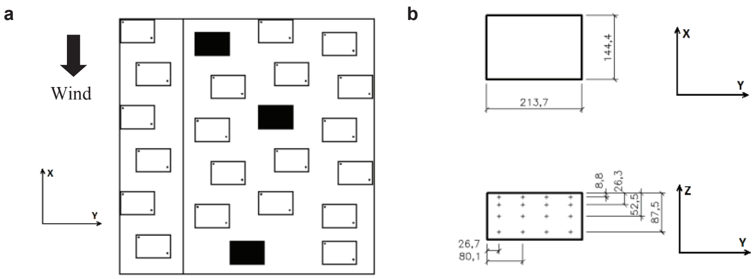

Fig. 1. (a) Schematic diagram of the buildings layout, (b) Details of pressure module.

2.2. Wind tunnel test

The wind tunnel experiment, which was conducted in the Laboratory of Wind Tunnel and Water Flume at Harbin Institute of Technology (WTWF-HIT), was carried out to be a calibration of the CFD simulation with the wind environment and aerodynamic effects of inhomogeneous buildings. The facility is a closed backflow atmospheric boundary wind tunnel with a rectangular cross-section, which contains a test section of 3.0 m in height, 4.0 m in width and 25 m in length. Under steady conditions, the wind tunnel has a high reliable flow field, where the non-uniformity of the wind velocity distribution is less than 1%, the turbulence intensity is less than 0.46%, and the average flow deviation angle is less than 0.5° over the entire test section in the empty wind tunnel. In this wind tunnel test, a fully turbulent flow, atmospheric boundary layer, throughout the test section was maintained for the reference stream velocity approximately 11 m/s, which was measured by a pitot-static tube at a height of 2.7 m. A special test platform (Fig. 2) was used to measure the surface pressure of buildings and the total drag in WTWF-HIT. Surface wind pressures were measured by the Scanivalve DSM3400 pressure scanning valve system with the sampling frequency of 312.5 Hz and last for 36 s in each sample. Each test had three samples, and the averaged values were used in the following analysis.

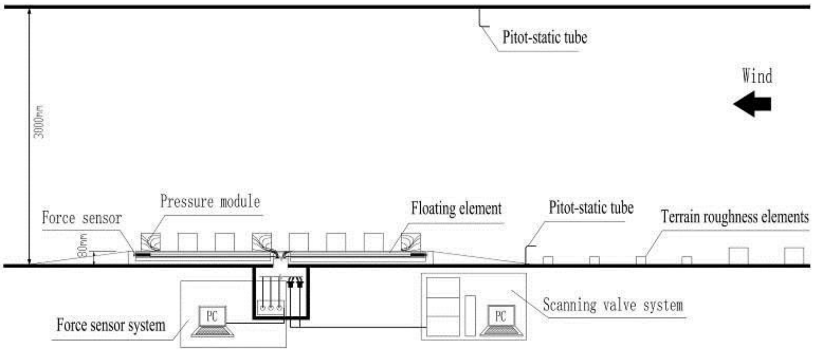

Fig. 2. Schematic diagram of the wind tunnel test.

2.3. Computational domain and grid

A numerical wind tunnel simulation model of buildings was set up exactly the same as in the wind tunnel test, as shown in Fig. 3, composing of inhomogeneous buildings, the test platform surrounding a ramp-transition, and the test body section of the wind tunnel. The dimensions of the computational domain were was deployed under restrictions of computer calculation capacity, steady turbulence inflow, a long enough test section to ensure fully developed outflow [6], and the real wind tunnel test details. The resulting dimensions of the domain were X × Y × Z = 7000 × 4000 × 3000 mm, with the upstream domain length is 19H = 1995 mm, where H referred to as the height of the buildings model. The computational grid was created using the Hexa-cells, which can get a better calculation domain for the structured gridding, resulting a fine hexahedral grid with almost 2 million cells in total that executed by the gird sensitive analysis. The height of the first layer of wall adjacent cells, with all the four walls of wind tunnel, the test platform, were carefully adjusted to fix y* requirements for different codes respectively [11]. Moreover, the y* should meet requirements of different wall functions.

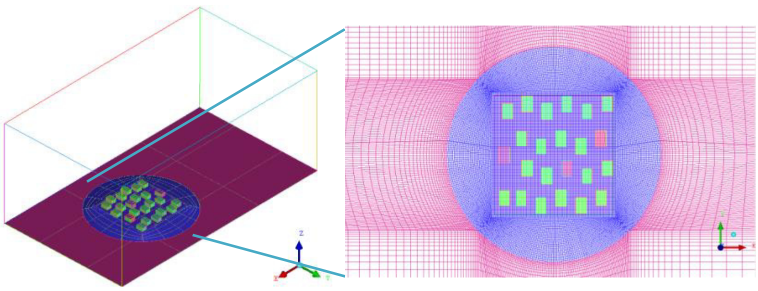

Fig. 3. Computational domain and grid details near buildings.

2.4. Boundary conditions

With all turbulence models were used in this study, the same geometry model and boundary conditions were arranged. In numerical simulations, the inlet boundary conditions were based on the measured incident profiles of mean wind speed U and turbulence intensity I_u (Fig. 4) in the wind tunnel test. The turbulent kinetic energy k and the turbulence dissipation rate ε were calculated from U and I_u following Eq. (1) and Eq. (2), respectively.

$$k(z) \cong \alpha\left(I_u(z)U(z)\right)^2 \tag{1}$$

$$\varepsilon(z) \cong \frac{C_\mu^{\frac{3}{4}} k_z^{\frac{3}{2}}}{KL_u} \tag{2}$$

where α is a parameter that equals 1.5, constant C_μ is 0.09, κ is the von Karman constant (= 0.4) here, L_u is the length scale of the turbulence intensity that defines by Eq. (3), and λ_l is the scaled ratio (in this study is 200).

$$L_u \cong \frac{100}{\lambda_l^{\frac{1}{2}}} \cdot \left(\frac{z}{30}\right)^{\frac{1}{2}} \tag{3}$$

In addition, all walls were non-slip wall boundary conditions, and the values of the sand-grain roughness height k_s (m) and the roughness constant C_s of the floor surface were determined from the aerodynamic parameters that deduced by the incident profile in wind tunnel; all the other walls were zero roughness for lack of physics information. Outflow conditions were applied at the outlet for the considering of fully developed free outflow. Particularly, the profiles measured in the empty wind tunnel at the location where the buildings will be positioned were defined as the inlet profiles in simulations, shown as Fig. 4.

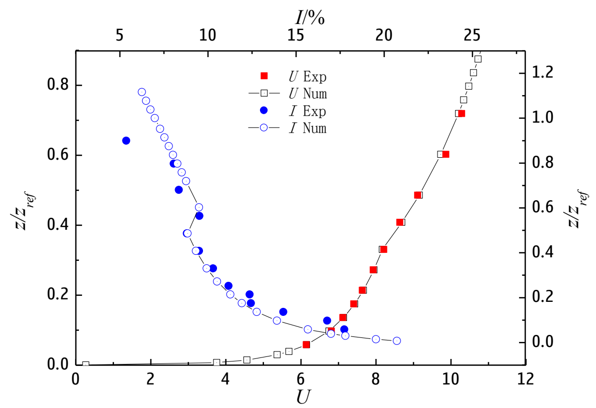

Fig. 4. Vertical profiles of numerical simulation at inlet boundary deduced from experiment incident profiles.

2.5. Turbulence models and solver settings

In this study, all numerical simulations were based on the commercial CFD code Fluent, and turbulence solves methods evaluated here that were all 3D steady RANS models. Considering the common utilization among RANS models in the buildings wind environment simulation, this study mainly conducted Spalart-Allmaras (S-A), k-ε, and k-ω models, which employed the Boussinesq hypothesis to get a relatively low cost computation for the turbulence viscosity. In addition, to ensure high performance results with the surface pressure, this study mainly conducted the enhanced wall function except those ones of special setup. Then, pressure-velocity coupling calculated by the SIMPLE algorithm; pressure interpolation was second order; both the convection terms and the viscous terms in the governing equations used second-order discretization schemes. Furthermore, we assumed convergences were obtained when all the scaled residuals levelled off and reached a minimum of 10^{-6} for x, y, z momentum, 10^{-4} or 10^{-3} for k, ε, ω and continuity, and 10^{-3} for drag coefficient.

3. Results and analyses

All simulations, with the same boundary conditions and solver settings, are conducted for this evaluation study, yielding results by one equation and two equations models. The current near wall treatment is mainly the enhanced wall functions or the low-Re corrections in this study, except those specific simulations which were made for other purposes.

3.1. Wind environments

Fig. 5 displays contours of the magnitude velocity in a horizontal plane at 2/3 of buildings height above ground for all evaluation cases. The following observations are made:

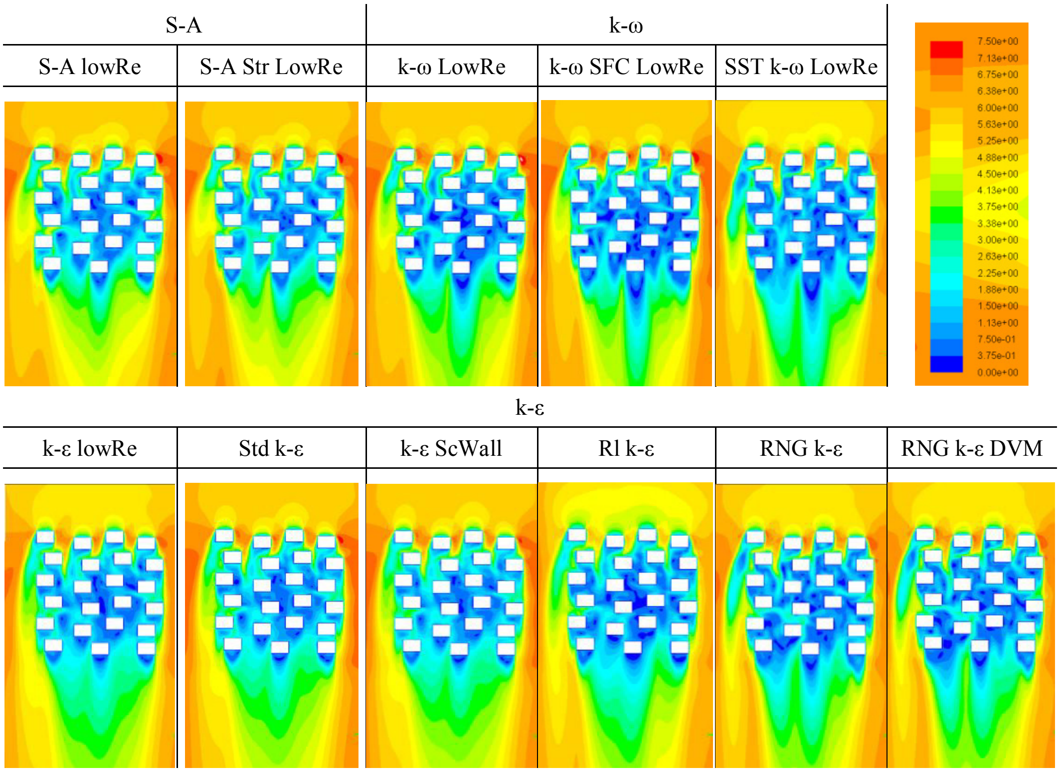

Fig. 5. Contours of the magnitude velocity in a horizontal plane at 2/3 of buildings height.

1. Fig. 5(S-A) illustrates that the results of S-A low-Re model setting shows very similar wind environments to the results of S-A Strain/Vorticity based model setting. While the Strain based methods gives less

pronounced wake flows both in downstream of each building and the whole buildings area. Because the Strain/Vorticity based methods [12] including both the rotation and strain rate reduces the production of eddy viscosity and consequently reduces the eddy viscosity itself in regions where the measure of vorticity exceeds that of strain rate.

2. From Fig. 5 (k-ω), it can be observed that the horizontal distributions of velocity computed by these three kinds of turbulence models for the same condition of buildings are similar. However, the regions of wakes for the k-ω one are obviously shorter than others, and the high speed regions in upstream area for the SST k-ω one can be the smallest. The reason for these is that, for standard k-ω model, the computational results relatively strong depend on the freestream values outside the shear layer. Furthermore, once the shear flow correction (SFC) method was included in the standard k-ω model, the velocities decrease outside of buildings effect area for the improvement of predicting accuracy with free shear flows, and the flow pattern is comparable with that of the SST k-ω model.
3. In Fig. 5 (k-ε), generally, there are mainly three types of flow patterns in contours of simulations for various models, which indicates the group of k-ε models with different near wall treatments, the Realizable k-ε model (Rl k-ε), and the group of RNG k-ε models. The group of RNG k-ε model assesses the most pronounced and the longest wake flow area, and the Realizable k-ε model gives the greatest lower velocity value area in wake flows. Because the RNG model has additional term in its ε equation, includes effect of swirl on turbulence, and provides an analytical formula for turbulent Prandtl numbers, which can improve the accuracy and reliability in computing the wind environments.

3.2. Drag coefficient

To clarify the difference simulation results of various turbulence models, drag coefficient for each of the three pressure modules, based on wind tunnel pressure measurements in upstream, midstream, and downstream areas, were compared in this study. Following results and analyses are acquired:

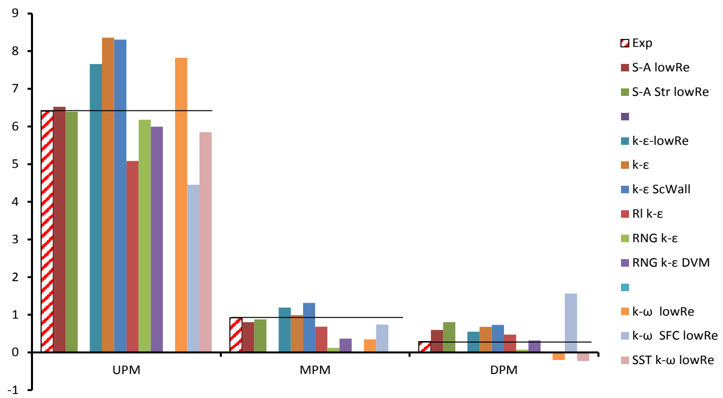

Fig. 6. Drag coefficients for each pressure modules based on wind tunnel pressure measurements.

1. Fig. 6 shows good comparison of S-A simulation with the wind tunnel measurement, and the Strain/Vorticity based production method are better than the Vorticity based one at the upstream and midstream pressure blocks. The Stain/Vorticity based method can give more reductions of eddy viscosity, which should increase the flow velocity in wakes. Because the production of eddy viscosity in S-A model used the Vorticity based method tends to be over predicted.
2. The group of the standard k-ε models overrates the drag coefficients of all three pressure modules, while all others almost underestimate that. The Realizable k-ε model has a lowest drag coefficient among all k-ε based models at the UPM. Moreover, the RNG k-ε models using Differential Viscosity Modifications (DVM) method shows the most comparable values in the aggregate of all drag

coefficients of the three pressure modules. Because the DVM method improve the effective viscosity calculation accuracy at the near-wall regions by a differential formula when the grid density is well resolved.

3. The disadvantage of the k-ω model sensitivity of the free stream results the less accuracy of drag coefficients, used the standard k-ω model with low-Re modifications, at all three pressure measurements buildings than those of the standard k-ε case. The UPM drag coefficient is over predicted, while MPM and DPM drag coefficients get relatively strong insufficient. Furthermore in present study, Fig. 6 illustrates that the SST k-ω model can be better than the k-ω model at upstream with assessments of drag coefficient, whereas it gets worse at the MPM and DPM. In addition, the k-ω model with the Shear Flow Correction (SFC) method enabled conducts very pronounced adverse improvements at the drag coefficient of UPM and DPM, for the over correction of dissipation of k and ω. So that the contour of SFC method in Fig. 5 has more intense wake eddies and less high speed flows.

In general, with the comparison of all results for various turbulence models and near wall treatments by the drag coefficient based on surface pressure measurements, the S-A models can be more appropriate for surface pressure measurements, while all the others show relative large deviation. The RNG k-ε model using the DVM method also has a relative good predicting result. While the group of the standard k-ε models and the low-Re k-ω model overestimate the aggregate performance of drag coefficients, all the improvement or modification methods and models for standard k-ε and k-ω models underestimate that.

4. Conclusions

This paper concerns the investigation on effective turbulence models for assessment of buildings wind environments based on the wind tunnel test with buildings surface pressure measurements. Totally 3 general types of one-equation and two-equation turbulence models, including 11 models and special treatments, were conducted in the present study. Results were presented and discussed based on comparisons between the wind tunnel measurements and computational flow predictions, the following conclusions are drawn. First, the plan view contour varies mainly for different types of turbulence models, in other words, contours of the same group of turbulence models are similar. Second, since overall trends of drag coefficients with the three pressure modules are compared, the S-A models show more appropriate predicting results for surface pressure measurements among all models presented here. But its reliability is still unclear whether the S-A models can undervalue or overvalue the turbulence length scale, which need more investigations in further studies. Third, the RNG k- model including the DVM treatment has a relative good predicting both for the plan view contour and the surface pressure, which is recommended for buildings wind environment simulations. Further evaluations will be needed based on both flow fields and total drag force predictions.

Acknowledgements

This work was supported by the State Key Laboratory of Urban Water Resource and Environment under Grant no. 2010TS04; and the China State Construction Mechanical and Electrical Research on Megatall Buildings under Grant no. CSCEC-2010-Z-01-03.

References

[1] K. Robel, and C.X. Lin, CFD Modeling of Pressure Coefficients for a Natural Ventilation Study in an Experimental Low Rise Building. ASME 2013 International Mechanical Engineering Congress and Exposition, 2013:V06BT07A008.

[2] M. Lin, J. Hang, Y. Li, Quantitative ventilation assessments of idealized urban canopy layers with various urban layouts and the same building packing density, Build. Environ. 79 (2014) 152-167.

[3] H. Montazeri, B. Blocken, CFD Simulation of Wind-induced Pressure Coefficients on Buildings with and Without Balconies: Validation and Sensitivity Analysis, Build. Environ. 60 (2012) 137-149.
[4] Y. Tominaga, A. Mochida, S. Murakami, S. Sawaki, Comparison of Various Revised K-ε Models and Les Applied to Flow Around a High-rise Building Model with 1:1:2 Shape Placed Within the Surface Boundary Layer, J. Wind. Eng. Ind. Aerod. 96 (2008) 389.
[5] W. Rodi, Comparison of Les and Rans Calculations of the Flow Around Bluff Bodies, J. Wind. Eng. Ind. Aerod. 71 (1997) 55.
[6] Y. Tominaga, A. Mochida, R. Yoshie, H. Kataoka, T. Nozu, Yoshikawa M., Shirasawa T, AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, J. Wind. Eng. Ind. Aerod. 96 (2008) 1749-1761.
[7] Y. Tominaga, T. Stathopoulos, Numerical Simulation of Dispersion Around an Isolated Cubic Building: Comparison of Various Types of K-ε Models, Atmos. Environ. 43 (2009) 3200.
[8] J.Z. Zhai, Z. Zhang, W. Zhang, Q. Chen, Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments By Cfd: Part 1-summary of Prevalent Turbulence Models, HVAC&R Res, 13 (2007) 853-870.
[9] Z. Zhang, W. Zhang, J.Z. Zhai, Q. Chen, Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments By Cfd: Part 2-comparison with Experimental Data From Literature, HVAC&R Res, 13 (2007) 871-886.
[10] B. Li, J. Liu, M. Li, Wind Tunnel Study on the Morphological Parameterization of Building Non-uniformity, J. Wind. Eng. Ind. Aerod. 121 (2013) 60-69.
[11] B. Blocken, J. Carmeliet, T. Stathopoulos, CFD Evaluation of Wind Speed Conditions in Passages between Parallel Buildings—effect of Wall-function Roughness Modifications for the Atmospheric Boundary Layer Flow, J. Wind. Eng. Ind. Aerod. 95 (2007) 941-962.
[12] J. Dacles-Mariani, P. Bradshaw, J.S. Chow; G.G. Zilliac, Numerical/experimental Study of a Wingtip Vortex in the Near Field, AIAA J, 33 (2012) 1561-1568.