Vol 15 No 1, January 2006
1009-1963/2006/15(01)/0149-07

Chinese Physics

© 2006 Chin. Phys. Soc.
and IOP Publishing Ltd

Numerical simulation of the flow within and over an intersection model with Reynolds-averaged Navier–Stokes method*

Li Lei(李 磊)[a)b)], Hu Fei(胡 非)[a)], Cheng Xue-Ling(程雪玲)[a)†]
Jiang Jin-Hua(姜金华)[a)], and Ma Xiao-Guang(马晓光)[a)]

[a)] *State Key Laboratory of Atmospheric Boundary Layer Physics and Atmospheric Chemistry, Institute of Atmospheric Physics, Chinese Academy of Sciences, Beijing* 100029, *China*

[b)] *Beijing Meteorological Bureau, Beijing* 100089, *China*

(Received 20 May 2005; revised manuscript received 29 July 2005)

In this study, the Reynolds-averaged Navier–Stokes (RANS) method is employed to simulate the flow within and over an intersection model with three kinds of k–ε turbulence closure schemes, namely, standard model, renormalization group (RNG) model and realizable k–ε model. The comparison between the simulated and observed flow fields shows that the RANS simulation with all the three turbulence models cannot completely and accurately reproduce the observed flow field in all details. A detailed comparison between the predicted profiles of wind velocities and the measured data shows that the realizable k–ε model is the best one among the three turbulence closure models in general. However, the extent to which the improvement is achieved by the realizable k–ε model is still not enough to completely and accurately describe the turbulent flow in a relatively complex environment.

Keywords: RANS, numerical simulation, intersection, turbulence closure model
PACC: 4700, 4725

1. Introduction

The surface of an urban area is covered with all kinds of elements, such as buildings, concrete sealed grounds, and vegetated gardens. The complexity of the urban surface heavily disturbs the flow over the urban area and makes it difficult to predict the flow fields in the urban canopy layer. Numerical simulation is one of the most important ways to improve our understanding of the surface–atmosphere interaction in urban areas and the characters of the disturbed flow between buildings, which can help solve some significant problems related to the daily lives of urban residents, such as wind comfort/safety problems and air pollution issues in urban streets.

In recent years, computational fluid dynamics (CFD) has been applied in the numerical simulation of the flow around buildings and the pollutant transport process in streets.[1−3] Though much effort has been exerted to develop new concepts and new methods to improve and extend the capabilities of CFD,[4−6] traditional Navier–Stokes equations are still the cornerstone of CFD codes applied in practical studies. Based on Navier–Stokes equations, two methods, namely the large-eddy simulation (LES) method and the Reynolds-averaged Navier–Stokes (RANS) method are developed to model the turbulent flows. Generally speaking, the LES method can afford results with more details than the RANS method, but it also requires grid setting with much smaller cells, and in consequence, more computation resources, which may be far beyond the capabilities of the personal computers widely used in operational jobs nowadays. Thus, until now, in most practical studies or operational jobs, the RANS method is still the first choice.

As is known, RANS equations cannot constitute a close system, and additional equations should be added to cope with the so-called turbulence closure trouble. In the past decades, quite a few turbulence schemes have been developed in order to close the system of RANS equations, which have different advan-

*Project supported by the National Natural Science Foundation of China (Grant Nos 40233030, 40405004 and 40405014) and the Special Program of the Scientific and Social Practices for Graduate Students in Chinese Academy of Sciences, China.
†Corresponding author. E-mail: chengxl@mail.lapc.ac.cn

tages and deficiencies. Thus, one of the key factors to determine the accuracy of CFD numerical simulation is the appropriate turbulence scheme adopted in the simulation. Unfortunately, as pointed out by Gosman,[7] no clearly superior turbulence model can work well over a wide range of applications, and the only reliable guide to select the turbulence closure model is the obtained experience on a similar problem. In previous studies, many comparisons have been performed among k–ε turbulence models by use of wind tunnel data.[8−10] However, the configurations of all these wind tunnel experiments are quite simple. For example, Lien and Yee[8] compared standard and Kato–Launder k–ε models with wind tunnel data, and the wind tunnel experiment was carried out to simulate the neutrally stratified atmospheric boundary-layer flow over an array of blocks, which are very regularly arranged and cannot be compared with urban blocks in real cities. In the authors' previous study, three kinds of k–ε models integrated in CFD codes FLUENT were validated and compared, and the wind tunnel data used in this study were acquired from a wind tunnel experiment with even simple configuration, in which only a single block was mounted in the wind tunnel to depict a building.[9] Though the results of these previous studies have already shown that the RANS method can reasonably reproduce the turbulent flow around structures, it is still necessary to make further effort to validate the RANS method with data obtained from wind tunnel experiments with more complex configuration which should be more realistic than previous ones and can be compared with real urban blocks to a certain extent.

In this paper, a numerical study of turbulent flow within and over an intersection model is undertaken with the RANS method, and three k–ε models, namely, standard k–ε, renormalization group (RNG) k–ε and realizable k–ε models are validated and compared.

2. Description of wind tunnel data

Leitl and Schatzmann[11] introduced the compilation of experimental data for validation of microscale dispersion models (CEDVAL) in 1998 and have been updating this data set since then. As stated by Leitl and Schatzmann, the primary goal of CEDVAL is to provide the validation of data at a higher level of quality than most of the data available can provide so far. All data sets within CEDVAL follow a high-quality standard in terms of complete documentation of boundary conditions and quality assurance during measurements.[11]

The subset labelled with B1-4 of CEDVAL was gathered from the BLASIUS wind tunnel at the University of Hamburg, and the B1-4 experiment was carried out to simulate the neutrally-stratified flow within and over a three-dimensional intersection model. As illustrated in Fig.1, the intersection consists of four square-shaped rings of model buildings forming an idealized intersection with an offset between two lanes in

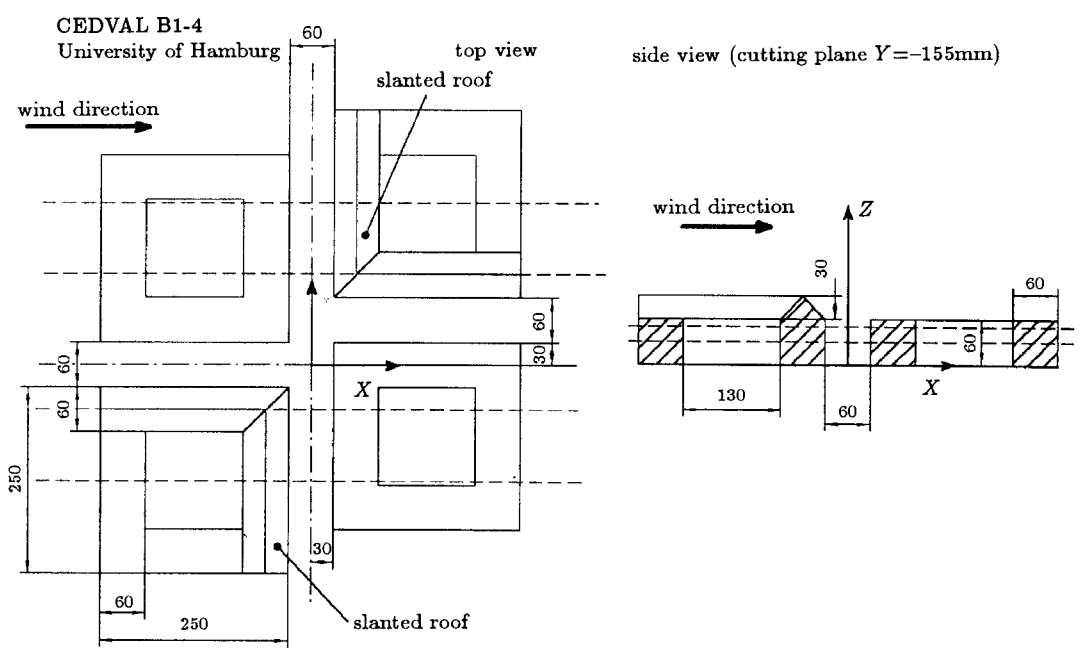


Fig.1. Sketch map of the B1-4 wind tunnel experiment of CEDVAL (unit: mm)[11].

the x-direction. Two of the four buildings surrounding the intersection have been equipped with a slanted roof (45° roof angle). It can be seen from Fig.1 that the geometric shape and the layout of the model buildings in B1-4 experiment are complex enough to be compared with some real blocks in urban area.

The B1-4 experiment was carried out under a neutrally stratified condition and the approach flow at the inlet side was of a power-law profile, which can be expressed as $u = u_0 \left(\frac{z}{0.06}\right)^{0.22}$, where u is the velocity of approach flow (m/s), z is the vertical coordinate (m), and u_0 is the flow velocity at a height of 0.06m, and its value was set as 5.37m/s in the B1-4 experiment.

3. Numerical calculation

As the flow velocity in wind tunnel experiment is much slower than acoustic speed, the atmosphere can be assumed to be incompressible here and the RANS governing equations can be expressed as follows:

Momentum equation:

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\mu}{\rho}\frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j}(\overline{u_i' u_j'}) + f_i. \quad (1)$$

Continuity equation:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0. \quad (2)$$

Here, $\overline{u_i}$ is the average velocity of air flow, $\overline{u_i' u_j'}$ is the Reynolds stress, ρ is the density of atmosphere, μ is the molecular viscosity, and f_i is the body force. It can be seen that the Reynolds stress is a new unknown term in the RANS equations and needs further equations to be predicted. Launder and Spalding[12] put forward a closure scheme named as standard k–ε model in which two new equations were introduced to predict the turbulent kinetic energy k and its dissipation rate ε. And these two physical quantities can help to determine the value of Reynolds stress and close the RANS equations. Using a mathematical technique called "renormalization group" (RNG) methods, Yakhot and Orszag[13] derived another closure scheme, which also consists of two equations for predicting k and ε and was called as RNG k–ε model. Compared with the former two models, the realizable k–ε model developed by Shih *et al* is a relatively new one.[14] This model is also expressed with two main equations for predicting k and ε. The k equation in the realizable k–ε model is the same as that in the other two k–ε models, however, the ε equation was modified to satisfy certain mathematical constraints on the normal stresses, consistent with the physics of turbulent flows. The most noteworthy feature is that the production term in the ε equation does not involve the production of k, and this modification is believed to better represent the spectral energy transfer of turbulent flows. These three models are used in the numerical simulation of this study, whereas their associated equations are not listed in this paper, for they can easily be found in original references.

Commercial CFD codes FLUENT was employed in the numerical calculation. The computational model of intersection has been set before the calculation was carried out. A cuboid with a 4m×1m×1m geometric size was set as a calculation domain, and the outflow and the velocity inlet boundaries were specified at the outlet and the inlet sides of the calculation domain respectively, while the non-slip wall boundaries were specified at the ground, the surfaces of structures and the surrounding walls of the wind tunnel. The whole calculation domain was meshed with hexahedral cells, and the grid in the areas near the structures was adapted into greater cell density in order to capture enough flow details in the area adjacent to the structures. Finally, the SIMPLE algorithm was used to solve the governing equations.[15]

4. Results and analysis

A coarse grid of about 540000 cells and a fine grid of about 730000 cells were respectively employed in the simulation to investigate the dependence of the solutions on grid resolution, and the data comparison performed at x=0.015m in the vertical plane of y=0.120m shows that there is a good agreement between the coarse-grid and the fine-grid solutions. Thus, the simulations with fine grid presented in this paper can be regarded as grid-independent.

Figure 2(a) shows the observed average flow field within and above the street canyon in the vertical plane of y=3.5H provided by Leitl and Schatzmann,[11] and Figs.2(b)–(2d) show the simulated results with three k–ε models, where H, the reference height in the B1-4 wind tunnel experiment, is set as 0.06m . It should be noted that the lines over the surfaces and roofs of the structures in Figs.2(b)–(2d) indicate the area where the grid has been adapted into more cell density. In Figs.3, the observed and simulated results in the vertical plane of y=–2.58H

are illustrated.

It can be seen from Fig.2(a) and Fig.3(a) that there exist vortices observed in the vertical planes, and the vortices also appear in the simulated results from all the three turbulence models. In Fig.2, there is no obvious difference among the simulated flow patterns from the three turbulence models, and the central positions of the simulated vortices from all the models are quite similar. In addition, the simulated results are also quite close to the observed results in Fig.2, and the centres of the simulated vortices are almost at the right location when compared with the observed results. But the simulated results in Fig.3 are worse than those in Fig.2. The central position of the observed vortex in the vertical plane of y=–2.58H is behind the slanted roof and above the street canyon, while the central positions of the simulated vortices are below the roof and within the street canyon for all turbulence models. Thus it can be concluded that the numerical simulation based on the RANS method cannot completely and accurately reproduce all details of the observed flow field and no turbulence model can present much more accurate results than other ones when applied to a case with a complex configuration as far as the central positions of the vortices is concerned.

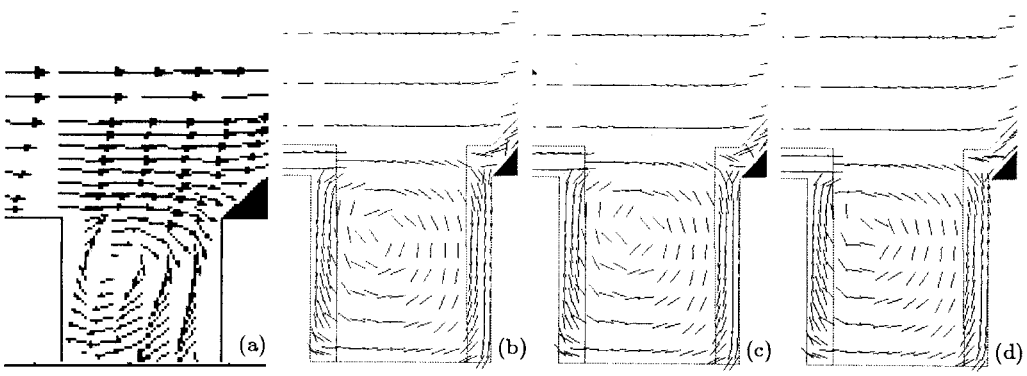


Fig.2. The comparison between the measured and calculated flow field in the vertical plane of y=3.5H (a) measured, (b) standard k–ε model, (c) RNG k–ε model, (d) realizable k–ε model).

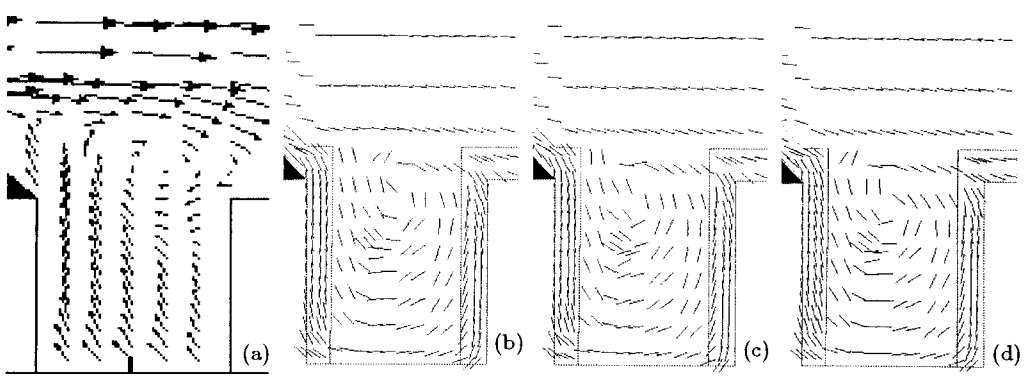


Fig.3. The comparison between the measured and calculated flow fields in the vertical plane of y=–2.58H (a) measured, (b) standard k–ε model, (c) RNG k–ε model, (d) realizable k–ε model).

In order to take a further evaluation of the simulation results, there were in total six locations selected to perform the comparison between the measured and the calculated data. These six locations cling to the lines displayed in Fig.4, and two of them are located within the street canyon and parallel to the y-axis, while the other four are parallel to the z-axis. The details of these six locations are listed in Table 1, and the values of the coordinates in Table 1 are all normalized by the reference height H (0.06m).

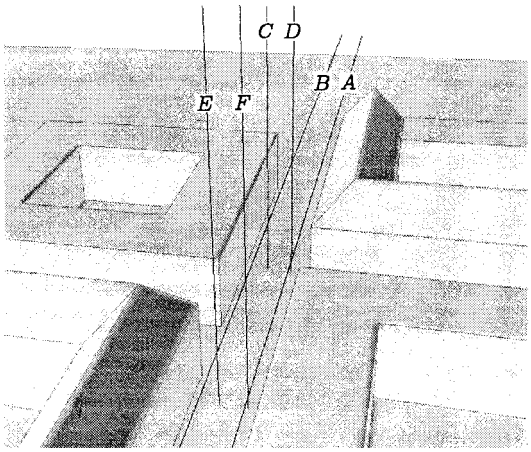


Fig.4. The six locations selected to perform the data comparison, the details of the location A to F are listed in Table 1.

Table 1. The details of the six locations selected to perform the data comparison.

Location	Range of x/H	Range of y/H	Range of z/H
A	0.25	3.883~5.583	0.5
B	–0.25	3.883~5.583	0.5
C	–0.167	2.0	0.083~1.167
D	0.167	2.0	0.083~1.167
E	–0.167	–1.0	0.083~1.167
F	0.167	–1.0	0.083~1.167

The profiles of both measured and predicted wind velocities at the six locations are illustrated in Fig.5 and Fig.6. Here Fig.5 corresponds to the u component while Fig.6 correspond to the v or w component. It should be pointed out that the values of the wind velocities and the coordinates are all normalized by

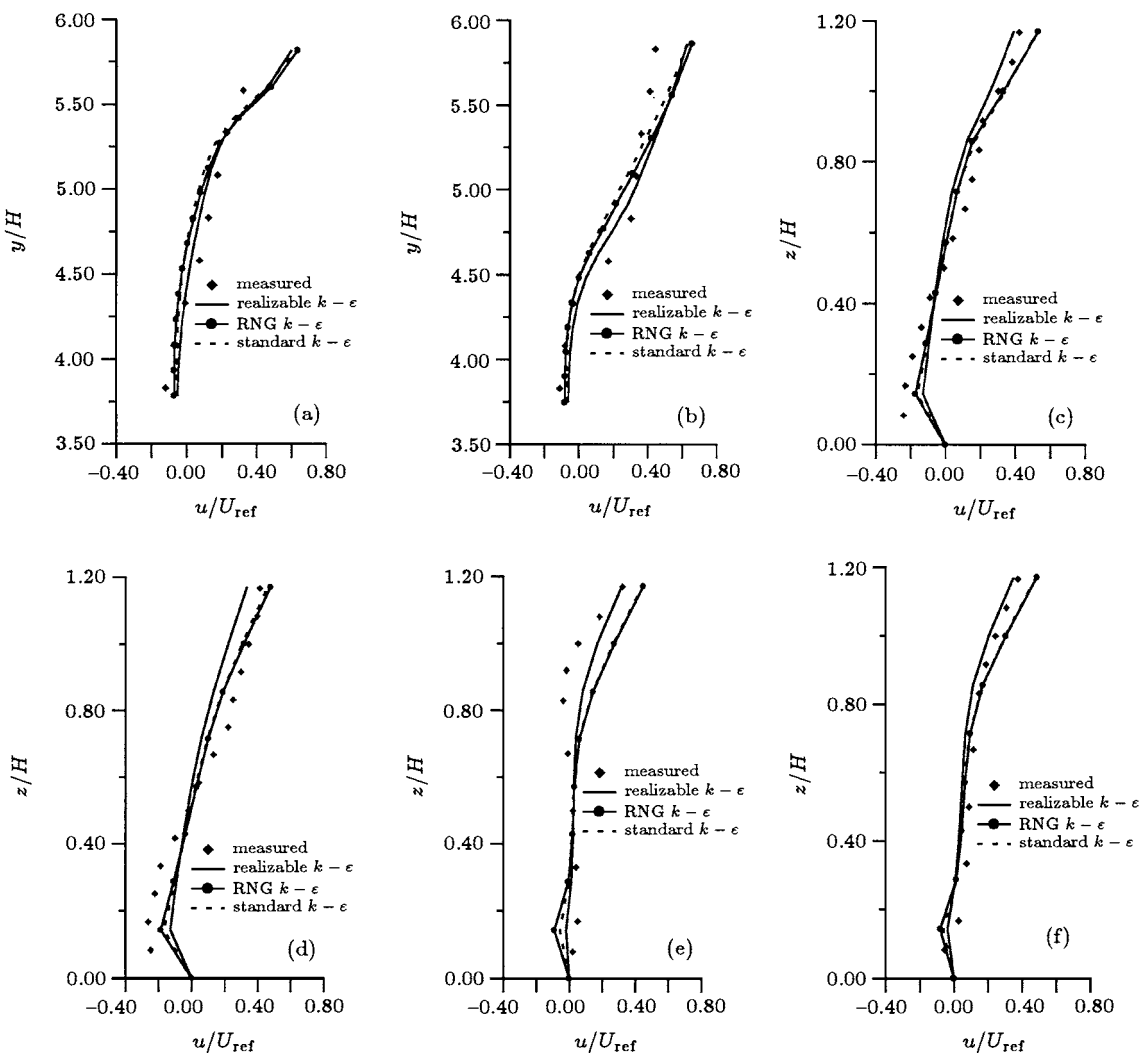


Fig.5. The comparison between measured and predicted profiles of wind velocity (u component).

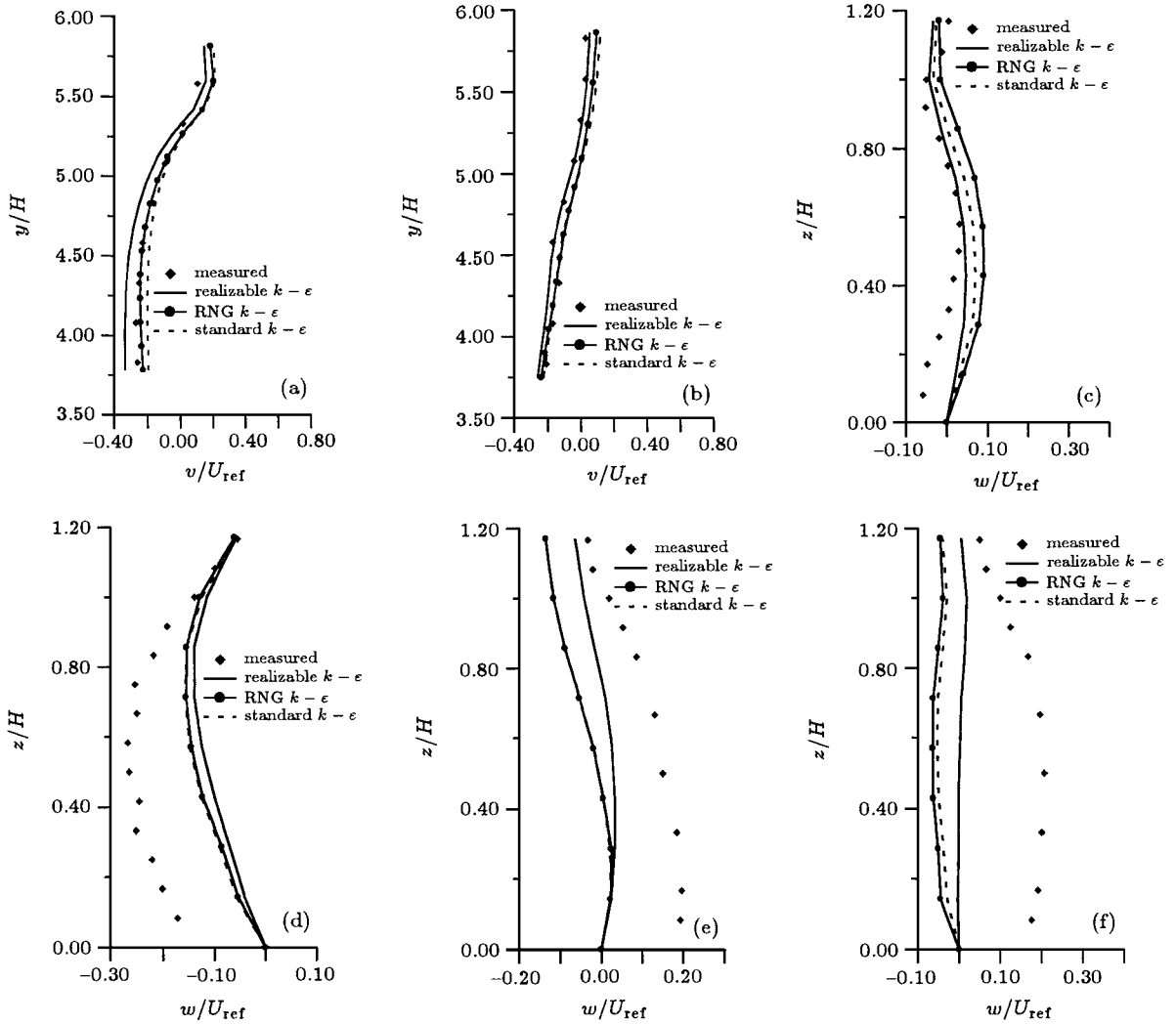


Fig.6. The comparison between measured and predicted profiles of wind velocity (v or w component).

the reference velocity $U_{\rm ref}$ (6m/s) and reference height H (0.06m) respectively. In Figs.5 and 6, the labels (a) to (f) just correspond to the locations A to F listed in Table 1. From Figs.5 and 6, it can be found that the predicted data from the standard k–ε model and RNG k–ε model are always very similar, while the predicted data from the realizable k–ε model are quite different from those obtained with the other two turbulence models.

In Fig.5, the variation trends of predicted u components and measured data against the height are quite similar for all turbulence models, and the predicted profiles are close to the measured data. However, it is hard to tell which turbulence model works better in predicting the u component of wind velocity at the locations A and B, for the differences among the results from the three k–ε models are not obvious in Figs.5(a) and 5(b). In Figs.5(c), 5(e) and 5(f), the predicted results from the realizable k–ε model are slightly closer to the measured results than those from the other two models. However, in Fig.5(d), the difference between the predicted data from the realizable k–ε model and measured data is a little larger than the that between the predicted data from either of the other two models and the measured data.

Figure 6 compares the data of v or w. The predicted profiles are in good agreement with the measured data in Figs.6(a) and 6(b), which shows that the predicted v components of wind velocity are quite accurate. As is seen in Figs.6(a) and 6(b), the realizable k–ε model works worse than the other two at the location A but better at the location B. From Figs.6(c)–6(f), it can be seen that the predicted values of w components from the three turbulence models are all very poor. The difference between the measured data and the predicted data is dramatically large, especially in Fig.6(f), there is even no comparability between the measured data and the predicted data in the

variation trends of w components against the height. A significant improvement in the predicted data from the realizable k–ε model can be seen in Figs.6(c), 6(e) and 6(f), but the extent to which the improvement is achieved is still not enough to obtain the results that can be called accurate. However, it should be pointed out that the magnitude of w components is almost one order less than that of u components, so, though the predicted w components are poor for all k–ε models, the RANS simulated results are still quite acceptable on the condition that only the magnitude of wind velocity draws attention in a study.

In the previous studies, the realizable k–ε model was reported to work better than the other two k–ε models,[9] for it could quite accurately reproduce the measured data. However, in this study, all k–ε models including the realizable k–ε model cannot completely and accurately reproduce all measured data when employed to simulate the flow over a geometrically complex intersection model, but the comparison between predicted profiles and measured data shows that the realizable k–ε model is still the best one among the three turbulence models in general.

5. Conclusions

The RANS method was employed to simulate the flow within and over an intersection model with three k–ε turbulence closure models, namely, standard k–ε, model RNG k–ε model and realizable k–ε model. The CFD codes FLUENT are used in the numerical simulation, and the simulated results are compared with the wind tunnel data (B1-4 subset of CEDVAL) obtained from Hamburg University. The configuration of the B1-4 wind tunnel experiment is much more complex than those cited in previous studies, and this complexity will help validate the capability of the RANS method in a more realistic situation.

The comparison between the simulated and observed flow fields shows that the RANS simulation with all the three turbulence models cannot completely and accurately reproduce the observed flow field in all details. The simulated central positions of the vortices behind the slanted roof of the structure are obviously wrong. A detailed comparison between predicted profiles of wind velocity components and measured data are also made, which shows that though the RANS method can provide reasonable profiles for u and v components of wind velocity, it cannot accurately reproduce the profile of w component. In addition, the realizable k–ε model is proved to be the best one among the three turbulence closure models in general. However, the extent to which the improvement is achieved by the realizable k–ε model is still not enough to completely and accurately describe the turbulent flow in a relatively complex environment.

The results of this study show that the RANS method may have some inherent defects, which seem unable to be conquered with mathematical skills. These defects may restrict the application of the RANS method to complex real environments when researchers are especially concerned with the details of flow, though the RANS method is still employed in many operational jobs in a long time because of its advantages of saving computational time. In the future, with the rapid development of the computer science, it can be expected that the LES method will be used in a much wider area and may have a more important status in the field of numerical simulation.

Acknowledgments

The authors would like to thank Dr Leitl and his colleagues for their gathering the wind tunnel data of high quality and kindly keeping these data open to all researchers.

References

[1] Ferreira A D, Sousa A C and Viegas D X 2002 *J. Wind Eng. Ind. Aerod.* **90** 305
[2] He J and Song C S 1999 *J. Wind Eng. Ind. Aerod.* **81** 295
[3] Cheng X L and Hu F 2005 *Adv. Atmos. Sci.* **22** 290
[4] Zhou X Y, Shi B C and Wang N C 2004 *Chin. Phys.* **13** 712
[5] He N Z, Wang N C, Shi B C *et al* 2004 *Chin. Phys.* **13** 40
[6] Feng G L, Cao H X, Dong W J *et al* 2001*Chin. Phys.* **10** 1004
[7] Gosman A D 1999 *J. Wind Eng. Ind. Aerod.* **81** 21
[8] Lien F S and Yee E 2004 *Bound. Lay. Meteorol.* **112** 427
[9] Li L, Hu F and Cheng X L 2004 *Chin. Phys.* **13** 1070
[10] Chang C H and Meroney R N 2001 *J. Wind Eng. Ind. Aerod.* **89** 1325
[11] Leitl B and Schatzmann M 1998 *Compilation of Experimental Data for Validation Purposes, CEDVAL* (Meteorology Institute, Hamburg University)
[12] Launder B E and Spalding D B 1972 *Lectures in Mathematical Models of Turbulence* (London: Academic)
[13] Yakhot V and Orszag S A 1986 *J. Sci. Comput.* **1** 1
[14] Shih T H, Liou W W, Shabbir A *et al* 1995 *Comput. Fluids.* **24** 227
[15] Patankar S V 1980 *Numerical Heat Transfer and Fluid Flow* (New York: Hemisphere)