Vol 13 No 7, July 2004
1009-1963/2004/13(07)/1070-06

Chinese Physics

© 2004 Chin. Phys. Soc.
and IOP Publishing Ltd

The application of computational fluid dynamics to pedestrian level wind safety problem induced by high-rise buildings*

Li Lei(李 磊)[a)b)†], Hu Fei(胡 非)[a)]
Cheng Xue-Ling(程雪玲)[a)], and Han Hao-Yu(韩浩玉)[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)] *Graduate School of Chinese Academy of Sciences, Beijing* 100039, *China*

(Received 3 November 2003; revised manuscript received 18 March 2004)

In this paper, the pedestrian level wind safety problem induced by high-rise buildings has been studied using the computational fluid dynamics (CFD) codes Fluent. The verification by use of wind tunnel data shows that Fluent can fairly reproduce the flow field in the areas adjacent to the structure when a realizable k-ε turbulence model is adopted in calculations. The results of the numerical simulations including seven cases show that the existence of high-rise buildings does increase the wind hazard probability at the pedestrian level; furthermore, the wind direction, the geometric size of structures and the layout of structures can obviously affect the pedestrian level wind environment. However, trees on the pavement do not contribute much in reducing the wind hazard probability.

Keywords: CFD, pedestrian level wind safety, high-rise building, realizable k-ε turbulence model
PACC: 4700, 9260G

1. Introduction

Over the last 20 years, with the rapid development of the Chinese economy and society, big cities like Beijing have become more and more crowded. Accompanied with the increase of population in big cities, more and more high-rise buildings have appeared, which can provide a large number of dwelling units per unit space. However, as introduced by Aynsley,[1] many problems are induced by the emergence of high-rise buildings, among which the most serious one is increase of high wind hazards, such as knocking pedestrians to ground and hurting them. It is reasonable to associate the wind hazards with the layout and the geometry of high-rise buildings, thus an assessment of the pedestrian wind environment around high-rise buildings is necessary during the design phase so that some modifications can be made to avoid such hazards.

In the past, most assessments on the wind environment around buildings were carried out by wind tunnel test, which is apparently not a convenient and economical method for assessment. In fact, it is impossible to conduct assessment on every high-rise building project by wind tunnel test in a developing country like China. Fortunately, the rapid growth of computational fluid dynamics (CFD) provides a new and economical way to solve this problem. As reviewed by Murakami *et al*,[2] the CFD technique has already been applied to the analysis of wind climate from human scale to urban scale. Furthermore, accompanied by the introduction of new concepts,[3,4] new schemes [5,6] and possible new turbulence models into the CFD codes, the application of CFD can be extended to further areas. At present, CFD has been taken as a very useful tool to assess the wind environment around structures. He and Song [7] evaluated the pedestrian winds in urban area using the CFD approach. Ferreira *et al* [8] analysed the interference effect created by two auxiliary buildings located upstream of a recreation area in Lisbon. However, most studies on wind environment by using CFD are concentrated on pollutant diffusion analysis around structures, while many wind safety problems induced by

*Project supported by the National Natural Science Foundation of China (Grant No 40233030).
†Corresponding author. E-mail: lil@mail.lapc.ac.cn

high-rise buildings, such as the relationship between the risk probability of hurting pedestrians and the wind direction, are not yet answered.

In this paper, the pedestrian level wind safety problem induced by high-rise buildings is studied by commercial CFD codes Fluent. The main target of this study is to simulate the wind environment around high-rise buildings in an idealized urban community and to evaluate the wind hazards under different conditions. Before detailed numerical tests are conducted, the accuracy of the CFD codes Fluent is verified using the wind tunnel data. Based on the numerical simulation results, seven cases are compared to each other, from which some facts on the pedestrian level wind safety issue are concluded.

2. Verification of CFD tool fluent

As is known, the foundation of the CFD technique is the Navier–Stokes equation set. One of the key factors to determine the accuracy of the result obtained from the Navier–Stokes equation set is the selected turbulence closure scheme. In Ref.[9], Fluent was used to simulate the flow field in model urban street canyons. By contrast with the result obtained from wind tunnel, it was concluded that both standard k-ε model and renormalized group theory k-ε (RNG k-ε) model can provide fine results. However, in a review published in 1999, Gosman [10] pointed out that 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 previous experience on a similar problem. Thus verifying the CFD tool Fluent and selecting the appropriate turbulence closure scheme is necessary before further detailed case studies are conducted.

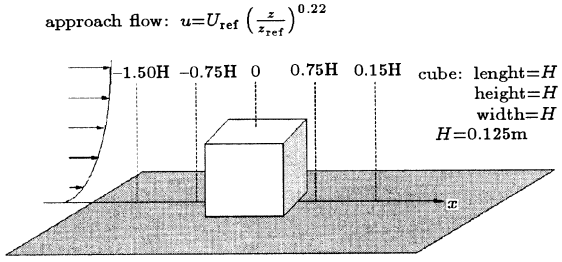


Fig.1. The sketch map of the Blasius wind tunnel measurement.

In this paper, the verification job is conducted using the wind tunnel data acquired from the Blasius wind tunnel, University of Hamburg.[11] In the wind tunnel measurement, a cube (0.125m×0.125m×0.125m in model scale) is set as an idealized model building of 25m in size (in full scale). As described in Ref.[11], the measurement is carried out under a neutrally stratified condition and the approach flow is of a power-law exponent 0.22 profile. The selected positions to conduct the comparison between the measured and simulated data are shown in Fig.1.

In the numerical simulation, a cuboid of 3m×1m×1m size is set as the calculation domain. Outflow and velocity inlet boundaries are specified at the outlet and inlet sides of the calculation domain, while symmetric boundaries are specified at the lateral and top sides.

How to properly set the grid in the areas adjacent to the structures is also an important factor in the pedestrian level wind safety issue, for the flows in the boundary layer over the structures are very complicated and whether the grid is properly set in the boundary layer or not decides the accuracy of the numerical simulation in the whole calculation domain. In order to capture the details of the flow field in the areas adjacent to the structures, the grid should be adapted with greater cell density in these areas. As shown in Fig.2, the grid in the areas near the cube was adapted into much greater cell density than in other areas in the numerical simulation.

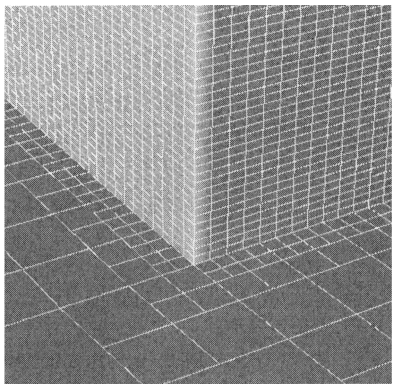


Fig.2. The grid in the areas adjacent to the cube was adapted into greater cell density.

The most significant feature of the flow near the structures is associated with the high Reynolds number, and many turbulent, twisted and vortical flows are induced by the structures, thus the distribution of wind energy is very complicated. A proper turbulence model is a determinative factor to accurately describe these features. In order to choose an appropriate turbulence closure scheme, three models have been compared in the verification, which are the standard k-ε,

the RNG k-ε and the realizable k-ε models. Compared with the other two, the realizable k-ε model is a relatively new turbulence closure model, which was developed by Shih *et al*[12] in order to compensate for the deficiencies of the other two k-ε models. The detailed equations are not listed in this paper for they can easily be found in the original paper;[12] however, the features of this model should be stressed here. The model is expressed with two main equations: one for turbulent kinetic energy (k) and the other for its dissipation rate (ε). The k equation of the realizable k-ε model is the same as that in the other two k-ε models; however, the ε equation is 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.

Detailed analysis on numerical simulation results shows that the realizable k-ε model does work much better than the other two models. The simulated flow field near the cube using the realizable k-ε model is illustrated in Fig.3. The flow pattern, the reattachment length and the position of the vortices are in fair accordance with the measured results illustrated in Ref.[11].

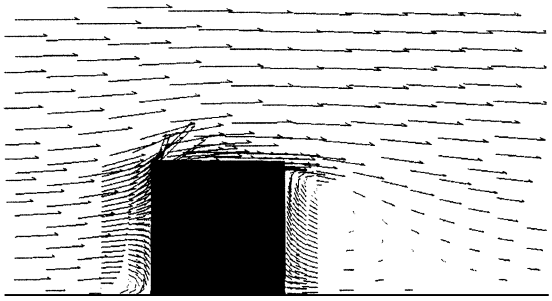
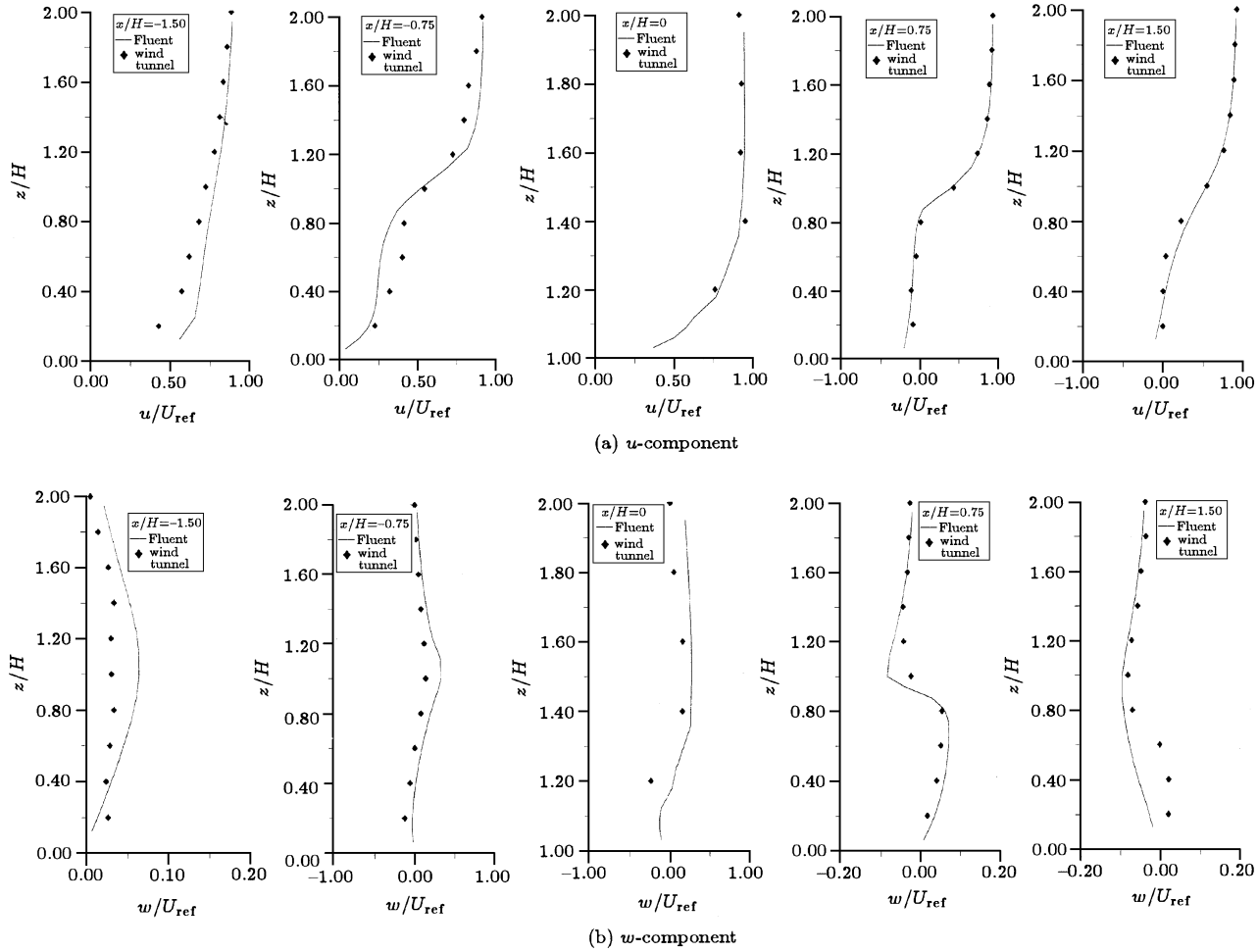


Fig.3. The pattern of the flow field near the cube.



(a) u-component

(b) w-component

Fig.4. The comparison between the numerical simulation using Fluent and the wind tunnel data.

Figure 4 shows the comparison between the simulated and measured data below the height of $2H$ (0.25m), where only the results obtained using the realizable k-ε model are presented. In Fig.4, both u and w are normalized by a reference velocity $U_{\rm ref}$ (5.0m/s), while the z-coordinate in the vertical direction is normalized by the cube height H (0.125m). Figure 4(a) shows the vertical profiles of the u-component at different positions illustrated in Fig.1, while Fig.4(b) shows the profiles of the w-component. It can be seen that the agreement between the simulated and the measured u-component data is quite good when the realizable k-ε model is applied in the simulation, especially at x/H=0, 0.75 and 1.5. Though the simulated and the measured u-component data at x/H=–0.75 and–1.5 do not match as well as at the former three positions, the variation tendencies of the simulated and the measured data against the height are in accord at the latter two positions. However, the agreement between the measured and the simulated w-component data is relatively poor, especially at x/H=–1.5. Fortunately, similar variation tendencies of both the simulated and the measured w-component data against the height can be found at x/H=-0.75, 0, 0.75 and 1.5. Furthermore, it should be noted that the magnitude of the w-component is nearly one order less than that of the u-component, from which it can be concluded that the accuracy of the Fluent simulation results is acceptable when the realizable k-ε model is adopted, for in the wind safety issue the magnitude of the total wind velocity is the most important factor.

3. Numerical tests and assessment criterion

Based on the verification in Sec.2, it can be concluded that the realizable k-ε model is preferred in this study, for it can quite accurately reproduce the flow field in the areas adjacent to the isolated structure, and these areas concern us most in this study. Thus the realizable k-ε turbulence model is adopted in detailed numerical tests.

The layout of the idealized community model is illustrated in Fig.5. Blocks A and B are mounted as high-rise buildings, the geometric size of which will be changed in order to study different wind environments under different conditions. In addition, there are a total of 27 blocks of 10m in height and 15m in width in the calculation domain: 15 of these blocks are set to depict low-rise structures on the other side of the main street, while the others represent structures on the same side as the high-rise buildings. The shortest distance between the two block groups is 40m, which is a normal width of main streets including pavements in Chinese big cities like Beijing.

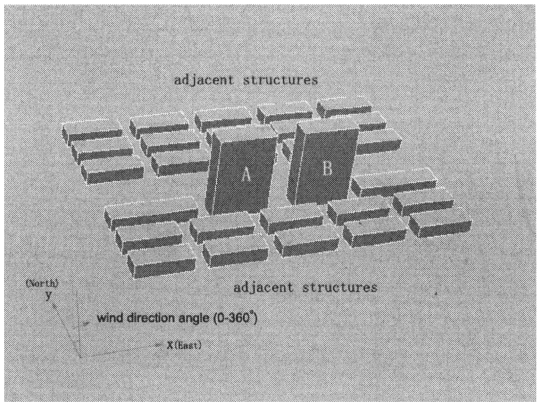


Fig.5. Schematic map of the idealized community.

There are in total seven cases designed in this study, as given in Table 1. The influence of different factors, such as the high-rise building geometric size, the distance between two nearby buildings and the wind direction, on the wind environment is expected to be analysed from these cases. It should be mentioned that in all cases the approach flow of a $U_{\rm ref}(z/z_{\rm ref})^{0.14}$ profile is adopted on the inflow boundaries, where $U_{\rm ref}$ is the wind velocity at the reference height (10m). The magnitude of $U_{\rm ref}$ is set at 5.0m/s here, which is a wind velocity measured frequently at the height of 10m in Beijing. In all cases, outflow and velocity inlet boundaries were specified at the outlet and inlet sides of the calculation domain, while a symmetric boundary condition was specified at the top. In addition, the grid in the boundary layer covering the structures was adapted into one of much greater cell density, as mentioned in Sec.2.

The evaluation on the wind hazards should be based on a set of criteria. As stated in Ref.[13], many different wind comfort criteria (including wind safety criteria) have been proposed in recent decades, most of which are expressed in a wind speed framework, since wind speed is directly related to the comfort and safety of pedestrians. Based on previous studies, Bottema [13] developed a set of methods for optimizing wind comfort criteria based on the framework of $U/U_{\rm ref}$, where U is the local mean wind speed and $U_{\rm ref}$ (the symbol $U_{\rm pot}$ was adopted in Ref.[13]) is the mean hourly wind speed at the height of 10m over an ideal meteorological site. In the light of Bottema's

study, discomfort and danger probability can be expressed as a function of a direction-independent wind velocity amplification factor $U/U_{\rm ref}$, thus a comfort or danger threshold of $U/U_{\rm ref}$ can be calculated. In Ref.[13], the danger thresholds of $U/U_{\rm ref}$ in different cities have been presented, most of which vary near the value of 0.7. Therefore, 0.7 is set as the threshold value of $U/U_{\rm ref}$ for the evaluation of the wind hazards in the next section. A value of $U/U_{\rm ref}$ higher than the threshold value in a certain area means that there is a probability to hurt pedestrians there; and accompanied with the increase of the value of $U/U_{\rm ref}$, the hazard probability also increases.

Table 1. Numerical simulation cases.

No.	Wind incident angle/(°)	Height of A/m	Height of B/m	Shortest distance between A and B/m	Additional models
1	0	60	60	20	None
2	180	60	60	20	None
3	315	60	60	20	None
4	0	60	60	10	None
5	0	20	60	10	None
6	0	20	20	20	None
7	0	60	60	20	Trees

4. Results and analysis

Fluent can describe the flow field in the whole calculation domain in detail. As an example, the flow field near the structures at the height of 2m in the calculation domain is illustrated in Fig.6, which shows the result of case 1. From Fig.6, it can be found that the flow field near the structures at this height is very complicated and many vortical and twisted flows are formed in the areas adjacent to the structures.

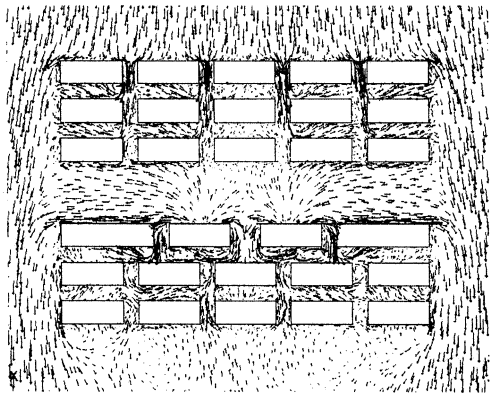


Fig.6. The flow field near structures at the height of 2m (case 1).

Table 2. The maximum value of $U/U_{\rm ref}$ in the space between structures A and B at the height of 2m.

Case	1	2	3	4	5	6	7
Maximum $U/U_{\rm ref}$	0.98	1.12	1.35	1.07	1.16	0.67	0.97

The maximum values of $U/U_{\rm ref}$ in the space between structures A and B at the height of 2m are presented in Table 2, and further details about the results in the simulations are listed below:

(A) Only one of the cases (case 6) meets the safety demand, in which the maximum value of $U/U_{\rm ref}$ (0.67) does not exceed the danger threshold 0.7. In this case, the maximum height of buildings is 20m. However, in all the other cases, there is at least one building of 60m height. Thus it can be concluded that the existence of high-rise buildings does increase the wind hazard probability at the pedestrian level.

(B) The value of $U/U_{\rm ref}$ is sensitive to wind direction when the layout of the structures is fixed. By comparing cases 1, 2 and 3, the influence of wind direction on wind environment between structures A and B can be seen. In case 1, the wind incident angle is 0°, which means the approach flow comes from the north and is parallel to the central line between A and B, and the maximum value of $U/U_{\rm ref}$ is 0.98. In case 2, the approach flow comes from the northwest and the maximum value of $U/U_{\rm ref}$ (1.12) is higher than that in case 1. In case 3, the approach flow comes from south and is also parallel to the central line between A and B. In this case, the maximum value of $U/U_{\rm ref}$ (1.35) is even higher and the expected shelter effect of the most adjacent structures does not appear.

(C) By comparing cases 1 and 4, the influence of the distance between A and B can be seen. The space between A and B is narrower in case 4 than in case 1. Consequently, the maximum value of $U/U_{\rm ref}$ is higher in case 4 (1.07) than in case 1, which has been expected before simulations are conducted. This comparison shows that the narrow space between structures does increase the wind hazard probability.

(D) In the light of the results of cases 4 and 5, the influence of the height of the building can be found. In case 5, the height of structure A is 20m, and the maximum value of $U/U_{\rm ref}$ (1.16) is higher than that

in case 4. Thus it may be deduced that, compared with that located near another high-rise building, a high-rise building will induce more hazard probability when it is located near a relatively low-rise building.

(E) Case 6 is the only safe one among all the cases, because there is no high-rise building in this case at all.

(F) Case 7 is designed to study the effect of the trees on pavement. Several pyramids of 6m in height serve as trees in case 7 (see Fig.7). The result of this simulation shows that the drag effect of the trees is not as obvious as might have been expected. Compared with the result of case 1, the maximum value of $U/U_{\rm ref}$ in case 7 (0.97) is just slightly lower and still exceeds the danger threshold value of $U/U_{\rm ref}$ 0.7.

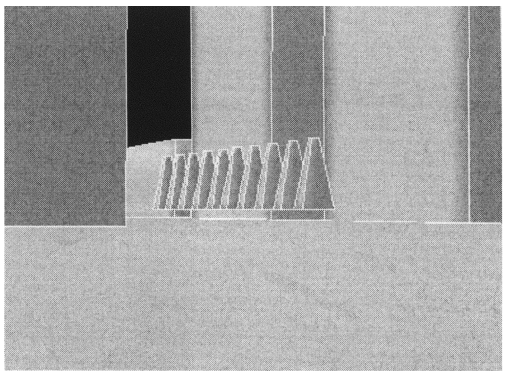


Fig.7. The model of trees on the pavement before the structures A and B.

5. Conclusions

In this paper, the pedestrian level wind safety problem induced by high-rise buildings has been studied using the commercial CFD codes Fluent.

Before detailed numerical simulations are conducted, the accuracy of Fluent is verified by using the wind tunnel data. The result of verification shows that Fluent can fairly accurately reproduce the measured data in the areas adjacent to the structure when the realizable k-ε turbulence model is adopted in the numerical simulations.

There are a total of seven cases designed for the numerical tests, and a set of safety criteria using a $U/U_{\rm ref}$ framework is adopted in the evaluation of the wind safety probability. Based on the result of the numerical simulations, several important conclusions can be drawn:

(A) The existence of high-rise buildings does increase the pedestrian level wind hazard.

(B) The wind direction, the geometry of structures and the layout of structures can obviously affect the value of $U/U_{\rm ref}$, and consequently affect the pedestrian level wind hazard probability.

(C) Trees on the pavement do not contribute much to the reduction of the pedestrian level wind hazard probability.

References

[1] Ansley R W 1980 *Build. Environ.* **24** 291

[2] Murakami S, Ooka R, Mochida A *et al* 1999 *J. Wind Eng. Indust. Aerodynam.* **81** 57

[3] Zhou X Y, Shi B C and Wang N C 2004 *Chin. Phys.* **13** 712

[4] He N Z, Wang N C, Shi B C *et al* 2004 *Chin. Phys.* **13** 40

[5] Feng G L and Dong W J 2003 *Chin. Phys.* **12** 1

[6] Feng G L, Cao H X, Dong W J *et al* 2001 *Chin. Phys.***10** 1004

[7] He J and Song C S 1999 *J. Wind Eng. Indust. Aerodynam.* **81** 295

[8] Ferreira A D, Sousa A C and Viegas D X 2002 *J. Wind Eng. Indust. Aerodynam.* **90** 305

[9] Chang C H and Meroney R N 2001 *J. Wind Eng. Indust. Aerodynam.* **89** 1325

[10] Gosman A D 1999 *J. Wind Eng. Indust. Aerodynam.* **81** 21

[11] Leitl B and Schatzmann M 1998 *Compilation of Experimental Data for Validation Purposes, CEDVAL* (Meteorology Institute, Hamburg University)

[12] Shih T H, Liou W W and Shabbir A 1995 *Comput. Fluids* **24** 227

[13] Bottema M 2000 *Build. Environ.* **35** 1