Wall function at the upwind sharp convex corners in simulating incident air flow on a cube

Y. Gao\textsuperscript{a}, W.K. Chow\textsuperscript{b}

\textsuperscript{a}Department of Building Engineering, Harbin Engineering University, Harbin, Heilongjiang, China
\textsuperscript{b}Areas of Strength: Fire Safety Engineering, Research Centre for Fire Engineering, Department of Building Services Engineering, The Hong Kong Polytechnic University, Hong Kong, China

ABSTRACT: Incident air flow towards a cube will be simulated in this paper. The standard k-\varepsilon turbulence model is used. Wall function is applied to the staggered grid at the upwind sharp convex corners by limiting the velocity component in the flow direction. It appears that the standard k-\varepsilon model is suitable for predicting air flow at the top of a cube. Both separation and reattachment around the cube can be predicted.

1 INTRODUCTION

Three-dimensional turbulent flows around a surface-mounted cube were studied extensively in the literature. One of the applications is for better understanding the air flows around buildings induced by wind. Such a surface-mounted rib or cube with sharp edges and corners can be taken as a building prototype for understanding separation and reattachment of the turbulent flow. Air flow around a surface-mounted cube is very complicated. Turbulence, separation and reattachment, bending of streamlines, high pressure gradients and swirling flows are involved. A popular engineering tool is to simulate the flow field with computational fluid dynamics (CFD), or known as computational wind engineering (CWE). However, there are many problems for flows with high Reynolds number Re. Separation at the sharp convex corners was not predicted well under the staggered grid arrangement with log-law type of wall function.

As pointed out by Murakami and coworkers\cite{1-4}, there would be problems in applying the standard k-\varepsilon model to simulate flow field around bluff-shaped bodies. Turbulent kinetic energy k will be overestimated in the impinging region. Mean flow might not be reattached as observed\cite{5}. A revised k-\varepsilon model would be better than the standard k-\varepsilon model.

In studying this problem with any CFD commercial packages, wall function is widely used. The difference in half a grid is usually not considered in assigning grids. Data concerned at the key position of the flow field might be changed, leading to wrong predictions at the upwind sharp convex corners.

In wind tunnel test, maximum peak of the turbulent kinetic energy k was found above the center of the roof. But k was relatively small around the frontal corner of the roof as observed from Figure 1a. However, k around the frontal corner was overestimated by the standard k-\varepsilon model as in Figure 1b. km\text{max} appeared at the sharp corner. The value of km\text{max} is much larger than the experimental value. In addition, separation was not obvious at the top of the cube.

The distribution of k predicted by the revised k-\varepsilon model is shown in Figure 1c. It is observed that overestimation of k was improved by the MMK model, giving better agreement with the wind tunnel experiment. The MMK model gave flow separation and reattachment on the roof. This is similar to what was observed in the wind tunnel test.
Figure 1. Comparison of k around a cube as reported by Tsuchiya et al. (1997).

2 QUESTIONS RAISED AT THE FRONTAL SHARP CORNER

The following should be further studied:
• Whether it is the problem of the standard k-ε model itself?
• Or the problem of an inappropriate boundary wall condition?

It is generally believed that the standard k-ε model has a good capability and wide applicability for representing recirculation flow. A question is:
• What are the reasons for getting seriously incorrect results at the upwind frontal sharp corners?

In the first staggered grid near the sharp corner, \( \rho u \) and \( \rho v \) intersected strongly. The local velocities cannot be determined. Three more questions will be raised:
• In this staggered grid arrangement, can wall function be applied as for parallel wall?
• How to calculate the grid size at the sharp corner?
• If the calculations are inaccurate, to what extent will the variations of the inflowing and outflowing mass and velocity in the cell be affected, and in turn affect the calculations of velocity?

3 NUMERICAL EXPERIMENTS

Numerical simulations were carried out [6,7] in a similar geometry as shown in Figure 2. The inflow conditions were the same as the experimental conditions as in references [1-5]. A computational domain same as those reported of 15.7 \( H_b \) (downstream length), 9.7 \( H_b \) (lateral width) and 5.2 \( H_b \) (vertical height) was used for the CFD simulations. The numbers of grids were 45, 38 and 57 along the x-, y- and z-directions.

Three-dimensional calculation results for upstream flow velocity \( u = y^{1/4} \) and same Re of \( 7 \times 10^4 \). In both calculations, \( u_p/u_2 = 0.025 \) \( u_o \) (bulk velocity). Wall function was used at the sharp corner.
The predicted distributions of $k$ are shown in Figure 3b, and the streamline distribution in Figure 3b. Note that the bubble length due to recirculation flow and separation at the top of the cube is $Z_R \approx 0.7 H$ as shown in Figure 3b.

![Figure 2. Geometry of the grid system.](image)

![Figure 3. With separation on the symmetry plane ($u_p$ was confined).](image)

4 DISCUSSION

Analyzing the momentum equation might give the reason why adding the south wall at the sharp corner would increase $u_p$ to $u_{pwall}$. If the convection component is much greater than the diffusion part, the following would be observed [6]:

$$u_{pwall} = \left(1 + (\rho v)_s \Delta x / \sum_i [W,E,N] a_i\right) u_p$$  \hspace{1cm} (1)

If $a_W = a_E < 0.1$ and $a_N = 0.1a_S$, then $u_{pwall} > 1.83u_p$.

At the sharp corner, $v_s$ is generally more than ten times or even up to a hundred times larger than $u_p$. If diffusion effect is neglected, the same conclusion can be drawn from the mass conservation equation at the $u_p$-cell.

If the surface $S$ of the $u_p$-cell is with $\Delta x/2$ as the wall, $(\rho v)_s \Delta x$ in equation (1) should be $(\rho v)_s \Delta x/2$. Practically, the reason for increasing $u_{pwall}$ is mainly due to the very high flow velocity $v_s$ along the $y$-direction in the $u_p$-cell at the frontal sharp corner.
Overestimation of $u_{\text{wall}}$ might be observed from:

$$u_{\text{wall}} \approx \lim_{a_s \to 0} u_p = \bar{u} + \frac{(P_p - P_e)}{\sum_{i} a_i} \tag{2}$$

When the surface $S$ is taken as a wall, $a_s = 0$. From $u_s < \bar{u}$, the mean value would become:

$$\bar{u} = \frac{\sum_{i} a_i u_i}{\sum_{i} a_i} \tag{3}$$

The smaller the value of $u_s$, the larger the value of $(\bar{u} - u_p)$. When $u_s$ is approaching zero as at the frontal sharp corner ($u_s \approx 0$), $u_p$ would be reaching $u_{p\text{max}}$. Similarly, $v_p$ will be changed to $v_{p\text{min}}$.

5 CONCLUSION

It is demonstrated that the standard k-ε model can be applied to study the incident air flow on a cube. This is achieved by specifying the associated wall functions at the upwind sharp convex corners in a proper grid system. The longitudinal velocities in the first cell adjacent to the sharp edge of the cube have to be limited. In this way, the position with maximum turbulent kinetic energy, the flow separation region, and flow reattachment region can be predicted.

The variation of pressure coefficient $C_p$ around the sharp corner can also be predicted accurately. The predicted distribution pattern of $k$ is similar to the experiment. The maximum $k$ is at the top of the rib and cube, but not at the windward frontal corner.

As observed from the above CFD results, the value and position of $k_{\text{max}}$, and the mean pressure coefficient $C_p$ are correlated with the degree of separation and reattachment length at the top of the cube.

REFERENCES