Optimization of roughness parameters for staggered arrayed cubic blocks using experimental data

T. MARUYAMA
Research Associate, Disaster Prevention Research Institute Kyoto University, Gokasyo, Ujishi, Kyoto, 611, Japan

Abstract
The roughness parameters introduced in a refined $k-\varepsilon$ turbulence model were optimized for relevant application on staggered arrayed cubic blocks. The roughness parameters were adjusted so that the values of calculated surface drag agree with those of experimental data. Variations of roughness parameters for the roughness volume density and the angle of roughness elements relative to the wind were obtained. Simulated flows using the optimized roughness parameters showed good agreement with experimental data.

1. INTRODUCTION
A refined $k-\varepsilon$ turbulence model was proposed and examined for its applicability to two- and three-dimensional simulations of turbulent boundary layer over rough surfaces [1,2]. The effect of roughness elements can be expressed in form of drag coefficient and length scale (hereafter termed roughness parameters). The values of these roughness parameters should be evaluated theoretically or experimentally. However, the theoretical approach is hard to succeed due to complications of flow around the roughness elements and the excessive amount of calculation required for direct simulation under the roughness height. Therefore, the optimization of roughness parameters was conducted by comparing simulated results with experimental data.

2. WIND TUNNEL EXPERIMENT
2.1 Measurement of surface drag
In order to compare with the calculated results, a series of wind tunnel tests were performed to investigate the variation of the surface drag caused by different patterns of staggered arrayed cubic blocks. Using the roughness shown in Figure 1 the turbulent boundary layer was generated in the wind tunnel. The roughness was located upwind and downwind of the test section as shown in Figure 2. The drag on the rough surface composed of cubic blocks was directly measured by a float built in the floor of the wind tunnel. Load sensors were set up to detect the drag $F_r$ on the float as shown in Figure 3. The drags on the roughness were measured by other three methods. Firstly, by the measurement of Reynolds
shear stress by the hot wire anemometer; secondly, by the drag on a roughness element and thirdly, by mean pressure measurements on the walls of the roughness element. The surface drag coefficients were calculated by these four different methods, results from which all fell within about 10% difference. Therefore, the direct measurement by the float was comparably reliable.

2.2 Variation of drag coefficient

The results of the drag measurement by the float are shown in Figure 4. The drag is expressed by the drag coefficient $C_d$. Here $C_d$ is defined by

$$C_d = \frac{F_d}{\frac{1}{2} \rho U_s^2 S_f}$$

where $\rho$ is the air density, $U_s$ is the free stream wind speed and $S_f$ the area of float.

The variations of the surface drag coefficients $C_d$ with the different angles of roughness element to the wind are shown in the figure against the roughness concentration ratio $\lambda$. 

---

**Figure 1.** Configuration of roughness.

**Figure 2.** Experimental arrangement.

**Figure 3.** Drag measurement of rough surface using float.

**Figure 4.** Variation of surface drag coefficients $C_d$ for staggered arrayed cubic blocks with roughness concentration ratio $\lambda$. 

---
3. NUMERICAL CALCULATIONS

3.1 Turbulence model

Here, the refined $k - \varepsilon$ turbulence model [4] is used. This can simulate incompressible turbulent flows with high Reynolds numbers over rough surfaces. The continuity and the Navier-Stokes equations are space-time-averaged and can be written as follows:

Continuity,

$$\frac{\partial G U_i}{\partial x_i} = 0$$  \hspace{1cm} (2)

Transport of momentum,

$$\frac{G}{\partial t} \frac{\partial U_i}{\partial x_j} + \frac{\partial (G U_i U_j)}{\partial x_i} = -\frac{1}{\rho} \frac{\partial G P}{\partial x_i} - \frac{\partial (G U_i U_j)}{\partial x_j} - G F_{ix}$$  \hspace{1cm} (3)

Transport of turbulent kinetic energy $K$,

$$\frac{G}{\partial t} \frac{\partial U_i K}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\nu_i}{\sigma_k} \frac{\partial G K}{\partial x_j} \right) + G \left( S - \varepsilon + F_k \right)$$  \hspace{1cm} (4)

Transport of energy dissipation rate $\varepsilon$,

$$\frac{G}{\partial t} \frac{\partial U_i \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\nu_i}{\sigma_\varepsilon} \frac{\partial G \varepsilon}{\partial x_j} \right) + G \frac{\varepsilon}{K} \left( C_{1\varepsilon} S - C_{2\varepsilon} \varepsilon + F_\varepsilon \right)$$  \hspace{1cm} (5)

Eddy viscosity $\nu_i = \frac{C_v K^2}{\varepsilon}$  \hspace{1cm} (6)

and

$$S = -\frac{U_i U_j}{G} \frac{\partial G U_i}{\partial x_j}$$  \hspace{1cm} (7)

$$U_i U_j = -\frac{\nu_i}{G} \left( \frac{\partial G U_i}{\partial x_j} + \frac{\partial G U_j}{\partial x_i} \right) + \frac{2}{3} K \delta_{ij}$$  \hspace{1cm} (8)

where $i, j = 1, 2, 3$; $x, y$ and $z$ respectively. $U_i = U$, $U_2 = V$, $U_3 = W$; $\rho$, air density; $\delta_{ij}$, the Kroneker’s delta, $= 0$ if $i \neq j$ and $= 1$ if $i = j$; $P$, pressure; $t$, time. All valuables are space-time-averaged quantities per unit fluid volume, and are non-dimensionalized. $G$ is the effective fluid volume defined as the fluid volume against unit volume $= V_i / V_0$ as shown in Figure 5. $G$ is less than 1.0 within the roughness. The influence of the roughness element is introduced into the transport equations as terms $F_{ix}$, $F_k$ and $F_\varepsilon$. These were approximated as...
Figure 5. Mesh discretization, coordinate system and arrangement of parameters.

\[ F_d = \frac{1}{2} a_n C_n |U_i| U \]  
(9)

\[ F_k = U_i F_k \]  
(10)

\[ F_e = C_p K^{1/2} \frac{\bar{\nu}_0}{L} \]  
(11)

where \( F_d \) and \( C_n \) are the \( x_i \)-directional drag caused by roughness and the drag coefficient respectively. The roughness frontal area density \( a_n \) is defined as the ratio of \( x_i \)-directional surface area to the fluid volume within the roughness. \( F_k \) and \( F_e \) are the productions of \( K \) and \( \varepsilon \) due to the roughness. \( L \) is the turbulence length scale in the roughness and here \( L \) was set to the quarter of the average value of the length surrounding each roughness element. The values of the model constants from the original model [5] were set equal to those of the standard model and hence,

\[ C_p = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3, \quad C_{1k} = 1.44, \quad C_{2k} = 1.92 \]  
(12)

This model can incorporate the effect of roughness elements and simulate the space-time-averaged velocity profiles in the layer below the roughness height, which cannot be expressed by conventional logarithmic or power law forms. In the region with no roughness elements, \( G \) is 1.0, \( F_v \), \( F_k \) and \( F_e \) are set to zero and the turbulence model becomes equivalent to the standard \( k-\varepsilon \) model.

3.2 Method of calculation

The staggered grid system in MAC method [6] was used for discretization as shown in Figure 5. Mesh discretization, the coordinate system and the arrangement of parameters are also shown in Figure. Appropriate configuration of grids were chosen to satisfy the geometry of computational domains. A non-uniform spacing mesh system is used for \( z \)-direction and the mesh distribution is concentrated near the floor and the ceiling. The mesh interval adjacent to the roughness was \( h/6 \), where \( h \) is the roughness height. We used this mesh system to satisfy the magnitude of accuracy required, the calculated values for the roughness shown in Figure 1 using a fine mesh showed little difference with that for the mesh in Figure 5 which
was twice the size. The simulated flow over the flat plane which has similar characteristics to
the experimental data was used as the inlet condition. The normal gradients of valuables at
the downstream boundary were set to zero. At the flat wall boundary such as the floor and
ceiling regions of the wind tunnel, the tangential velocity was assumed to obey a power low
(the exponent is set to 1/7) and $E = C_p^{3/4}K^{7/4}/K\Delta z$ at the first grid points adjacent to
the wall, where $K$ is the Von Karman constant ($= 0.4$) and $\Delta z$ is the distance of grid points
from the wall. The normal velocity and the normal gradient of $P, K$ and $E$ were considered as
zero at the wall. The governing equations were approximated by a finite difference scheme to
simulate the two-dimensional flow. A second-order centered difference scheme was adopted
for spatial derivatives except for convective terms of scalar quantities ($K$ and $E$), using a
first-order upwind scheme. At the surface of the roughness element, the flat wall boundary
condition was used proportionally to the fraction of the adjacent grid surface that was occupied
by the roughness. For example, assuming $G_{ik}$ (the effective fluid volume at grid $(i, k)$) is
greater than $G_{ik-1}$ as shown in Figure 6, the fraction of the boundary AB occupied by the
surface of roughness element is $G_{ik}-G_{ik-1}$ and the flat wall boundary condition is used
proportionally to $(G_{ik}-G_{ik-1})/G_{ik}$ in order to approximate the $z$-directional derivatives at
grid point $(i, k)$. The Adams-Bashforth scheme was employed for time-marching. Numerical
integrations were conducted according to the SMAC method [7]. The values of roughness
parameters are assumed to be constant in the roughness as long as the roughness configuration
does not change.

![Figure 6. Fraction of the boundary occupied by the surface of the roughness element at the
boundary AB, between grids $(i, k)$ and $(i, k-1)$.](image)

4. **OPTIMIZATION OF ROUGHNESS PARAMETERS**

The values of $G, a_i$, and $L$ in the governing equations can be obtained when the configuration
of roughness is defined. On the other hand, the roughness parameters, i.e. the drag coefficients
$C_d$ and the ratio of turbulence scale $C_{\theta}$, are model coefficients, and these vary depending
on the configuration of roughness. The values of these roughness parameters should be
evaluated for each roughness. In order to compare the simulated results with the experimental
data, a number of two-dimensional calculations were carried out. The computational domain
was created from the floor to the ceiling of the wind tunnel as shown in Figure 5.

From comparison of the two-dimensional calculations with experimental data, the values
of roughness parameters were optimized to fit the calculated results with the experiments for
Figure 7. Variation of roughness parameters with roughness volume density $\rho_r (= 1 - G)$.

Figure 8. Comparison of two-dimensional calculation values with experimental data at various positions: experimental data $\bigcirc$, $U_0\square$, $K$; calculated values $-$; $U_0$ is freestream velocity; $\alpha = 0^\circ$, roughness height $h = 30\text{mm}$. 
various roughness configurations. The variations of roughness parameters with the roughness volume density \( \rho_r = 1 - \frac{1}{C} \) were obtained up to \( \rho_r = 0.33 \) at \( \alpha = 0^\circ \) and up to \( \rho_r = 0.45 \) at \( \alpha = 45^\circ \) as shown in Figure 7. Some examples of simulated results using those optimized values for three different roughnesses show good agreements with experimental data as shown in Figure 8.

5. CONCLUSION

Direct measurement of the surface drag was performed on staggered arrayed cubic blocks using the float. The values of the roughness parameters introduced in a refined \( k-\epsilon \) turbulence model were evaluated by comparison of two-dimensional calculations with experimental data. Calculated results showed good correlation with experimental data, indicating the practical relevance of the roughness parameters obtained for flow simulations over staggered arrayed cubic blocks.

Acknowledgments

The author is grateful to Mr. H. Hiraoka (assistant of Kyoto University, M. Eng.) for his advice and kind guidance for the development of programs.

References