Wind and Structures, Vol. 5, No. 2~4 (2002) 369-378
DOI: http://dx.doi.org/10.12989/was.2002.5.2_4.369

Large eddy simulation using a curvilinear coordinate system for the flow around a square cylinder

Yoshiyuki Ono†

Technical Research Institute Obayashi Corporation 4-640, Shimokiyoto, Kiyose, Tokyo 204-0011, Japan

Tetsuro Tamura‡

Tokyo Institute of Technology 4259, Nagatsuda, Midori-ku, Yokohama 226-8502, Japan

Abstruct. The application of Large Eddy Simulation (LES) in a curvilinear coordinate system to the flow around a square cylinder is presented. In order to obtain sufficient resolution near the side of the cylinder, we use an *O*-type grid. Even with a curvilinear coordinate system, it is difficult to avoid the numerical oscillation arising in high-Reynolds-number flows past a bluff body, without using an extremely fine grid used. An upwind scheme has the effect of removing the numerical oscillations, but, it is accompanied by numerical dissipation that is a kind of an additional sub-grid scale effect. Firstly, we investigate the effect of numerical dissipation on the computational results in a case where turbulent dissipation is removed in order to clarify the differences between the effect of numerical dissipation. Next, the applicability and the limitations of the present method, which combine the dynamic SGS model with acceptable numerical dissipation, are discussed.

Key words: Large-eddy-simulation; square cylinder; dynamic SGS model; numerical dissipation

1. Introduction

Large Eddy Simulation (LES) can deal with the unsteady features of turbulent flows directly, while only smaller scale fluctuations than the computational grid size are modeled. For many years, LES has been mainly applied to relatively simple flows. However, the state of this research has not reached an understanding about the applicability of LES to the complex flows such as the turbulent flow past a bluff body.

In workshops organized by Rodi and Ferziger (1995, Tegernsee) (Rodi 1998), the flows around a square cylinder at $Re = 22{,}000$ were given as a computational model for the test of LES from an engineering point of view. However, according to the report (Rodi 1998, Rodi *et al*. 1997) of this workshop, none of the calculation results by participants were completely satisfactory. Some of the reasons for the lack of agreement with experimental data are as follows; 1) The resolution near the side of the cylinder was not enough to accurately capture the development and transition of the separated shear layer. Participants in Tegernsee used Cartesian coordinate system, but it is not easy to use a sufficiently fine grid in this system. 2) LES using upwind-methods tended to predict the

† Research Engineer
‡ Professor

narrow recirculation zones. Without the upwind scheme, numerical oscillation cannot be avoided from the front corner of the square cylinder for reasonable grid resolutions. In this case, it becomes difficult to estimate the unsteady characteristics of pressure acting on the cylinder. Hence, it is necessary to examine the effect of the upwind schemes on a solution.

The objective of this paper is to investigate the applicability of LES using curvilinear coordinate system to the flow around a square cylinder. In order to get sufficient resolution near the side of the cylinder, we use an *O*-type grid in which one set of coordinate lines encircle the cylinder. Even in the case of using a curvilinear coordinate system, it is difficult to avoid numerical oscillation arising in high-Reynolds-number flows past a bluff body, without using an extremely fine grid. The upwind scheme has an effect to remove the numerical oscillations, however, it is accompanied by numerical dissipation that is a kind of additional sub-grid scale effect. Here, we study the accuracy of computational wake structures and aerodynamic quantities predicted by the present method, using a fine grid that resolves the near-wall flow, through a comparison with those in Tegernsee. First, we investigate the effect of numerical dissipation on the computational results in a case where turbulent dissipation is removed in order to clarify the differences between the effect of numerical and turbulent dissipation. Next, the applicability and the limitations of the present method, which combines the dynamic SGS model with acceptable numerical dissipation, are discussed.

2. Problem formulation

The governing equations are given by the following Navier-Stokes and continuity equations :

$$\frac{\partial u_i}{\partial t} = -\frac{\partial}{\partial x_j}u_i u_j + \frac{1}{Re}\frac{\partial^2 u_i}{\partial x_j \partial x_j} - \frac{\partial p}{\partial x_i}, \qquad (i = 1, 2, 3) \tag{1}$$

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{2}$$

where u_i ($i = 1, 2, 3$), p, t and Re denote the velocity , pressure, time and the Reynolds number.

To advance the solutions of velocities and pressure in time, a fractional step method is employed. The time integral of the momentum equation is hybrid; that is to say the Crank-Nicolson scheme is applied to the viscous terms and an explicit third-order Runge-Kutta method is used for convective terms. In this simulation, the original governing equations are transformed to a curvilinear coordinate system. The present scheme can be written as;

$$\frac{J\tilde{u}_i^k - Ju_i^{k-1}}{\Delta t} = -\gamma_k N(u_i^{k-1}) - \delta_k N(u_i^{k-2}) + \alpha_k L(\tilde{u}_i^k + u_i^{k-1}) \tag{3}$$

$$\frac{Ju_i^k - J\tilde{u}_i^k}{\Delta t} = -\frac{\partial}{\partial \xi^m}\frac{\partial \xi^m}{\partial x_i}\phi^k \qquad (k = 1, 2, 3) \tag{4}$$

$$\frac{1}{J}\left(\frac{\partial (JU^i)^k}{\partial \xi_i}\right) = 0, \qquad U^i = \frac{\partial \xi^i}{\partial x^m}u_m \tag{5}$$

where $k = 1, 2, 3$ denotes the sub-step number for Runge-Kutta method, and u_i^0 and u_i^3 are velocities at time step n and $n+1$. The coefficients x^m and J denote the metrics and Jacobian of the transformation, respectively. $L(u_i)$, $N(u_i)$ represent finite difference approximations to the viscous and convective terms :

$$L(u_j) = \frac{\partial}{\partial \xi^m}\left(\left(\frac{1}{Re} + \nu_{SGS}\right) J \frac{\partial \xi^m}{\partial x_j} S_{ij}\right) \tag{6}$$

$$N(u_i) = \partial \frac{\partial}{\xi^m}(JU^m) u_i \tag{7}$$

where

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial \xi^m}\frac{\partial \xi^m}{\partial x_j} + \frac{\partial u_j}{\partial \xi^n}\frac{\partial \xi^n}{\partial x_i}\right) \tag{8}$$

$$\nu_{SGS} = 2CJ^{2/3}|S| \tag{9}$$

$$|S| = \sqrt{2S_{ij}S_{ij}} \tag{10}$$

The parameters in Eq. (3) are given by

$$\alpha_1 = \frac{4}{15},\quad \alpha_2 = \frac{1}{15},\quad \alpha_3 = \frac{1}{6}$$

$$\gamma_1 = \frac{8}{15},\quad \gamma_2 = \frac{5}{12},\quad \gamma_3 = \frac{3}{4}$$

$$\delta_1 = 0,\quad \delta_2 = -\frac{17}{60},\quad \delta_3 = -\frac{5}{12} \tag{11}$$

To obtain solutions that satisfy the conservation laws, a finite volume method is applied in a collocated grid system. Spatial derivatives of variables are treated as a second-order central difference. Convective terms are approximated using the higher order interpolation-method (Kajisima 1993). To avoid the numerical instability near the front corners of a square cylinder, the numerical dissipation is added to the convective terms. Namely, the convective terms are approximated as follows.

$$\frac{\partial}{\partial \xi} JU_1 u_i = \delta_\xi (JU_1 \bar{u}_i^\xi) + \alpha J|U_1| \frac{u_{i-2} - 4u_{i-1} + 6u_i - 4u_{i+1} + u_{i+2}}{12} \tag{12}$$

$$\bar{u}_i^\xi = \frac{-u_{i-2/3} + 9u_{i-1/2} + 9u_{i+1/2} - u_{i+3/2}}{16} \tag{13}$$

$$\delta_\xi = \frac{f_{i-3/2} - 27f_{i-1/2} + 27f_{i+1/2} - f_{i+3/2}}{24} \tag{14}$$

We use the dynamic SGS model (Lilly 1992). The unknown parameter C is computed by using the method of Jordan & Ragab (1998) as follows.

$$CJ^{2/3} = -\frac{L_i^k M_i^k}{2M_i^k M_i^k} \tag{15}$$

$$L_i^k = \overline{\bar{u}_i \bar{U}}^k - \bar{u}_i \overline{\bar{U}}^k \tag{16}$$

$$M_i^k = \gamma^2 \overline{|\bar{S}|\bar{S}_i}^k - \overline{|\bar{S}|\bar{S}}_i^k \tag{17}$$

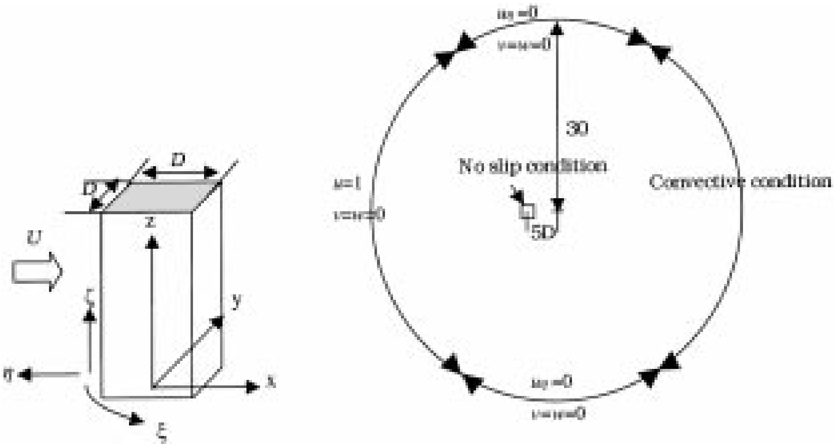

Fig. 1-1 Computational domain and boundary condition in present simulation

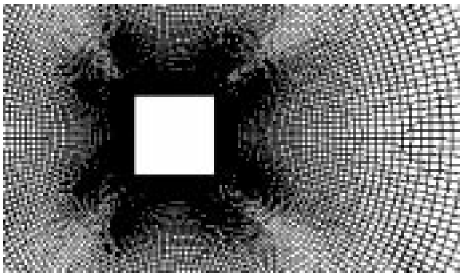

Fig. 1-2 Computational grid near the cylinder

$$\overline{U^k} = J\frac{\partial \xi_k}{\partial x_j}\bar{u}_j \tag{18}$$

$$\bar{S}_i^k = J\frac{\partial \xi_k}{\partial x_j}\bar{u}_j\bar{S}_{ij} \tag{19}$$

A filter width ratio of $\gamma = 2$ is chosen. The tensors L_i^k and M_i^k are determined through application of a box filter. In order to avoid numerical instabilities, negative values of C are truncated to zero.

As shown in Fig. 1-1, the computational region is a circle with radius of 30D (D = cylinder dimension). Fig. 1-2 shows the computational mesh (200×130) near the cylinder. The resolution close to the cylinder is made much finer ($0.1D/(Re)^{1/2}$) than that of Tegernsee, where the smallest grid size is 0.01D. We use 30 grid points over 2D length in the span-wise direction.

Standard inflow conditions, $u = 1$, $v = w = 0$, are imposed at the upstream boundary and a convective condition is used at downstream. A no-slip condition is used at the cylinder surface. The sampling time for statistics is 200 dimensionless time, which is about 25 periods of vortex shedding.

3. Computational results

We discuss the accuracy of the computational wake structures and aerodynamics quantities predicted by the present method, using a fine grid that resolves the near-wall flow, through a comparison with those in Tegernsee.

3.1. The effect of numerical dissipation on computational results

We investigate the effect of numerical dissipation on computational results. Four factors of numerical dissipation ($\alpha = 0.2$, 0.5, 1, 3) are used without any SGS models. The Reynolds number is 22,000, equal to that of the experiment by Lyn (1989).

Table 1 summarises various aerodynamic quantities such as the dimensionless shedding frequency (Strouhal number $St = fD/U$), time-averaged drag coefficient C_{Dave}, the RMS values of the fluctuations of drag and lift coefficients C_{Drms} and C_{Lrms}.

The values in all cases are approximately in agreement with the experimental data (Okajima 1983, Lyn 1989), though the simulation of $\alpha = 1$ has a slightly different values of St and C_{Lrms} from the others. Concerning the drag coefficient, the calculations in Tegernsee using a no-slip condition in a coarse grid tend to produce the higher values, while the present calculations using a finer grid with curvilinear coordinates show better agreement with the experiments (McLean and Gartshore 1992, Lee 1988). Concerning the effects of the numerical dissipation factor on the computational results, the calculations including large numerical dissipation tend to produce higher values of C_{Dave} and C_{Drms} .

Table 1 Aerodynamic coefficients(Non-SGS-Model)

α	St	C_{Dave}	C_{Drms}	C_{Lrms}
0.2	0.127	1.91	0.22	1.21
0.5	0.128	1.90	0.28	1.19
1	0.122	2.16	0.30	1.50
3	0.125	2.17	0.33	1.23
Exp.[6]~[9]	0.125~0.132	1.9~2.1	0.16~0.23	0.7~1.4

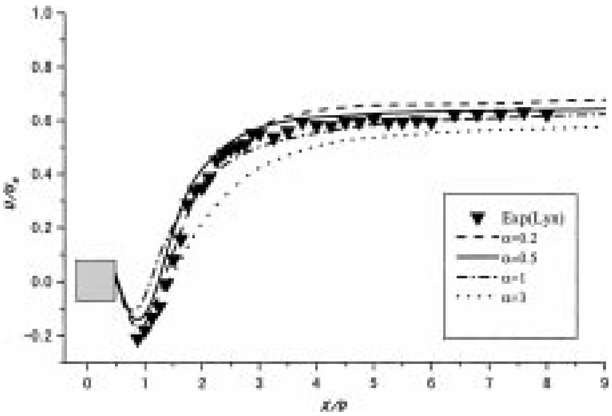

Fig. 2 The mean velocity on the center plane of the cylinder

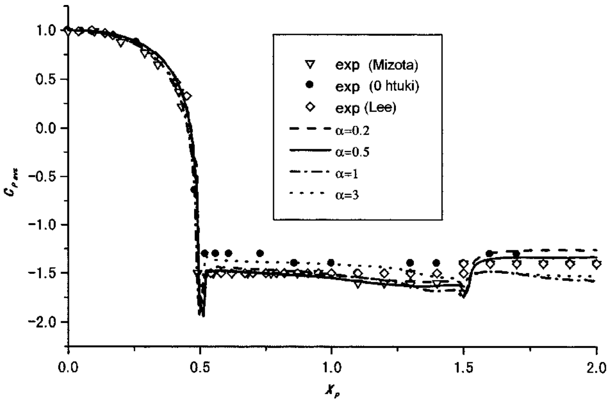

Fig. 3 The averaged pressure distribution on the cylinder

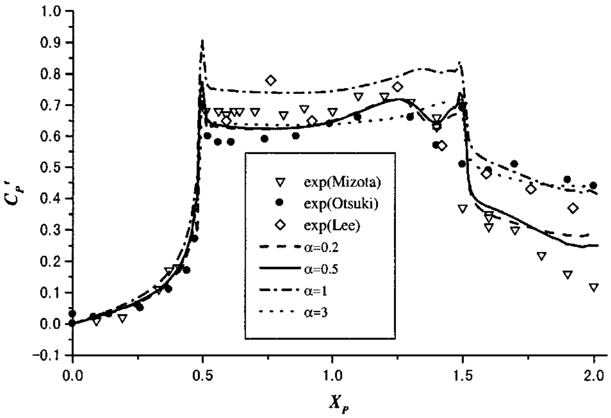

Fig. 4 The fluctuating pressure coefficient on the cylinder

The results for the mean velocity on the center plane of the cylinder are shown in Fig. 2. The experimental velocity (Lyn 1989) recovers very slowly in the downstream region and nearly levels off at about 0.6 of the upstream free stream level. Most results in Tegernsee show stronger recovery of the velocity than the experiments. On the other hand, the present simulations show close values the experimental ones, except in the case of $\alpha = 3$. Looking at the near cylinder-region, the calculations of the small dissipation factor tend to produce similar values for downstream recirculation length and maximum negative stream-wise velocity.

In Fig. 3, we present the averaged pressure distribution around the cylinder, including the experimental results (Lee 1988, Ohtsuki 1978, Mizota *et al.* 1988). The computational results are in reasonably good agreement with the experimental data, though not with Tegernsee. Looking at the downstream face in detail, the simulation of $\alpha = 3$ has little recovery of C_{pave} .

As shown in Fig. 4, the fluctuating pressure coefficient is qualitatively well predicted, but there are significant quantitative differences among the results, compared with the experimental data (Ohtsuki 1978, Mizota *et al.* 1988). The calculations of $\alpha = 0.2$ and 0.5, having the same tendency, can predict the fluctuation curve near the leeward corners of the cylinder, while the cases of $\alpha = 1$ and 3 cannot.

Consequently, the calculation using a curvilinear coordinate system to get finer resolution, gives good results compared with those in Tegernsee. However, the calculations using large numerical dissipation factors tend to show lack of agreement with the experiments.

3.2. A applicability and limitations of the Dynamic SGS Model

Next, we introduce the dynamic SGS model (DSGSM) to a method which has an acceptable numerical dissipation ($\alpha = 0.2$, 0.5), and discuss the applicability and the limitations of this method to the flow around a square cylinder.

Table 2 Aerodynamic coefficients (Dynamic SGS model)

α	St	C_{Dave}	C_{Drms}	C_{Lrms}
0.2	0.127	2.04	0.18	1.43
0.5	0.125	1.93	0.23	1.37
Exp. [6]~[9]	0.125~0.132	1.9~2.1	0.16~0.23	0.7~1.4

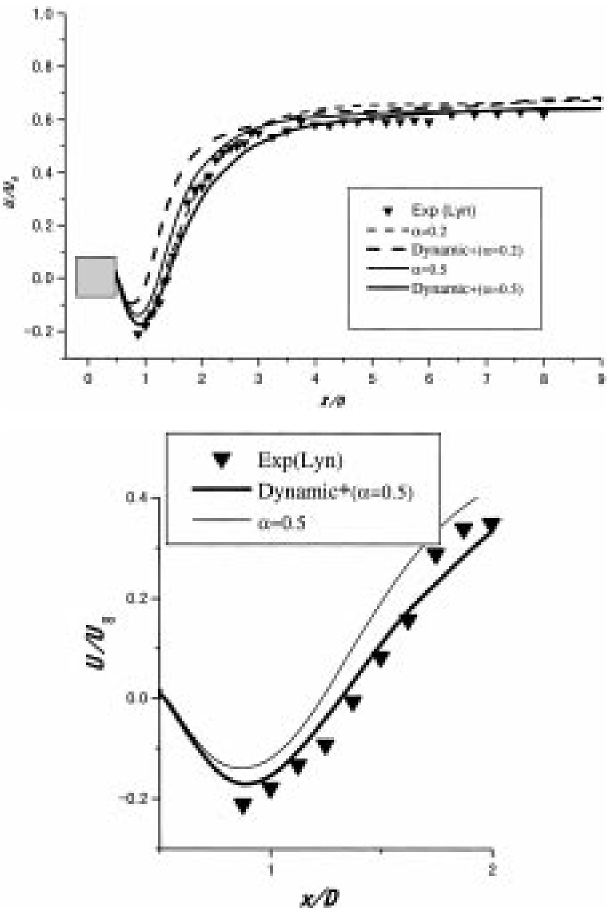

Fig. 5 The mean velocity on the center plane of the cylinder

The aerodynamic coefficients are presented in Table 2. The calculations by DSGSM predict higher values of C_{Dave} and C_{Lrms} than the previous non-SGS model (Table 1), but they are within the limits of the experimental data.

Fig. 5 shows the results for the mean velocity on the center plane of the cylinder. There are not large differences between the velocities in the far wake predicted by DSGSM and those without model. Looking at the near-cylinder region, it can be seen that the calculation of $\alpha = 0.5$ with DSGSM exhibits longer recirculation length and lower maximum negative stream-wise velocity value than those without the SGS Model. As a result, the calculation of $\alpha = 0.5$ with DSGSM show results closer to the experimental data (Lyn 1989). On the other hand, the results of $\alpha = 0.2$ with DSGSM is in disagreement with the experimental data, though the calculation without the SGS model shows better agreement.

Fig. 6, Fig. 7 show the averaged pressure distribution and the RMS value of the fluctuating pressure coefficient around the cylinder. The results of $\alpha = 0.5$ with DSGSM are in good agreement with experimental ones (Lee 1988, Ohtsuki 1978, Mizota *et al.* 1988) as well as those without

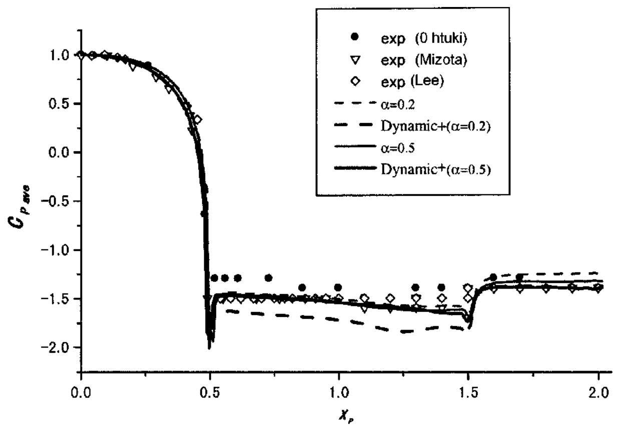

Fig. 6 The averaged pressure distribution on the cylinder

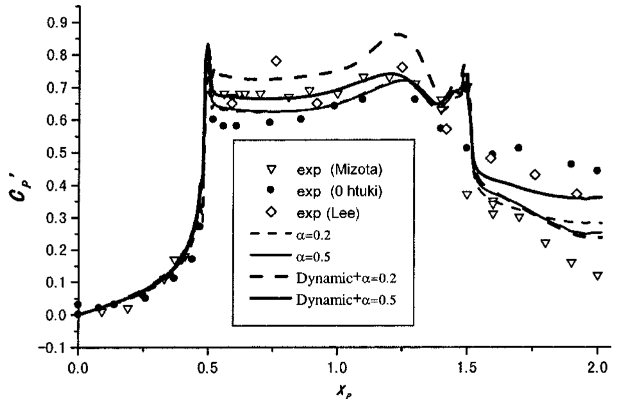

Fig. 7 The fluctuating pressure coefficient on the cylinder

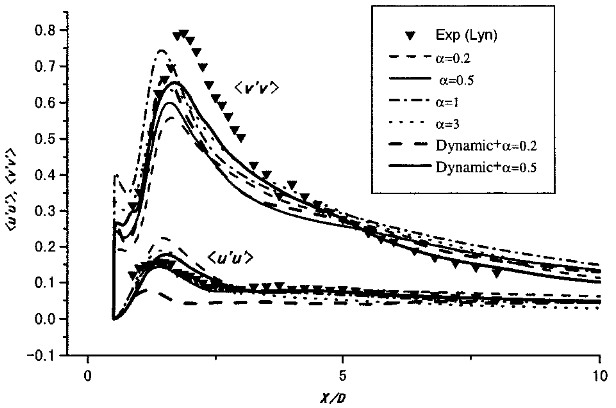

Fig. 8 The distribution of the total fluctuation components, $<u'u'>$ and $<v'v'>$ along the centre-line

model, while the simulation of $\alpha = 0.2$ with DSGSM has a lower value of C_{pave} on the side surface of the cylinder than other cases. One reason for the loss of accuracy might be that the effect of the SGS model is disturbed by the numerical oscillations occurring from the front corner of the square cylinder.

Fig. 8 shows the distribution of the total fluctuation components, $<u'u'>$ and $<v'v'>$, the along the centre-line in comparison with the experimental results (Lyn 1989). Concerning $<u'u'>$, except the case of $\alpha = 0.2$ with and without DSGSM, the results of the calculations agree with each other and with the experimental ones. However, concerning $<v'v'>$, none of the simulations is satisfactory, though the results of DSGSM tend to be closer to the experimental data than those without model. In these points, the present calculations do not show much improvement from those of Tegernsee.

4. Conclusions

In order to get a fine resolution enough to accurately capture the development and transition of the separated shear layer, a dynamic sub-grid scale model, using a curvilinear coordinate system, was applied to the flow around a square cylinder. We investigated the effect of numerical dissipation on the computational results in a case where the turbulent dissipation was removed in order to clarify the differences between the effect of numerical and turbulent dissipation. As a result, the calculation gave results closer to the experimental data than those of Tegernsee. Especially, the result of DSGSM with very small numerical dissipation, which can remove the numerical oscillation, is the closest. However, prediction of the fluctuation components was not improved.

Reference

Jordan, S.A. and Ragab, S.A. (1998), " Large eddy simulation of the near wake of a circular cylinder", *J. Fluids Eng.*, **120**, 243-252.

Kajisima, T. (1993), "Higher-order finite difference methods for wall-bounded incompressible flows", *Proc 5th Int. Symp. Comp. Fluid*, 414-419.

Lilly, D.K. (1992), "A proposed modification of the germano subgrid-scale closure method", *Phys. Fluids*, A.4, 633-635.

Lyn, D.A. (1989), "Phase-averaged turbulence measurements in the separated shear flow around square cylinder", *Proc. 23rd. Cong. Int. Ass. Hydraulic Research*, Ottawa, Canada, A85-A92.

Lee, B.E. (1988), "Some observation of the effect of aspect ratio on the influence of turbulence on the drag of rectangular cylinders", *The International Colloquium on Bluff Body Aerodynamics and its Applications*, Kyoto.

McLean, I. and Gartshore, I. (1992), "Spanwise correlations of pressure on a rigid square cylinder", *J. Wind Eng. Ind. Aerod.* **41**, 779.

Mizota, T., *et al*. (1988), "Aerodynamic characteristics of fundamental structures (Part 1)", *J. of Wind Eng.*, **36**, 49 (in Japanese).

Okajima, A. (1983), "Flow around a rectangular cylinder with a section of various width/height ratios", *J. of Wind Eng*., **17**, 1-13 (in Japanese).

Ohtsuki, Y. (1978), "Wind tunnel experiments on aerodynamic forces and pressure distributions of rectangular cylinders in a uniform flow", *Proc. 5th Symp. on Wind Effects on Structures*, Tokyo Japan.

Rodi. W. (1998), "Large-eddy simulations of the flow past bluff bodies: state-of-the Art", *JSME Int. J.*, **41**, 361-373.

Rodi, W., Ferziger, J.H., Breuer, M. and Pourquie, M. (1997), "Status of large-eddy simulation; results of a workshop", *J. Fluids Eng*., **119**, 248-262.