KSME International Journal, Vol. 11, No. 3, pp. 339~347, 1997

Spatial Resolution Effects on Unsteady Turbulent Flow Computations for the Rectangular Cylinders by k-ε Turbulence Models

Sangsan Lee*

(Received July 3, 1996)

In predicting unsteady turbulent flows around a square cylinder using k-ε turbulence models, choice of right turbulence models was found to be critical. If a proper care is taken to choose a convection scheme and near-wall resolution, the conventional turbulence models may predict an unsteady turbulent flow at low Reynolds numbers with reasonable accuracy. A systematic computation is carried out to identify the effects of the aspect ratio of a rectangular cylinder and of the flow Reynold number on the spatial resolution requirement. It is found in general that the grid resolution requirement is more stringent for a cylinder with a smaller aspect ratio. By investigating high Reynolds number computations, the grid refinement in terms of viscous wall units is found unimportant in accurately predicting the unsteady aerodynamic forces on the cylinder. Instead, resolution of shear layers formed at the forward separation corners is found to be more critical.

Key Words : Unsteady Turbulent Flow, Vortex Shedding, Force Coefficients, Strouhal Number, Aspect Ratio, k-ε Turbulence Model.

1. Introduction

As the construction technology advances, civil structures are made larger not only for reducing construction cost but also for their beautiful appearances. The large-scale civil structures, such as long-span bridges and tall buildings, have relatively lower structural stiffness and damping, which are susceptible to the dynamic forcing. In the structure design process, therefore, it is critically important to consider the structure's dynamic stability due to the unsteady aerodynamic forces. The unsteady aerodynamic forces on a bluff body are generated mainly by an alternate flow separation with vortices shed downstream of the bluff body. Predicting such flow features is a pending issue in computational wind engineering. Most computational investigations are made for a geometrically simple case with a square cylinder(Franke & Rodi, 1991; Murakami, 1993; Kato & Launder, 1993), where mostly k-ε turbulence models are tried since they are most frequently used in engineering computation of turbulent flows. Even with the popularity of k-ε models, they are known to have some intrinsic deficiencies by neglecting rotation, anisotropy, and non-equilibrium effects. Extensive review on the collaborative testing of conventional turbulence models is given by Bradshaw et al.(1994).

The main source of errors in the computational fluid dynamics(CFD) predictions of bluff body flows is the poor modeling of complex flow characteristics, especially of turbulence. It had been known to be impossible to accurately predict the unsteady turbulent flows withe the conventional k-ε turbulence models, until Lee(1996) reported reasonable predictions with proper unmerical schemes and parameters. The work is also

* System Engineering Research Institute, P.O. Box 1, Yuseong, Daejeon 305-600
TEL: +82-42-869-0560, FAX: +82-42-869-0599, e-mail: sslee@seri.re.kr

consistent with Durbin (1995) in that numerical aspects are as critical as the turbulence model choice in predicting separated turbulent flows: The average wake size behind a triangular cylinder can be reasonably predicted only by an unsteaay computation.

This paper investigates the spatial resolution requirement for an accurate computation of turbulent flows over rectangular cylinders at different Reynolds numbers: Results from computations with four different aspect ratios at two Reynolds numbers are analyzed to investigate the requirement. In the following section, the problem is formulated with a brief description of the governing equations. Selection of numerical schemes and parameters is justified by the results of Lee (1996). In Sec. 3, sensitivities of unsteady wind loading prediction to the aspect ratio and to the Reynolds number are systematically investigated. The final section summarizes the results with implications of the present investigation on the availability of CFD in wind engineering.

2. Problem Formulation

The governing equations solved to predict unsteady turbulent flows in the previous work (Lee, 1996) and in the present study, are the conservation of mass and momentum, expressed as

$$\frac{\partial u_j}{\partial x_j}=0$$

$$\frac{\partial u_i}{\partial t}+\frac{\partial u_j u_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[(\nu+\nu_T)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]$$

where all the variables are ensemble averaged quantities and the repeated indices denote the summation over the streamwise and normal directions. The eddy viscosity ν_T is evaluated by turbulence kinetic energy (k) and its dissipation rate (ε) as $\nu_T=c_\mu f_\mu k^2/\varepsilon$, where c_μ is a model-dependent constant and f_μ is a parameter activated near the wall only for the low Reynolds number k-ε turbulence model. Turbulence quantities, k and ε, are governed by additional transport equations.

A systematic investigation had been made on the prediction of an unsteady turbulent flow over a square cylinder ($B/D=1$) at $\mathrm{Re}=2.2\times10^4$ with $\mathrm{Re}=UD/\nu$ (D: cylinder height, B: cylinder width, U: uniform inflow speed), where extensive experimental results (Lyn & Rodi, 1994) and large-eddy simulation (LES) results (Murakami & Mochida, 1995) are reported. Boundaries of the computational domain are away from the cylinder at least by $5D$ upstream, $10D$ downstream, and $6.5D$ in lateral directions. Uniform streamwise velocity is prescribed at the inlet with 2% turbulence intensity, and the convective boundary condition is used at the exit boundary. To simulate the experimental setup with minimal computational cost, symmetry boundary condition is imposed at the wind tunnel walls. Effects of the symmetry boundary condition instead of wall boundary condition are found negligible. Near the cylinder surface, wall function is used as a bundary condition except for the low Reynolds number model, where the governing equations are integrated to the solid surface. The governing euqations are approximated through a finite-volume approach with all the flow variables evaluated at the cell center. Second-order central differences are used to approximated derivative terms except for convection terms. Mass conservation is enforced by solving the SIMPLE-type pressure correction equation. Time advancement is carried out through an implicit Euler scheme, and the normalized time step, or a CFL number estimate, $U\Delta t/(\Delta x)_{\min}$ is taken as around 0.7, where $(\Delta x)_{\min}$ is the minimun grid spacing. The local CFL number exceeds 2 near the corner of the cylinder. It takes about 1 CPU sec for the reference simulation (case RSL in Table 1) to advance one time step in CRAY Y-MP C90 with the absolute cell mass flux residual sum to be less than 10^{-4} of the inflow mass flux at each time step.

The computational results are summarized in Table 1 with an emphasis on the prediction of aerodynamic forces on the body, where f_D and f_L are the ensemble averaged drag and lift force per unit depth of the cylinder and $\mathrm{St}=fD/U$ is the Strouhal number with f being the Stouhal fre-

Table 1 Summary of computational result (Lee, 1996) for a square cylinder at Re=2.2×10^4.

Case	Scheme	Model	c_D	c_L	St
RSL	QUICK	RNG	2.22±0.38	±2.35	0.133
SSL	QUICK	Standard	1.76±.003	±0.67	0.141
LSL	QUICK	Low Re	2.10±0.04	±1.61	0.134
hSL	Hybrid	RNG	1.90±0.05	±0.77	0.134
uSL	Upwind	RNG	1.83±0.02	±0.55	0.130
Exp*			2.14±0.09		0.134
LES+	2nd central	SGS	2.09±0.13	±1.60	0.132

Definition : $c_D=f_D/KD$, $c_L=f_L/KB$ with $K=\rho U^2/2$.

* Lyn & Rodi (1994)

+Murakami & Mochida (1995)

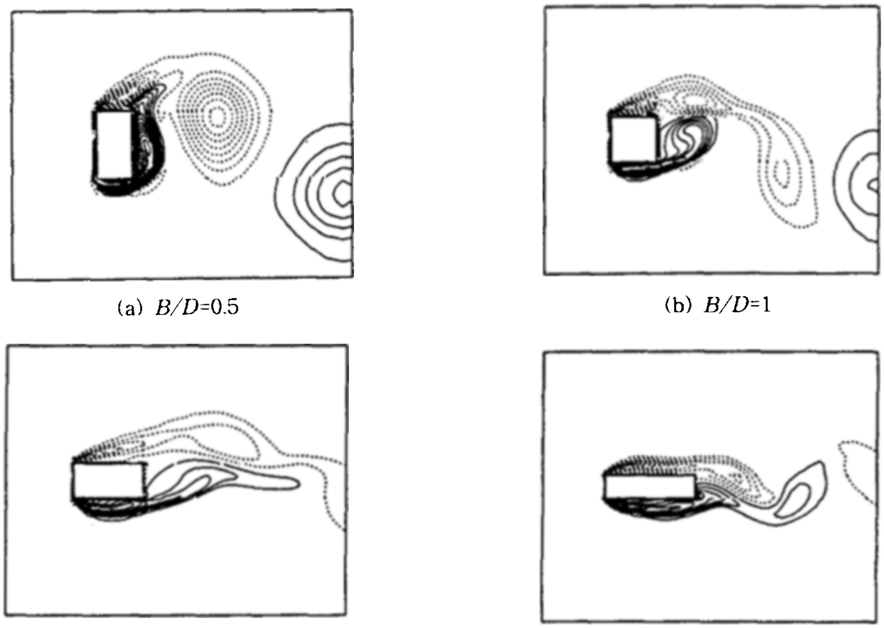
quency based on the lift force fluctuation. In evaluating the drag and lift forces, viscous stress is also included in addition to pressure. However, the viscous stress contributes less than a percent to the total force. The mean and the ensemble-averaged fluctuating amplitude for a square cylinder are given in the Table 1. Out of conventional turbulence models investigated, the standard k-ε model (Launder & Spalding, 1974) was unable to reasonably predict the unsteady forces. Performance of the RNG k-ε model (Yakhot et al., 1992) and the low Reynolds number k-ε model (Launder & Sharma, 1974), on the other hand, was reasonable in predicting the unsteady wind forces, where the near-wall grid spacing required for the latter are about *30 times finer* than that for the former. The RNG k-ε model, however, tens to overkill the eddy viscosity of the standard k-ε model, which leads to another extreme predictions of underestimating the wake size leading to overstimated unsteady forces in the present case, while the low Reynolds number k-ε model predict turbulence statistics to an extent comparable to LES (Murakami & Mochida, 1995). Detailed comparison of turbulence statistics is reported in another paper (Lee, 1996).

With lower order convection schemes of hybrid or upwind schemes, unsteady force coefficients are underpredicted, while those with a QUICK scheme are in better agreements with the experiment and the LES results. In general, a case with more dissipative scheme predicts smaller fluctuating force coefficients. In the computations referred, grid in the wall vicinity was required as fine as $(\Delta x_i^+)_{min}\simeq 20$ in the RNG model and $(\Delta x_i^+)_{min}\simeq 1$ in the low Reynolds number model, which are prohibitively stringent for flows around a civil structure with Re=$O(10^7)$ or higher.

In the present paper, the spatial resolution requirement is investigated for a computation of turbulent flows over a rectangular cylinder at different Reynolds numbers with the QUICK scheme for the convection term. The economical RNG k-ε turbulence model is used in all computations, since the computations with more accurated low Reynolds number k-ε model are prohibitively costly for high Reynolds number flows. Computations are conducted at two different Reynolds numbers, Re=2.2×10^4 and 10^6. In order to investigate the effects of spatial resolution on the solution accuracy at various aspect ratios, computations with different aspect rations (B/D=0.5, 1, 2 and 4) are carried out for comparison.

3. Results and Discussion

Typical instantaneous vorticity fields from the computations at various aspect ratios are shown in Fig. 1. Clear asymmetries in the flow field may be observed, which are also changing in time with a regular vortex street downstream of each cylin-



(a) B/D=0.5 (b) B/D=1 (c) B/D=2 (d) B/D=4

Fig. 1 Instantaneous vorticity fields at the minimum lift phase of the oscillation for case (a) RHL, (b) RSL, (c) R2L and (d) R4L. Solid lines denote for positive, and dashed lines for negative vorticity ($\Delta\omega=1.0$)

der. The time-dependent flow asymmetry results in unsteady aerodynamic forces on the body with finite amplitudes. A reference grid system used in the computation for $B/D=2$ (case R2L in Table 3) is with nonuniform 100×55 cartesian meshes. Mesh is coarsened in the downstream wake region away from the body, since the unsteady aerodynamic forces on the cylinder were found to be little affected by the wake resolution (Lee, 1996).

3.1 Effects of the aspect ration, B/D

The evolution of the drag and lift coefficients for case R2L (in Table 3) are shown in Fig. 2. It shows an initial transient followed by regular oscillations with a low frequency modulation. Time history of lift coefficient after an initial transient is taken for the Strouhal frequency estimation through FFT with a proper windowing (Press et al., 1986) to remove signal non-periodicity effects.

Not likely in the square cylinder case, no thorough experiment is found in the literature on the unsteady turbulent flow over the rectangular cylinder except that of Okajima (1982), where only the Strouhal number behavior for flows with $70\leq Re\leq2.0\times10^4$ is investigated. Since the available data is restricted at the relatively low Reynolds numbers, the simulation is validated by comparing the simulation results at $Re=2.2\times10^4$ with the Strouhal number available at $Re=2.0\times10^4$ in the experiment, which is given in Table 2.

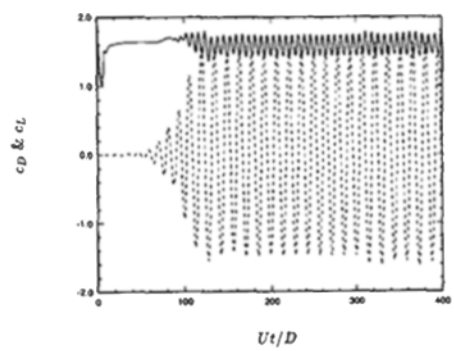


Fig. 2 Evolution of force coefficients in case R2L ($B/D=2$, $Re=2.2.\times10^4$) : —— c_D, ---- c_L

Table 2 Comparison of Strouhal numbers for flows over rectangular cylinders

B/D	0.5	1.0	2.0	3.0	4.0	Re
Present	0.133	0.135	0.067	—	0.131	2.2×10^4
Exp*	—	0.134	0.085	0.161	0.130	2.0×10^4

* Okajima (1982)

In the following, investigated are the effects of the near-wall grid spacing on the force coefficients: their mean and fluctuation amplitudes along with their Strouhal numbers. Sensitivity of those effects at different aspect ratios is investigated by comparing the results with $B/D=0.5$, 1.2, and 4. Since focus is only on the sensitivity of the results to the near-wall grid spacing, serious attention is not paid to guarantee the accuracy of the respective reference case simulations. A significant disagreement at $B/D=2$ may be noticed, and this may be attributed to a sudden jump in the Strouhal number around $B/D=2.1$ (Re$=1.0\times10^3$) in Okajima et al. (1992) which is quite close to the present case with $B/D=2$. But this is not consistent with the experiment (Okajima 1982), where the Strouhal number jump was observed near $B/D=3$. This disagreement is further to be investigated, and does not necessarily invalidate the present approach since we are mainly interested in the sensitivity of the results to the near-wall grid spacing.

Extensive numerical simulations have been performed, whose reuslts are summarized in Table 3 for Re$=2.2\times10^4$. Sensitivity of an unsteady computation to the grid spacing at various aspect ratios can be investigated by comparing the results in the Table. The Strouhal number is fairly insensitive to the grid spacing change with less than 5% except for the case with $B/D=0.5$. The average drage coefficient is somewhat sensitive to the grid spacing for the cases with $B/D\leq1.0$, while the fluctuating force coefficients are the most sensitive for the case with $B/D=1.0$, and less so for the case with $B/D=2$ than for $B/D=0.5$.

Flow around a rectangular cylinder can be characterized by two related features: detached shear layers from the forward corners and separation bubbles on the sides. Even though the two flow features are caused by the adverse pressure gradient, their downsteam evolutions are not sensitive to the wall layer resolution since they are governed primarily by the inviscid dynamics. Furthermore, differently from a circular cylinder case, the separation points are fixed at the for-

Table 3 Parameters of the computations with Re$=2.2\times10^4$

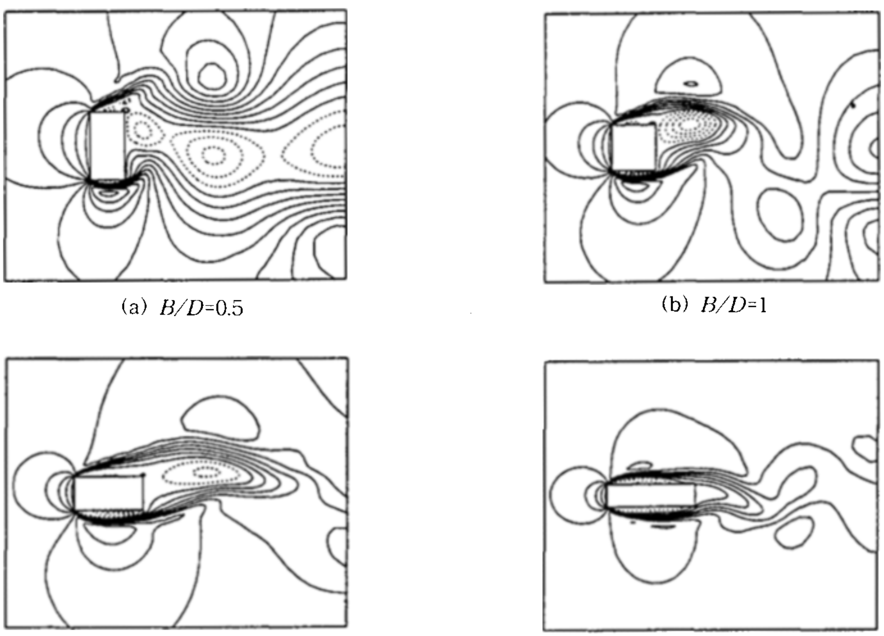
Case	B/D	Δx^*	c_D	c_L	St
RHL	0.5	0.035	3.74±0.65	±4.16	0.133
CHL	0.5	0.090	2.40±0.18	±2.34	0.200
RSL	1.0	0.035	2.22±0.38	±2.35	0.135
CSL	1.0	0.074	1.62±0.04	±0.41	0.140
R2L	2.0	0.035	1.60±0.18	±1.63	0.067
C2L	2.0	0.090	1.46±0.12	±1.06	0.070
R4L	4.0	0.034	1.00±.004	±0.11	0.131
C4L	4.0	0.072	0.95±.000	±.001	0.125

Definition : $\Delta X^*=(\Delta X_i)_{\min}/D$

ward corners of the cylinder, and the shear layers are little affected by the turbulent boundary layers before separation.

Instantaneous streamwise velocity fields are shown in Fig. 3 at the same time instants of those in Fig. 1, for more comprehensive understanding of the flow fieds. For the cases with $B/D=0.5$ and 1, the detached shear layers are alternately convected into the leeward side of the cylinder through fully separated regions on the sides. The instantaneous flow field, for example, in Fig. 3 (b) is taken just after the shear layer is advected around the bottom corner. Nearly circular vortices are formed from the advected shear layer and shed regularly behind the cylinder. In those cases, shear layer resolution is critical in predicting the unsteady forces, since its self-induced motion and the strength determine the advection and the flow separation, respectively. For $B/D=2$, the shear layers are convected mostly in the downstream direction, not toward the leeward side of the cylinder, and the reverse flow within the separation bubble behind the cylinder is not as strong as that for the case with $B/D\leq 1$. For $B/D=4$, the detached shear layers from the forward corners always reattach on the sides and alternately formed another weak secondary shear layers are advected into the leeward side of the cylinder. Since the main shear layers do not play important roles, grid resolution for $B/D\geq 2$ is not as important in the computations as in the cases with $B/D\leq 1$.

In summary, the computational results are more sensitive to the grid spacing for the cylinders with $B/D=0.5$ and 1 than for $B/D=2$ and 4. And the mesh is suggested to be refined in the shear layer region rather than in the wall vicintiy. However, the mesh in the wall vicinity is also expected to be fine, at least, to predict the formation of the shear layers. For a strutured grid system which is inconvenient to refine meshes locally in the shear layer region, the mesh refinement criteria could not directly be obtained in the present work, and are left for further investiga-



(a) $B/D=0.5$ (b) $B/D=1$

(c) $B/D=2$ (d) $B/D=4$

Fig. 3 Instantaneous streamwise velocity fields at the minimum lift phase of the oscillation. for case (a) RHL, (b) RSL, (c) R2L and (d) R4L. Solid lines denote for positive, and dashed lines for negative velocity. ($\Delta U=0.2$)

tion. In the present work, the grid spacing is changed more or less uniformly in space near the cylinder from the reference to the coarse grid case, where the mesh is coarsened not only in the shear layer region but also in the immediate wall vicinity.

3.2 Effects of the Reynolds, number, Re

The effects of the grid spacing at different Reynolds numbers on the force coefficients are investigated. Another set of computations summarized in Table 4 are carried out for the higher Reynolds number, $Re=10^6$. Sensitivity of the computational results to the grid spacing at different flow Reynolds numbers is investigated by comparing the results of $Re=2.2\times10^4$ with those of $Re=10^6$. Since focus is on the sensitivity of the results to the grid spacing, much attention is not paid to guarantee the accuracy of the respective reference cases at the high Reynolds number except for the case with $B/D=2$, where grid was refined to the level of $\Delta x_i \cong 20$(case F2H). Accuracy of the computations could not be justified directly for the high Reynolds number case, since no thorough experimental or computational results were available. Even though the grid spacing in case F2H is refined by about a factor of 30 from case R2H, not a significant difference is observed in the results, which indirectly verifies the properness of all the reference computations.

The evolution of the drag and lift coefficients for case F2H are shown in Fig. 4. Differently from the low Reynolds number case (Fig. 2), force coefficients show much irregular variations with finite number of dominant oscilation modes. Since the osillation amplitudes could not be defined unambiguously, the rms fluctuation amplitudes are given in Table 4 as a quantitative measure of force coefficients' oscillation.

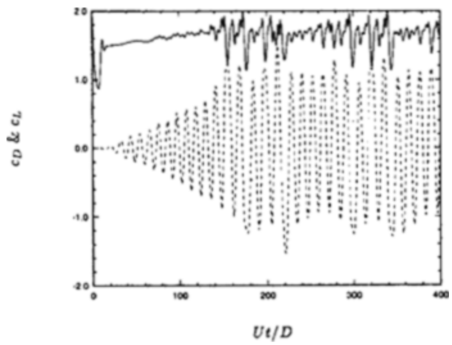


Fig. 4 Evolution of force coefficients in case F2H ($B/D=2$, $Re=1.0\times10^6$) : —— c_D, ---- c_L

Relative sensitivity of the results to the grid spacing at different Reynolds numbers can be identified by comparing the results at $Re=2.2\times10^4$ (Table 3) with those at $Re=10^6$ (Table 4). Grid spacing for the reference computations at the lower Reynolds number is $\Delta x_i \cong 20$, which is not practicable for flows with $Re=O(10^7)$ or higher. If the near-wall resolution is required in the same viscous wall unit, it is alsomt impossible to simu-

Table 4 Parameters of the computations with $Re=10^6$

Case	B/D	Δx^*	$(c_L)_{mean}$	$(c_L)_{rms}$	$(c_L)_{rms}$	St
RHH	0.5	0.035	3.87	0.72	2.86	0.136
CHH	0.5	0.090	2.52	0.14	1.82	0.200
RSH	1.0	0.035	2.18	0.33	1.28	0.130
CSH	1.0	0.074	1.65	.006	0.29	0.140
F2H	2.0	.0012	1.66	0.16	0.85	0.067
R2H	2.0	0.035	1.62	0.14	1.08	0.066
C2H	2.0	0.090	1.50	0.12	0.76	0.068
R4H	4.0	0.034	1.08	.016	0.22	0.131
C4H	4.0	0.072	0.98	.000	.002	0.126

late turbulent flows of engineering interest. In order to critically assess the grid spacing requirements, most computations for the high Reynolds number cases were also carried out with the same grids used for the cases with $Re=2.2\times10^4$.

With the increase of Reynolds number, amplitudes of unsteady force coefficients tend to increase (for a regular harmonic fluctuation, oscillation amplutude is $\sqrt{2}$ times larger than its rms value). It is found that the results are slightly less sensitive to the grid spacing at the high Reynolds number, where the near-wall grid spacing is usually of $O(10^3)$ in the viscous wall unit. If the unsteady flow structure is dominated by the near-wall turbulence, prediction accuracy may have been significantly affected by the near-wall grid spacing change, which is not the case with the present study. This is consistent with the observation (Hackman, 1982) that the reattachement lengh is fairly insensitive to the use of different wall functions used in the computations of the turbulent flow over a backward facing step. Relative insensitivity of the results to the near-wall grid spacing implies that the dominant feature of the flow may not be the near-wall turbulence, rather the free shear layer coming off the forward corners of the cylinder. Similar conclusion was reached by Djilali et al. (1991) for a bluff rectangular plate.

The above observation implies that the grid spacing should be specified to resolve the free shear layer evolution (which is less sensitive to the flow Reynolds number), instead of the turbulent boundary layer. Therefore, the non-dimensional grid spacing parameter used in Tables 3 and 4, Δx^*, may be a reasonable criterion for the computations. Furthermore, the grid refinement is to be made in the free shear layer region, rather than in the wall vicinity. In conclusion, unsteady wind loading may reasonably by computed for a bluff civil structure using a practicable grid system even at $Re=O(10^7)$ or higher.

4. Summary and Conclusion

Sensitivity of the computational results to the near-wall grid spacing for an unsteady turbulent flow around a rectangular cylinder is discussed in the presnet paper. Effects of the aspect ratio and the flow Reynolds number on the sensitivity are investigated.

The effects of the spatial resolution for the computations of unsteady flows over a rectangular cylinder become less pronounced at the high Reyolds number and also for a rectangular cylinder with $B/D>1$. Insensitivity to the near-wall grid spacing at the high Reynolds number is thought to be due to the dominance of inviscid flow characteristics, where grid refinement is to be made in terms of $\Delta x_i/D$ instead of Δx_i, which give hope in simulating extremely high Reynolds number turbulent flows of engineering importance. Insensitivity for the case with $B/D>1$ is thought to be from less critical dependence of the flow features on the spatial resolution. For cases with $B/D>1$, separation bubbles are closed on the sides ($B/D=4$) or very weak in strength, and the shear layers fromed at the forward corners are advected downstream, having little effects on the flow fields in the immediate neighbor of the cylinder. For cases with $B/D\leq1$, the side separation bubbles are alternately connected to the leeward recirculation zone, and the strong shear layers formed at the forward corners are advected along the cylinder surface. Therefore, the grid spacing, or the shear layer resolution, plays a decisive role in computing unsteady aerodynamic forces on the cylinder with $B/D\leq1$. The usual civil structures, however, have large aspect rations, and the computational method may be expected to shed some lights in the future of wind engineering.

Acknowledgment

Computations in the present work had been carried out using CRAY C90 supercomputer is SERI, and the generous CPU time and technical support by SERI Supercomputing Center are greatly appreciated.

References

Bradshaw, P., Launder, B. E. and Lumley, J.

L., 1994, "Collaborative Testing of Turbulence Models. In Advances in Computational Methods in Fluid Mechanics," (eds. Ghia, K. N. Ghia U. and Goldstein D.) pp. 77~82. *ASME*: New York, New York.

Djilali, N., Gartshore, I. S. and Salcudean, M., 1991, "Turbulent Flow Around a Bluff Rectangular Plate Part Ⅱ: Numerical Predictions," *J. Fluids Eng. Trans. ASME* 113, pp. 60~67.

Durbin. P. A., 1995, "Separated Flow Computations with the k-ε-ν^2 Model," *AIAA J.* 33, pp. 659~664.

Franke, R. and Rodi, W., 1991, "Calculation of Vortex Shedding Past a Square Cylinder with Various Turbulence Models," *In Proc. 8th Symp. on Turbulent Shear Flow*, Munich, Germany, Sept. 9-11, 20.1.1~20.1.6.

Hackman, L. P., 1982, A Numerical Study of the Turbulent Flow Over a Backward Facing Step Using a Two-Equation Turbulence Model," Ph.D. Thesis. Univ. of Waterloo: Waterloo, Canada.

Kato, M. and Launder, B. E., 1993, "The Modelling of Turbulent Flow Around Stationary and Vibrating Square Cylinders," *In Proc. 9th Symp. on Turbulent Shear Flow*, Kyoto, Japan, Aug. 16-18 10.4.1~10.4.6.

Launder, B. E. and Sharma, B. T. 1974, "Application of the Energy Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc," *Lett. Heat and Mass Transfer*, pp. 131~138.

Launder, B. E. and Spalding, D. B., 1974, "The Numerical Computation of Turbulent Flows," *Comp. Meth. Appl. Mech. Eng. 3*, pp. 269~289.

Lee, S, 1996, "Unsteady Aerodynamic Force Prediction on a Square Cylinder Using k-ε Turbulence Models," *In Seocnd Comp. Wind Eng. Conf.*, Fort Collins, USA, Aug., pp. 4~8.

Lyn, D. A. Rodi, W., 1994, "The Flapping Shear Layer Formed by Flow Separation From the Forward Corner of a Square Cylinder," *J. Fluid Mech* 267, pp. 353~376.

Murakami, S., 1993, "Comparison of Various Turbulence Models Applied to a Bluff body. *J. Wind Eng. Ind. Aerodyn.* 46, pp. 21~36.

Murakami, S. and Mochida, A., 1995, "On Turbulent Vortex Shedding Flow Past 2D Square Cylinder Predicted by CFD," *J. Wind Eng. Ind. Aerody.* 54, pp. 191~211.

Okajima, A., 1982, "Strouhal Numbers of Rectangular Cylinders," *J. Fluid Mech.* 123 pp. 279~398.

Okajima, A., Ueno, H. Sakai, H, 1992, "Numerical Simulation of Laminar and Turbulent Flows Around Rectangular Cylinders," *Int. J. Num. Meth. Fluids* 15, pp. 999~1012.

Press, W. H. Flannery, B. P., Teukolsky, S. A. and Vetterling, W. T., Numerical Recipes: The Art of Scientific Computing. Cambridge Univ. Press: New York, New York, pp. 423~429.

Yakhot, V., Orszag, S. A., Thangam, S., Gatski, T. B. and Speziale, C. G., 1992, "Development of Turbulence Models for Shear Flows by a Double Expansion Technique," *Phys. Fluids* A 4, pp. 1510~1520.