

SIMULATION OF WIND FLOW AROUND A GROUP OF SQUARE CYLINDERS WITH VARIABLE TRANSVERSE SPACING BY k - ϵ TURBULENCE MODEL

Muhammad Mustafa*, Amalesh Chandra Mandal, Nusrat Jahan Chhanda and Md. Quamrul Islam

Received: Jul 23, 2007; Revised: Nov 21, 2007; Accepted: Nov 23, 2007

Abstract

The effect of wind loading on buildings and structures has always been a major area of interest for structural engineers and architects. In most cases, the numerical simulations of wind flow are not available with the computational fluid dynamics. In the present paper, the theoretical investigations of wind effect as well as wind flow pattern on staggered square cylinders are studied. Wind flow around a group of square cylinders is simulated by a k - ϵ turbulence model (Mondal and Bhattacharia, 2005) in finite volume method and the relevant flow patterns of wind have been investigated. The calculated results are compared with the experimental results on square cylinders conducted by Mandal and Islam (1980 - 1981) and very good agreements are found.

Keywords: Group of square cylinders, k - ϵ turbulence modelling

Introduction

During the past half century much attention has been paid to the study of wind loading. The occurrences of certain disastrous collapses of suspension bridges and damage to buildings and structures should not be encountered as minor criteria for design purposes. Until now extensive research work has been carried out on isolated bluff bodies, but interference among such bodies is very important as well. Little information is available concerning the flow over staggered square cylinders, although this is a problem of considerable practical significance. The present work would contribute to the

knowledge of wind action on groups of square buildings and structures. Mandal and Islam (1980 - 1981) in their papers presented the study of wind effect on square cylinders. They measured the pressure distributions on a single cylinder at various angles of attack and also on a group of square cylinders at various transverse as well as longitudinal spacings. It was mentioned in their paper that a model study of the wind effect around a group of square cylinders would be useful to find the wind load on a group of tall square-shaped buildings. In their experimental investigation wind loadings

Mechanical Engineering Department, Bangladesh University of Engineering and Technology (BUET), Dhaka-1000, Bangladesh, Tel: +880-02-9665636; Fax: +880-02-8613046. E-mail: mmustafa@me.buet.ac.bd, amalesh@me.buet.ac.bd, njchhanda@gmail.com, quamrul@me.buet.ac.bd

** Corresponding author*

were obtained on a group of square cylinders with various transverse and longitudinal spacings. It was observed from the experimental results that the wind load on the individual cylinders of the group was less severe than that on a single cylinder in most of the cases. Nakamura and Matskawa (1987) experimentally investigated the vortex excitation of rectangular cylinders with a long side normal to the flow in a mode of lateral translation using free and force oscillation methods. Hua (1971) conducted measurements of fluctuating lift and oscillating amplitudes on a square cylinder in a wind tunnel test. Okajima (1982) carried out the experiments in a wind tunnel as well as in a water tank on the vortex shedding frequencies of various rectangular cylinders. Barriga *et al.* (1975) and Lee (1975) carried out work on single square cylinders. They measured mean pressure distributions at various angles of attack with different turbulence intensities and scales. Leuthesser (1971) presented in his paper the results of static wind loading obtained from wind tunnel tests on scale models of a typical building configuration consisting of four buildings each with a different height and cross section.

Flow behavior around circular cylinders is a classical problem in fluid mechanics with a variety of practical applications, ranging from tall chimneys exposed to atmospheric boundary layer flows to cooling systems of nuclear reactors. The proximity of the adjacent structures under certain conditions introduces adverse or beneficial effects. From an aerodynamic perspective, a strong interaction takes place in the flow field around multiple body configurations that are sensitive to approach flow characteristics, as well as the angle of attack. A host of studies has addressed the interference effects between two, three, and even four cylinders of finite height in uniform and/or turbulent flow which has been the focus of efforts in recent decades. Still, there is less information available on the aerodynamic characteristics of multiple finite cylinders.

The majority of this work has focused on the calculation of the mean and fluctuating

pressure distributions on cylinders in tandem as described by Luo *et al.* (1996). Some attention has also been given to other configurations, such as staggered and side-by-side alignments in work by Sun and Gu (1995) and Sun *et al.* (1992). Still, the majority of the work has been dedicated to the study of localized effects, with few studies giving primary consideration to lift and drag forces, especially their fluctuating components (Sun *et al.*, 1992) and their spectra. Furthermore, since most of the previous work was carried out in smooth flow, there is a shortage of information regarding the fluctuating forces and pressures on cylinders in turbulent boundary layer flows.

Vengadesan and Nakayama (2005) have evaluated turbulent flow over the bluff body by a large eddy simulation model. In the previous research, three numerical models were used for investigation of turbulent flow over a square cylinder, namely: (i) conventional Smagorinsky model (Smagorinsky, 1963), (ii) dynamic Smagorinsky model (Germano *et al.*, 1991) and (iii) one equation model (Yoshizawa and Horiuti, 1985).

The experimental results of pressure distribution on square cylinders are available (Mandal and Islam, 1980 - 1981) for the uniform flow having turbulence intensity of 0.4. The calculated values were obtained using a k - ϵ turbulence model with turbulence intensity of 0.1. For isotropic flow at a certain distance from the honey comb, the turbulent intensity is defined as the ratio of rms value of oscillation and the mean velocity. The program was originally developed for smooth flow based on turbulence intensity of 0.1, later the calculated results were compared with the experimental results for turbulence intensity of 0.4. Since the turbulence intensity for both the calculated and experimental results a very small, there will be negligible effect on the results for comparison.

k - ϵ Turbulence Model

The standard k - ϵ model of turbulence is a two-equation model in which the solution of

two separate transport equations allows the turbulence velocity and length scales to be independently determined. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism. It is based on a model transport equation for the turbulent kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation and that of ε is obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the k - ε model, flow is assumed as fully turbulent and the effect of molecular viscosity is considered as negligible. The following two equations represent the transport of turbulent kinetic energy, k and its rate of dissipation rate, ε respectively:

$$\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (1)$$

and

$$\rho \frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (2)$$

The above Eqns. (1) and (2) are used avoiding time dependent term in order to make the steady condition. In these equations, G_k represents the generation of turbulent kinetic energy due to the mean velocity gradients. From the exact equation for the transport of k , this term is defined as

$$G_k = -\rho \overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i}$$

To evaluate G_k in a manner consistent with the Boussinesq hypothesis,

$$G_k = \mu_t S^2$$

where S is the modulus of the mean rate-of-strain tensor, defined as

$$S \equiv \sqrt{2S_{ij}S_{ij}}$$

with the mean strain rate S_{ij} given by

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

G_b is the generation of turbulent kinetic energy due to buoyancy. It is calculated as

$$G_b = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i} \quad \text{where, } \beta \text{ is the coefficient}$$

of thermal expansion and Pr_t is the turbulent Prandtl number for energy. Y_M represents the contribution of fluctuating dilation in compressible turbulence to the overall dissipation rate. In the present model, $Y_M = 0$ as compressibility effect is neglected for incompressible flow. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants; σ_k and σ_ε are the turbulent Prandtl numbers for k and ε , and S_k and S_ε are user defined source items respectively.

The turbulent viscosity, μ_t is computed by combining k and ε as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (3)$$

In order to make good agreement with the experimental results, the values of the various constants were chosen as follows:

$$C_{1\varepsilon} = 3.5, C_{2\varepsilon} = 4.00, \text{ and } C_{3\varepsilon} = 2.01, \\ \sigma_k = 1.0, \text{ and } \sigma_\varepsilon = 1.3$$

Non-dimensional pressure, drag and lift coefficients were calculated as follows:

Pressure coefficient is defined as,

$$C_p = 2(P - P_0)/\rho U_0^2$$

Drag coefficient is defined as,

$$C_D = 2F_D/\rho U_0^2$$

Lift coefficient is defined as,

$$C_L = 2F_L/\rho U_0^2$$

where, P_0 is the free stream pressure, F_D is the drag force and F_L is the lift force.

Boundary Conditions

Arrangements of the cylinders and two dimensional meshing are shown in Figures 1 and 2 respectively. In respect of the first cylinder as shown in Figure 1, the boundaries are chosen at the front, rear, top and bottom up to $7D$, $11D$, $5D$, and $5D$ respectively from the surface of the cylinder where D is the side dimension of square cylinders.

These boundary conditions are kept fixed with respect to the first cylinder for any staggered arrangements of the cylinders. Based on this boundary condition the $k-\epsilon$ turbulence model is applied to evaluate the two-dimensional flow simulation around the group of square cylinders. At the front boundary condition the velocity is chosen as uniform keeping the Reynolds no. at 2.78×10^4 based on the side dimension of the cylinder. At the rear boundary outlet pressure is considered as 1 atm. Slip boundary conditions are considered at the top and bottom boundaries. Each face of the square cylinders is kept as wall-condition or no-slip condition. FLUENT software is used to solve the governing equation in the test section. Grid independency tests are done by taking total grid points 40,000, 50,000, 70,000, 90,000, and 100,000. The number of grid points is taken as 90,000 in the present modeling. Figure 2 shows the computational grids of wind flow field around a cylinder.

Results and Discussion

In Figure 3 the flow pattern of wind on the upstream cylinder at constant longitudinal spacing $L_1 = 1D$ is shown. It is observed from the figure that for various transverse spacings and also for the single cylinder a pair of vortices in opposite sense is generated at the rear of the cylinders. It can be further observed that with an increase of the transverse spacing the pair of vortices moves slightly towards the downstream side. It is clear from each flow pattern that the separation occurs at the corner of the front face with a tendency of reattachment as the flow advances. It is further noticed that the flow pattern of the cylinder with transverse spacing of $L_2 = 4D$ is nearly identical to that for the single cylinder. Due to the presence of the downstream cylinder there has been a significant effect in the flow field as can be noticed for the lower transverse spacing.

The flow pattern on the downstream cylinder for various transverse spacings at constant longitudinal spacing of $L_1 = 1D$ is presented in Figure 4. With an increase of transverse spacing the flow pattern varies as can be seen from Figure 4. At the top and bottom surface of the cylinder the pattern is different.

The C_p -distributions on the upstream cylinder with transverse spacings of $L_2 = 1D$, $2D$, and $4D$ at constant longitudinal spacing of

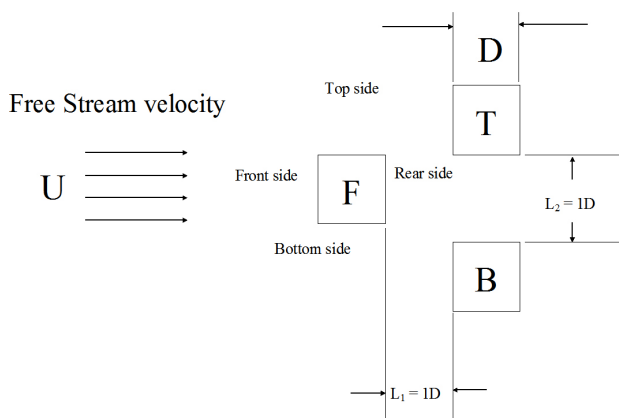


Figure 1. Arrangement of the cylinders in staggered condition

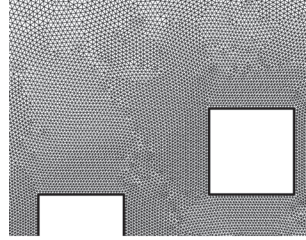


Figure 2. Two dimensional meshing around cylinders

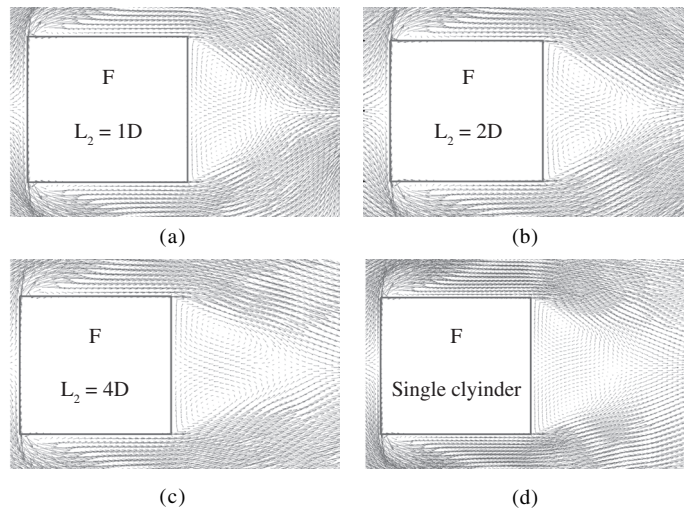


Figure 3. Flow pattern of wind on the upstream cylinder with various transverse spacings (L_2) at constant longitudinal spacing $L_1 = 1D$

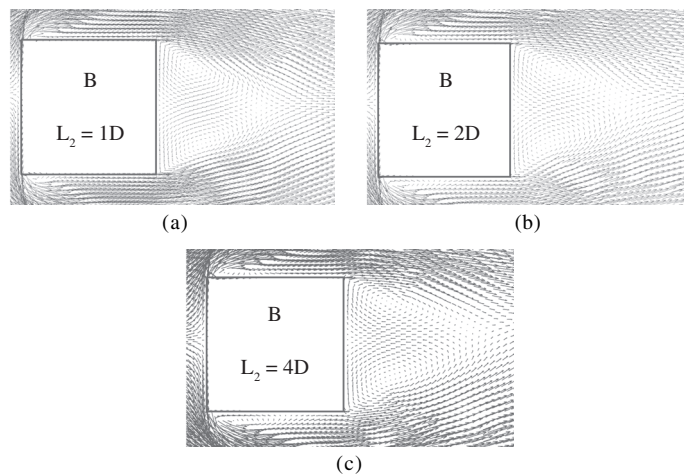


Figure 4. Flow pattern on downstream cylinder for various transverse spacings L_2 at constant longitudinal spacing of $L_1 = 1D$

$L_1 = 1D$ are shown in Figure 5. It is revealed from this figure that for all transverse spacings, the pressure distributions on the top and bottom surfaces are identical, which occurs due to symmetry. At the front face there are positive pressures whereas on the other faces the pressures are negative. It can be observed from this figure that with the decrease of the transverse spacing the back pressures begin to increase (negative C_p -values decrease). Due to the presence of the downstream cylinder, momentum transfer occurs which increases with the decrease of transverse spacing resulting in pressure recovery on the top, bottom and rear of the upstream cylinder. The calculated values of the pressure distributions give the very reasonable correlation with the experimental values for all transverse spacings, as the Figure reveals. It is further observed from the Figure that the parabolic distributions of pressures appear on the front face. At the middle of this face the stagnation point occurs where the C_p is unity. On the rear face of the upstream cylinder the pressure distributions are nearly of uniform nature. The back pressure is controlled by the distance of transverse spacings

compared with pressure distribution for the single cylinder. It is observed that at all the transverse spacings pressure increases on the top, bottom and rear surfaces.

In Figure 6 pressure distributions on the downstream cylinder for various transverse spacings at a constant longitudinal spacing of $L_1 = 1D$ are presented. Since the cylinders marked T or B are identical, only one of them is shown. The pressure distribution at the front face is affected remarkably.

At the transverse spacing of $L_1 = 1D$, which occurs due to the wake generated by the upstream cylinder, asymmetric pressure distributions are observed at the top and bottom surfaces of the downstream cylinder except at $L_2 = 4D$. At higher transverse spacing the downstream cylinder goes almost beyond the wake region produced by the upstream cylinder. In this case flow characteristics around the downstream cylinder appear nearly identical to those of the single cylinder.

With a decrease of the transverse spacing relatively larger wakes are generated. The flow on the front face never becomes potential as in the case of a single cylinder. The velocity with

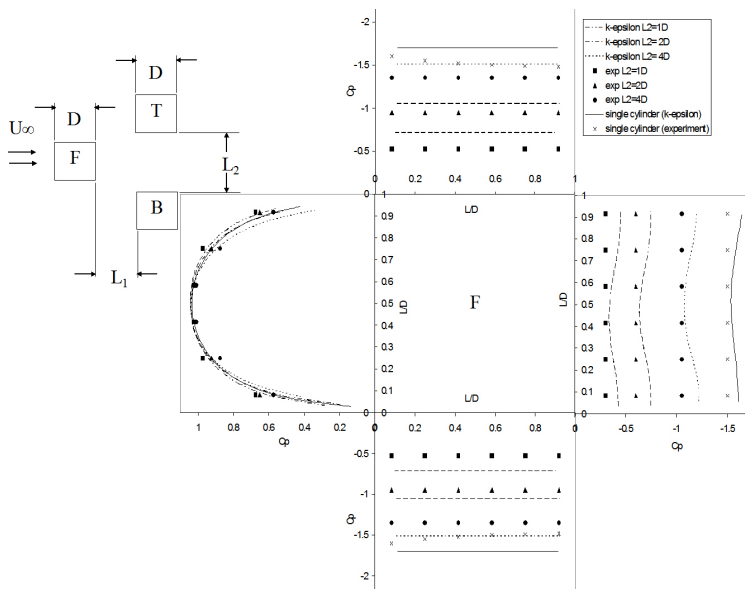


Figure 5. Pressure distributions for various transverse spacings L_2 at constant longitudinal spacing of $L_1 = 1D$

which the flow appears on the front face of the single cylinder is greater than the velocity with which it appears on the front face of the downstream cylinder, because the mean velocity in the wake which appears on the front face of the

downstream cylinder is less than the free stream velocity. Due to this cause the pressure distributions on the front face are quite different from those produced on the front face of the single cylinder.

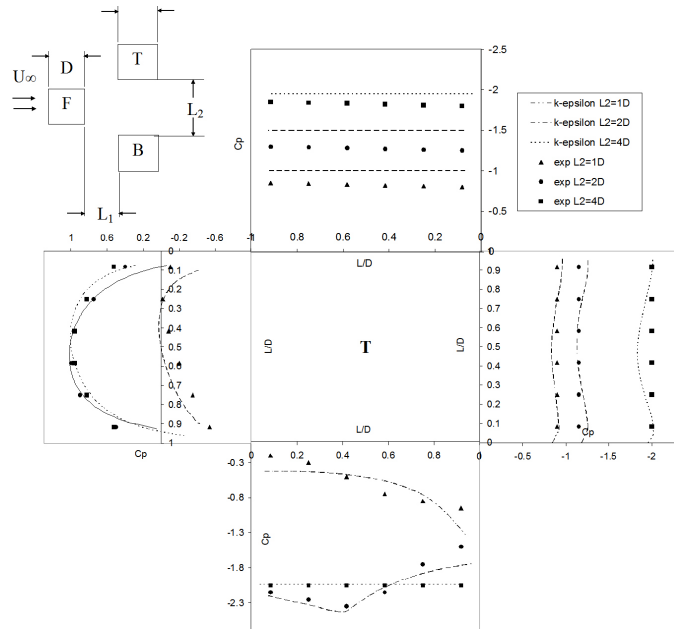


Figure 6. Pressure distribution on downstream cylinder for various transverse spacings at constant longitudinal spacing of $L_1 = 1D$

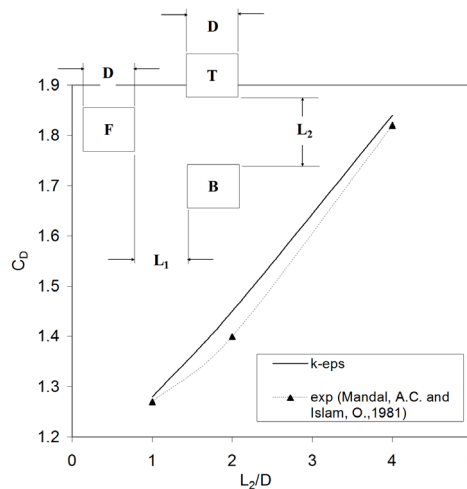


Figure 7. Variation of drag co-efficient (C_D) on upstream cylinder F with different transverse spacing (L_2) for constant longitudinal spacing ($L_1 = 1D$)

At the bottom surface of the downstream cylinder it is seen that, for $L_2 = 1D$, pressure recovery is tremendously high near the front corner, whereas towards the rear corner again separation appears. On the other hand at $L_2 = 2D$, high suction occurs on the front side and pressure recovery occurs on the rear side of the bottom face. However at $L_2 = 4D$ almost uniform pressure distribution occurs only through the entire face and on the top and bottom faces pressure distributions are almost identical. It occurs because at a large transverse spacing the downstream cylinder goes almost

out of the wake region generated by the front cylinder.

Variation of drag coefficient on the upstream and downstream cylinders with transverse spacing at constant longitudinal spacing of $L_1 = 1D$ is shown in Figures 7 and 8 respectively. It is seen from both the figures that with the increase of transverse spacing the drag coefficient increases. From the pressure distributions as shown in Figure 5 for the upstream cylinder it is seen that for various transverse spacings of $L_2 = 1D$, $2D$, and $4D$, the distributions are almost uniform along the

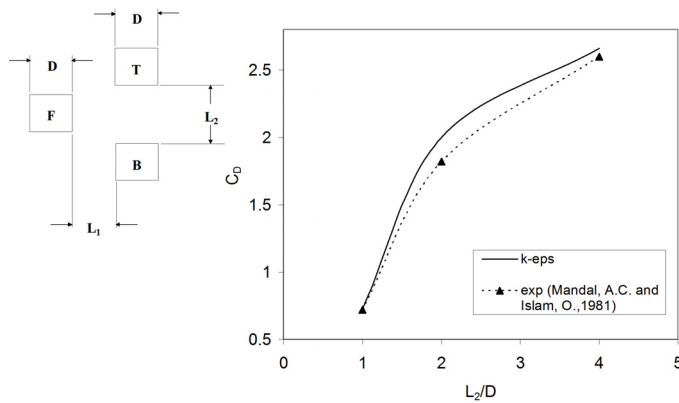


Figure 8. Variation of drag co-efficient (C_D) on downstream cylinder T or B with different transverse spacing (L_2) for constant longitudinal spacing ($L_1 = 1D$)

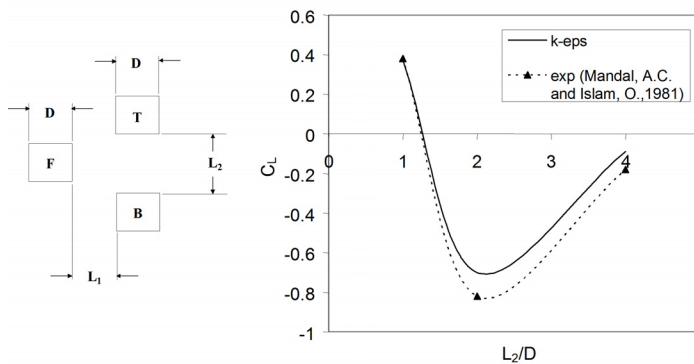


Figure 9. Variation of lift co-efficient (C_L) on downstream cylinder T or B with different transverse spacing (L_2) for constant longitudinal spacing ($L_1 = 1D$)

sides of the cylinder and with the increase of transverse spacing back pressure decreases. At a transverse spacing of $L_2 = 4D$, the back pressures are comparatively smaller. As a result the drag coefficient is higher at $L_2 = 4D$. There is very good agreement between the calculated and the experimental values of the drag coefficients. The nature of the drag coefficients as shown in Figure 8 for the downstream cylinder can be explained almost in the same way. The variation of the lift coefficients with transverse spacing on the downstream cylinder at constant longitudinal spacing of $L_1 = 1D$ is shown in Figure 9. The lift coefficients on the upstream cylinder are zero due to symmetric distribution of pressures on the top and bottom faces of the first cylinder. It can be seen from the figure that, at $L_2 = 4D$, the lift coefficient approaches near zero value which occurs for the single cylinder, that is, the downstream cylinder. At $L_2 = 4D$ it almost becomes similar to the single cylinder as it almost goes beyond the wake region generated at the rear of the upstream cylinder. From the pressure distribution as shown in Figure 6 for the downstream cylinder, the values of the lift characteristics can be analysed.

Conclusions

The following conclusions are drawn for simulation of wind flow around a group of square cylinders:

- It has been investigated that with a decrease of transverse spacing the back pressure of both the front and rear cylinders increases remarkably thereby decreasing drag coefficient.
- From the flow pattern it is seen that with the increase of the transverse spacing the pair of vortices generated behind the downstream cylinder and the wake region move towards the downstream side slightly.
- At the transverse spacing of $1D$, drag coefficient decreases remarkably compared with that for the single cylinder. At transverse spacing of $2D$, the lift coefficient increases reasonably.

- Due to the presence of downstream cylinders the C_p values increase on the top, bottom and rear sides of the front cylinder.

Acknowledgement

The authors acknowledge the help of Dr. M. Mamun, Assistant Professor, Department of Mechanical Engineering, Bangladesh University of Engineering and Technology, Bangladesh for allowing them to use the FLUENT software in his personal computer. Authors are also grateful to Dr. T. Setoguchi, Professor, Department of Mechanical Engineering, Saga University, Japan in this regard.

References

- Barriga, A., Crowe, C., and Roberson, J.A. (1975). Pressure distribution on a square cylinder at a small angle of attack in a turbulent cross flow. Proceedings of the 4th International Conference on Wind Effects on Buildings and Structure; June 19-21, 1975; London, UK, p. 89-93.
- Germano, M., Piomelli, U., Moin, P., and Cabot, W. (1991). A dynamic subgrid scale eddy viscosity model. *Phys. Fluids*, A.3:1,760-1,765.
- Hua, C.K. (1971). The behavior of lift fluctuation on the square cylinder in the wind tunnel test. Proceedings of the 3rd International Conference on the Wind Effects on Buildings and Structures; October 18-19, 1971; Tokyo, Japan, p. 911-920.
- Lee, B.E. (1975). The effect of turbulence on the Surface Pressure Field of a Square Prism. *J. of Fluid Mech.*, 69(2):263-282.
- Leutheusser, J. (1971). Static wind loading of grouped buildings. Proceedings of the 3rd International Conference on Wind Effects on Buildings and Structures; October 18-19, 1971; Tokyo, Japan, p. 211-220.
- Luo, S.C., Gan, T.L., and Chew, Y.T. (1996). Uniform flow past one (or two in tandem) finite length circular cylinder(s). *J. of Wind Engineering and Industrial*

- Aerodynamics, 59:69-93.
- Mandal, A.C. and Islam, O. (1980). A study of wind effect on a group of square cylinders with variable longitudinal spacings. *Mech. Eng. Res. Bull.*, Bangladesh University of Engineering and Technology, Dhaka, 3(1):21-26.
- Mandal, A.C. and Islam, O. (1981). A study of wind effect on a group of square cylinders with variable transverse and longitudinal spacings. *J. of Institution of Engineers*, Dhaka, Bangladesh, 9(1):35-39.
- Mondal, H.N. and Bhattacharia, S.K. (2005). Modeling and simulation of a gas combustor. *Proceedings of the International Conference on Mechanical Engineering*, 28-30 December, 2005; Dhaka, Bangladesh, p. FL-18(1-7).
- Nakamura, Y. and Matsukawa, T. (1987). Vortex excitation of rectangular cylinders with a long side normal to flow. *J. Fluid Mech.*, 180:171-191.
- Okajima, A. (1982). Strouhal numbers of rectangular cylinders. *J. Fluid Mech.*, 123:379-398.
- Smagorinsky, J. (1963). General circulation experiments with primitive equations-I. The Basic experiment. *Mon. Weather Rev.*, 91:99-165.
- Sun, T.F. and Gu, Z.F. (1995). Interference between wind loading on group of structures. *J. of Wind Engineering and Industrial Aerodynamics*, 55:213-225.
- Sun, T.F., Gu, Z.F., He, D.X., and Zhang, L.L. (1992). Fluctuating pressure on two circular cylinders at high reynolds numbers. *J. of Wind Engineering and Industrial Aerodynamics*, 41-44:577-588.
- Vengadesan, S. and Nakayama, A. (2005). Evaluation of LES models for flow over bluff Body from engineering application perspective. *Sdhan*, 30(1):11-20.
- Yoshizawa, A. and Horiuti, K. (1985). A statistically-derived subgrid-scale kinetic energy model for the large-eddy simulation of turbulent flows. *J. Phys. Soc. Jap.*, 54:2,834-2,839.