

Shear effects on flow past a rectangular cylinder with side ratio $B/D=5$

Qiang Zhou, Shuyang Cao*, Zhiyong Zhou

*State Key Laboratory of Disaster Reduction in Civil Engineering, Tongji University, Shanghai
200092, P. R. China*

ABSTRACT: Large Eddy Simulation (LES) is carried out to investigate shear effects on flow past a rectangular cylinder with side ratio $B/D=5$ at Reynolds number $Re=22,000$ (based on the thickness of the cylinder). Simulation results showed that the Strouhal number has no significant variation with oncoming velocity shear, while the peak fluctuation frequency of the drag coefficient becomes identical to that of the lift coefficient with increase in velocity shear. The intermittently-reattached flow that features the aerodynamics of the 5:1 rectangular cylinder in non-shear flow becomes more stably-reattached on the high-velocity side and more stably-separated on the low-velocity side. The mean of the drag force was found to have little variation while its standard-deviation increases with increase in velocity shear. The mean and standard-deviation of the lift and moment forces vary almost linearly with the velocity shear. The lift force acts from the high-velocity side to low-velocity side, which is similar to that of a circular cylinder but opposite to that of a square cylinder under the same oncoming shear flow condition.

KEYWORDS: A Rectangular cylinder, Shear parameter, Large eddy simulation, Aerodynamic forces, Vortex shedding, Flow reattachment

1 INTRODUCTION

Rectangular cylinders are common configurations in many structures, such as bridges, tall buildings and so on, thus the flow past a bluff body with a rectangular cross section is of direct relevance to the structural problems in which wind-induced vibration is one of the most important issues to consider. So far, the majority of studies on this issue have been conducted under uniform oncoming flow conditions in which vortices with equal strength alternately shed from each side of the cylinder. However, in many practical applications, a cylindrical structure is immersed in a non-uniform flow. A bridge deck in the atmospheric boundary layer is an example, in which the vertical mean wind profile is one factor to determine the wind load on bridge deck. The influence of velocity profile or velocity shear in the oncoming flow becomes more significant at the situation of non-synoptic wind like a downburst, where the wind speed increases rapidly near the ground and reaches its maximum at the height about 80-100m, and then decreases with increase in height resulting in a stronger velocity shear than in synoptic winds. Strong velocity shear may also be created in complex topography. Therefore, the variation of aerodynamic behaviors of a rectangular cylinder with the velocity shear needs to be studied in detail. In this study, a dimensionless shear parameter $\beta = G \times D / U_c = (dU / dy) \times (D / U_c)$ is defined to express the extent of velocity shear, where U_c is the mean velocity at the center plane, and D is the thickness of the rectangular cylinder and G is the velocity gradient, as illustrated in Fig. 1. The magnitude of the shear parameter implies the velocity difference between the up and down surfaces of the rectangular cylinder.

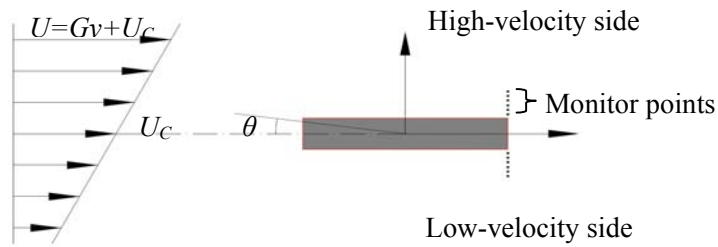


Fig.1. Schematic of shear flow configuration

Although the number of published papers is few, we are still able to find some studies about the shear flow past bluff bodies. Kiya et al. (1980) investigated the vortex shedding from a circular cylinder in shear flow and reported that the critical Reynolds number for the occurrence of vortex shedding becomes larger than in non-shear flow. Kiya's work was followed by many studies on the circular cylinder (e.g. Sumner and Akosile, 2003; Cao et al., 2010), and on the square cylinder (e.g. Cao et al., 2012). However, the common points of interest of these investigations were focused on how the Strouhal number varies with shear parameter and Reynolds number. Little attention was devoted to the shear effects on the aerodynamic forces and underlying background. In addition, there is no other reported study on the shear effects on a rectangular cylinder, except for the experimental study of Onirsuka et al. (2000). The insufficient study on the shear effect over a rectangular cylinder partly motivates present study.

In this paper, aerodynamic characteristics of a rectangular cylinder with side ratio $B/D=5$ in shear flows are investigated at $Re=22,000$, where the Reynolds number Re is based on the thickness of the rectangular cylinder D and the centerline velocity U_c . The side ratio 5:1 is chosen as the study objective because this kind of rectangular cylinder has often been adopted as a reference for investigations of the aerodynamics and aeroelasticity of a bridge deck and other structural members (Matsumoto, 1996; Bartoli and Righi, 2006). In addition, a $B/D=5$ rectangular cylinder has very delicate dynamic behaviors of vortex shedding that is characterized by massive flow separation due to the sharp leading edges and intermittent-reattachment on side surfaces forming unsteady separating bubbles, and deserves to be studied from the view points of fundamental study of bluff body aerodynamics.

With the occurrence of shear parameter, i.e. an asymmetry in the oncoming flow, the separated flow must differ on the two sides of the rectangular cylinder, resulting in different aerodynamic behaviors with those in non-shear flow. Non-zero-mean lift and moment forces, which are important factors in determining the behavior of flow-structure coupling, must appear. However, there is no reported research on the flow around 5:1 rectangular cylinder in shear flows. Onirsuka et al. (2000) studied experimentally the Strouhal number of rectangular cylinders in linear shear flows at $Re=3.2-9.7 \times 10^4$ at $B/D=0.2-3$. But the separated shear layer did not reattach on the side surface when $B/D=0.2-3$. The side ratio $B/D=5$ considered in this study involves intermittent flow reattachment, which makes the flow more sensitive to the oncoming velocity shear that brings separated shear layers with different strength and depth at the high and low velocity sides.

In this study, we investigated the vortex shedding and the aerodynamic forces on a rectangular cylinder in shear flows by Large Eddy Simulation (LES). We employed a structured grid mesh system for finite volume approximation of incompressible Navier-Stokes equations. The Fluent© package is used as a solver of the governing equations, but the options offered by Fluent© for simulation were carefully selected in order to achieve a good simulation. The numerical method and details applied in this study are described and validated by presenting the aerodynamic behaviors of a circular cylinder at $Re=3,900$. The aerodynamic characteristics of the 5:1 rectangular cylinder in shear flows at $Re=22,000$ are presented by comparison with available experimental and numerical results. Time-mean and unsteady flow patterns around the cylinder are

studied to enhance the understanding of the effects of velocity shear. In addition, the obtained results are compared with those of a circular cylinder (Cao et al. 2010) and a square cylinder (Cao et al. 2012) to achieve a comprehensive understanding of the shear flow around bluff bodies.

2 PROBLEM FORMATION AND NUMERICAL DETAILS

2.1 Problem formulation

The numerical model for the flow around a 5:1 rectangular cylinder is formulated using the Cartesian coordinate system. Eqs. (1) and (2) show the filtered continuity and Navier-Stokes equations for Large Eddy Simulation, in which the grid-scale turbulence is solved while the sub-grid-scale turbulence is modeled.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial (\bar{u}_i)}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\nu \frac{\partial \bar{u}_i}{\partial x_j} \right) - \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

where all the variables with the upper-marking are filtered components. The subgrid scale stresses (SGS stress), $\tau_{ij} = \bar{u_i u_j} - \bar{u_i} \bar{u_j}$, are expressed as $\tau_{ij} = -\frac{1}{3} \delta_{ij} \tau_{kk} + 2\nu_\tau \bar{S}_{ij}$

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right), \text{ and } \nu_\tau = l^2 |\bar{S}| \text{ where } l = C_s \bar{\Delta} \text{ (}\bar{\Delta} \text{ is the size of grid filter).}$$

We studied the flow around the 5:1 rectangular cylinder by performing three-dimensional unsteady simulation of the incompressible governing equations shown above with the aid of Fluent package. The options offered by Fluent© for simulation were carefully selected under the following considerations.

The velocity and the pressure are defined at the center of a control volume, while the volume fluxes are defined at the midpoint of their corresponding cell surfaces. In order to avoid the oscillating problems, the momentum Interpolation Method (MIM) developed by Rhie and Chow (1983) is used. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm proposed by Patankar and Spalding (1972) is utilized, in which governing equations are solved sequentially because of their non-linearity and coupling characteristics and the solution loop is carried out iteratively in order to obtain a converged numerical solution. The pressure field is extracted by solving a pressure correction equation that is obtained by manipulating continuity and momentum equations, while the velocity field is obtained from the momentum equations. In addition, the convergence criterion of the iterative calculation is set to be 10^{-6} , and it takes about 30 iterations to satisfy this criterion in the simulation.

In order to avoid instability caused by central-differencing schemes and non-physical wiggles, the bounded central differencing scheme is applied to spatial differencing of convection term, which is a composite normalized variable diagram (NVD, Leonard, 1991) scheme that consists of a pure central differencing, a blended scheme of the central differencing and the second-order upwind scheme, and the first-order upwind scheme. Meanwhile, a fully-implicit second-

order time-advancement scheme is chosen for temporal discretization to obtain stable and accurate simulation.

2.2 Numerical model and boundary conditions

As shown in Fig. 2 (a), the computational domain is $52.5D$ in x -direction, $18D$ in y -direction and $5D$ in z -direction. The blockage ratio is 5.55% that is smaller than the suggestion (6.4%) of So-hankar et al. (2000). The ratio of spanwise length L to chord length B is set to be $L/B=1$ in order to achieve a good simulation of the mean and RMS of aerodynamic forces (Tamura et al. , 1998). L/B was utility also in the simulation of Mannini et al. (2010) and Bruno et al. (2010) . A structured grid system with the number of $325(x) \times 164(y) \times 24(z)$ is used to adequately resolve the flow (see Fig.2) with the first grid Δd near the body surface given empirically as $0.1/\sqrt{Re}$ ($\Delta d / D = 6.5 \times 10^{-4}$). For the spatial discretization in spanwise direction, 24 cells are uniformly distributed with a grid length $\Delta z / B = 0.042$ that is smaller than the minimum requirement $\Delta z / B = 0.1$ recommended by Tamura et al. (1998). The non-dimensional time-step, Δt^* is $\Delta t^* = \Delta t U / D = 5 \times 10^{-3}$ which maintained the Courant number $CL < 1$.

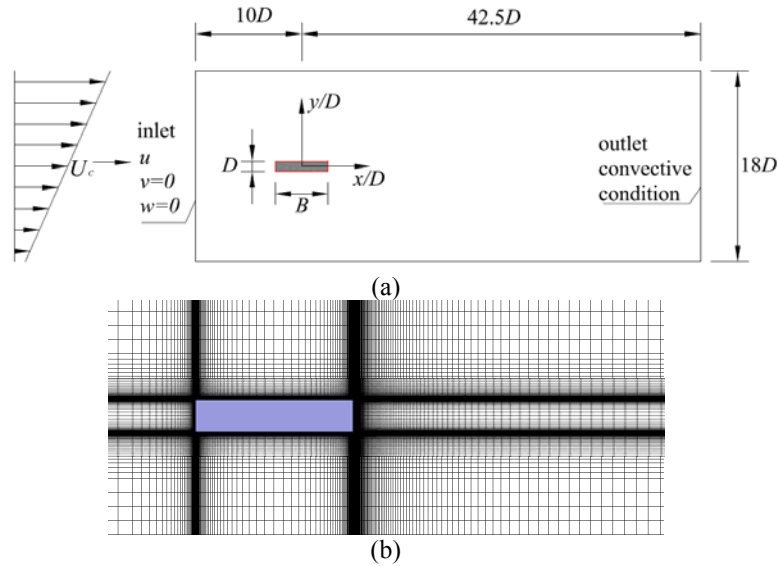


Fig.2 (a) Computational domain in x-y plane and boundary conditions, (b) Grid example near the cylinder

The boundary conditions for simulation are as follows:

Cylinder body surface: A no-slip condition for $u_t=0$ and a Neumann condition for pseudo-pressure ϕ are imposed.

Inlet: An UDF (User-Defined Function) condition, i.e., $U = 1 + Gy$, $v=0$ and $w=0$, and a Neumann condition of pseudo-pressure ϕ are imposed at the inlet boundary.

Outflow boundary: A convective boundary condition ($\partial \phi / \partial t + \bar{u} \cdot \partial \phi / \partial x = 0$) is applied for pseudo-pressure.

Spanwise: A periodic condition for velocity and pseudo-pressure are applied.

3 NUMERICAL VALIDATION

The flow over a circular cylinder at $Re=3900$ is often considered as a benchmark test model to check or confirm the accuracy of numerical simulation because there is plenty of reliable experimental and numerical data. Thus, in this study the numerical method is validated by comparing the mean and turbulence statistics of the flow around a circular cylinder at $Re=3,900$. Fig. 3(a) shows that the mean pressure coefficient distribution obtained by present numerical

shows that the mean pressure coefficient distribution obtained by present numerical method agrees well with other studies, while Fig. 3(b) shows that the streamwise mean velocity on the center line exhibits reasonable agreement also, especially at the near wake. Fig. 4 compares the turbulence statistics obtained at several downstream locations. Although the values deviate somewhat at several points, similar trendy of the variation of $u'u'$ and $u'v'$ in y direction is obtained. In short, the numerical method of present simulation produced reasonably good results, so it may be considered appropriate to simulation the flow over a bluff body with large scale vortex shedding.

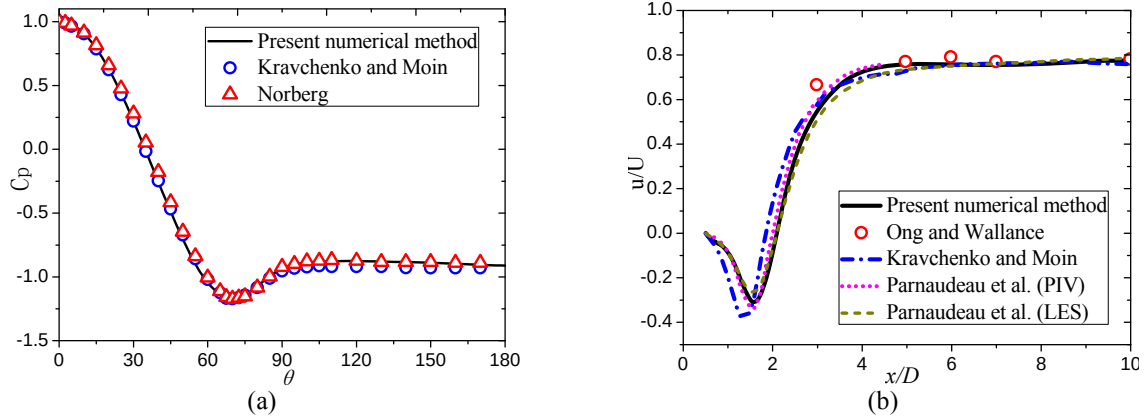


Fig. 3(a) Pressure coefficient distribution on the circular cylinder surface in non-shear flow ($Re=4,200$ in Norberg's experiment), (b) Mean streamwise velocity on the center line in the wake for the circular cylinder in non-shear flow

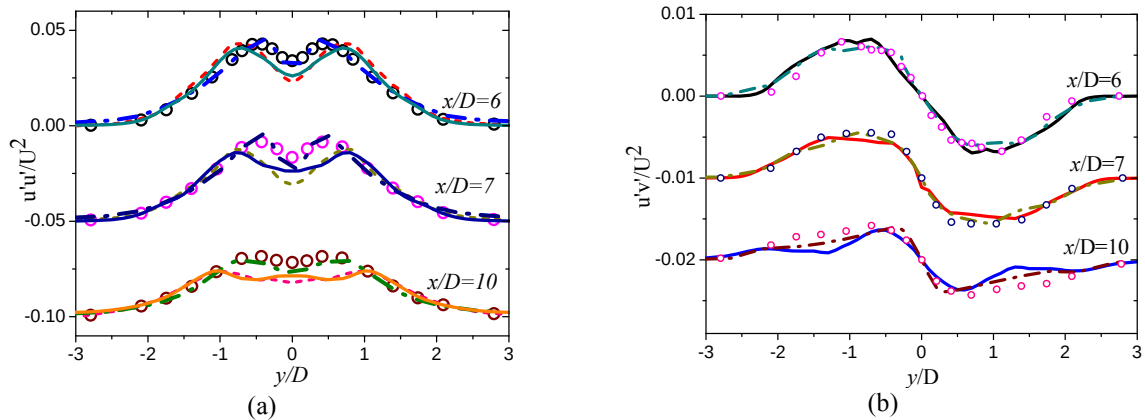


Fig.4 Comparison of the turbulence statistics in the wake of a circular cylinder at $Re=3900$, (a) Streamwise velocity fluctuations at three locations; (b) Reynolds shear stress at three locations. (—, Present LES; - - -, LES of Kravchenko and Moin, 2000; - . - , LES of Mittal and Moin, 1997; ○, experiment of Ong and Wallace, 1996)

4 NUMERICAL RESULTS AND DISCUSSIONS

4.1 Vortex shedding

The Strouhal number $St=fD/U$ is calculated from the FFT analysis of the lift coefficient for the flow around the 5:1 rectangular cylinder at non-shear and shear flows at $Re=22,000$, where f is the dominant vortex shedding frequency. The result obtained in present study are $St=0.118$ at $\beta=0$, $St=0.117$ at $\beta=0.05$ and $St=0.118$ at $\beta=0.1$, which is almost unchanged with shear param-

ter. In addition, the Strouhal number obtained in non-shear flow is compared with other studies at Fig. 5, which exhibits the variation of the Strouhal number with side ratio. The present result $St=0.118$ at non-shear flow is very close to the numerical results of Yu and Kareem (1996) with $St=0.114$, Bruno et al. (2010) with $St=0.115$, and the experimental result of Schewe (2009) with $St=0.111$. In addition, the result that Strouhal number does not vary significantly with shear parameter were also observed at other studies of Cao et al. (2010) on a circular cylinder and Cao et al. (2012) on a square cylinder. It seems reasonably to conclude that the shear parameter does not bring significant change to the Strouhal number although it creates asymmetry into the wake structure as shown later.

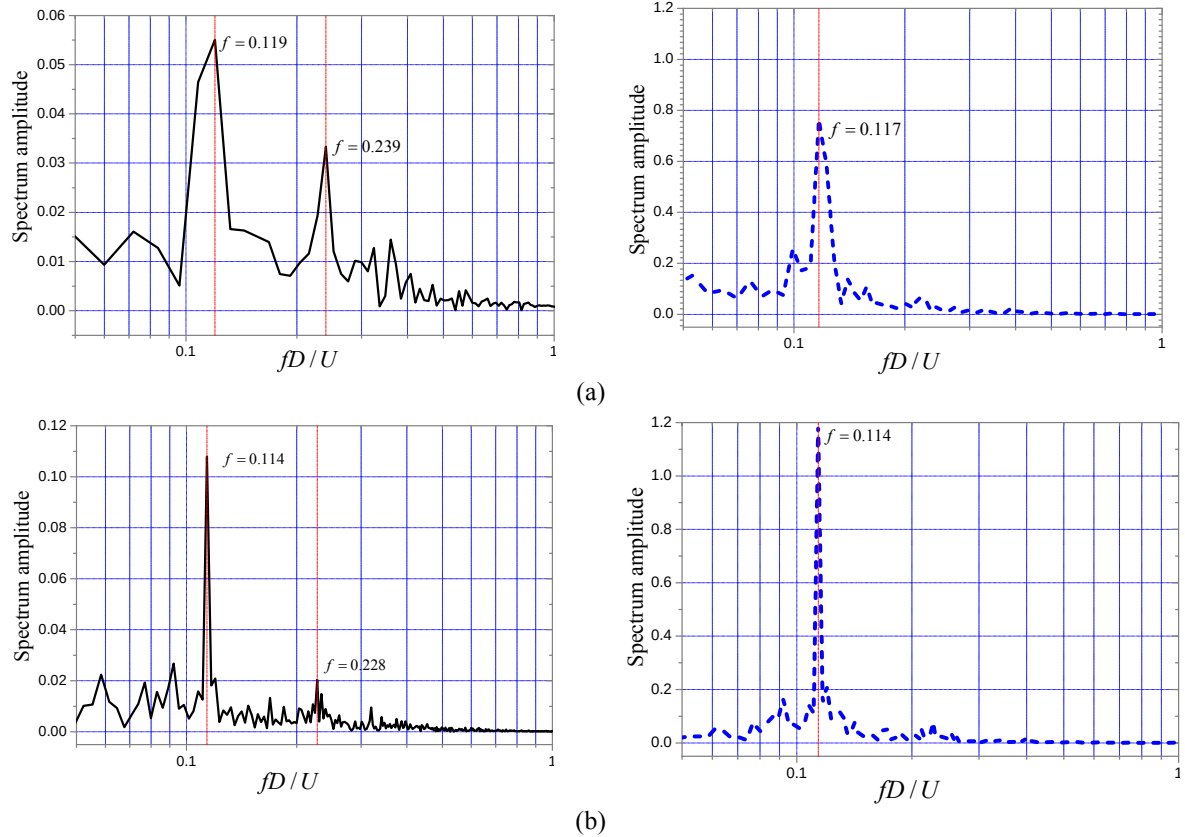


Fig. 6 Spectra of drag and lift coefficients at (a): $\beta=0.05$ and (b): $\beta=0.1$ (left: drag; right: lift)

Fig. 6 compares the power spectra of the instantaneous drag and lift coefficients at different shear parameters, $\beta=0.05$ and 0.1 at $Re=22,000$ for the 5:1 rectangular cylinder. The solid line shows the spectrum of drag coefficient C_D and the broken line shows that of lift coefficient C_L . Although the fluctuating lift coefficient still has one harmonic, a subharmonic has entered into the fluctuation of C_D . The strength of the subharmonic increases as the shear rate increases and it becomes dominant, resulting in the peak fluctuation frequencies of C_L and C_D becoming identical at $\beta=0.1$. A similar phenomenon was also reported in the case of a square cylinder (Cao et al. 2012).

Fig. 7 compares the instantaneous iso-vorticity surface of primary Karman vortex between no-shear and shear flows, where dashed and solid lines represents clockwise and counterclockwise vortices respectively. All the figures correspond to the moment when the lift coefficient is maximum. The vortices appears alternatively in the wake when $\beta=0$. However, with an increase in shear parameter, the counterclockwise vortices on the low-velocity side become weaker, and disappear in the far wake at $\beta=0.1$. The Karman Vortex Street is broken as result of the shear ef-

fect, but the counterclockwise vortex on the low-velocity still exists at the near wake and vortex shedding never disappears. This complicated phenomenon is also noticed by Saha et al (1999) from their two dimensional simulations on a square cylinder and Cao et al (2010, 2012) on a circular cylinder and a square cylinder. Therefore, it can be concluded that similar wake flow pattern exists for circular, square and rectangular cylinders in shear flows.

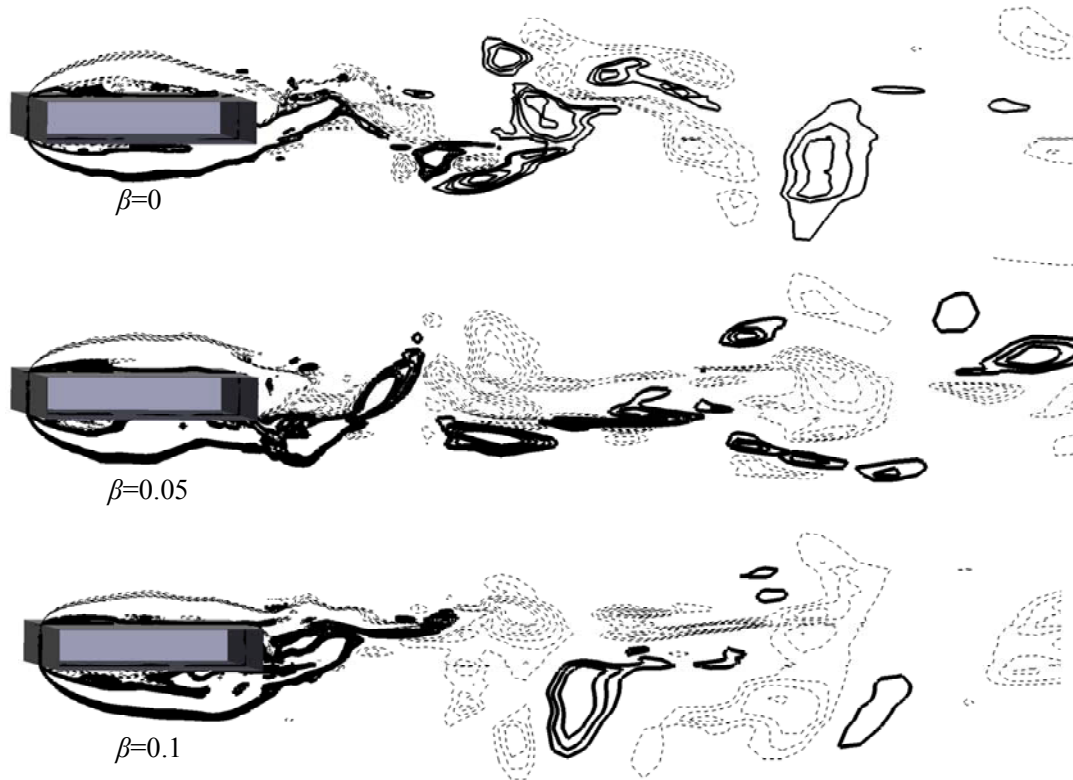


Fig. 7 Instantaneous wake structure with different shear parameters, $\omega_z = \pm 2$

4.2 Flow structures and Aerodynamic Forces

The time-averaged streamlines of flow field around the rectangular cylinder with different shear parameters are compared in Fig. 8. As in the non-shear flow, the first vortex (the main vortex) and the second vortex (the bubble) can be observed both on the upper and lower sides of the rectangular cylinder in shear flow. However, they are no longer symmetric as in non-shear flow. With increase of shear parameter, the center position of the first vortex moves upstream on high-velocity side and downstream on low-velocity side respectively. Meanwhile, the first vortex on the high-velocity side becomes thinner while its counterpart on the low-velocity side becomes thicker. The mean recirculation region was not formed after the trailing edge of the cylinder at $\beta=0.1$. These complicated flow phenomena influence the aerodynamic forces on the 5:1 rectangular cylinder directly.

Fig. 9 compares the mean friction coefficient distribution around the rectangular cylinder between in shear and non-shear flows. The distribution of mean friction coefficient distribution becomes asymmetrical in the shear flow. With the increase in shear parameter, the mean reattachment length L_r on the low-velocity side increases from $L_r/D = 4.632$ ($\beta=0$) to $L_r/D = 4.850$ ($\beta=0.05$) and $L_r/D = 4.868$ ($\beta=0.1$), while it decreases from $L_r/D = 4.632$ ($\beta=0$) to $L_r/D = 4.469$ ($\beta=0.05$) and $L_r/D = 4.073$ ($\beta=0.1$), which implies that the flow

will reattach more steadily on the high-velocity side and separate more steadily on the low-velocity side when the shear parameter is large enough.

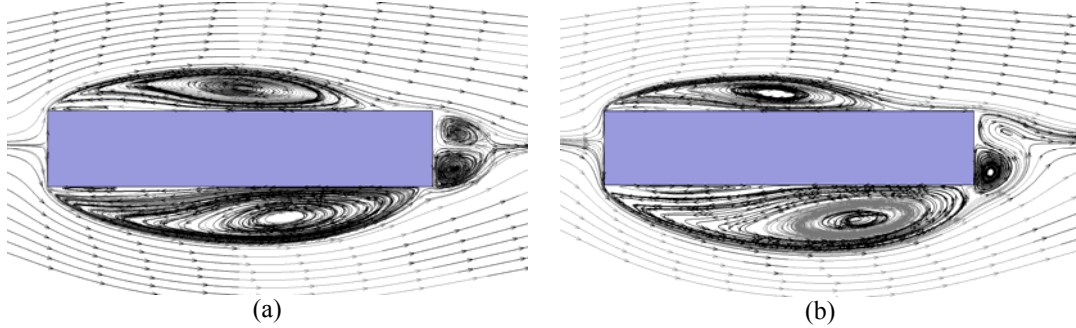


Fig. 8 Streamlines of the time-averaged flow field around the cylinder, (a) $\beta=0.05$; (b) $\beta=0.1$

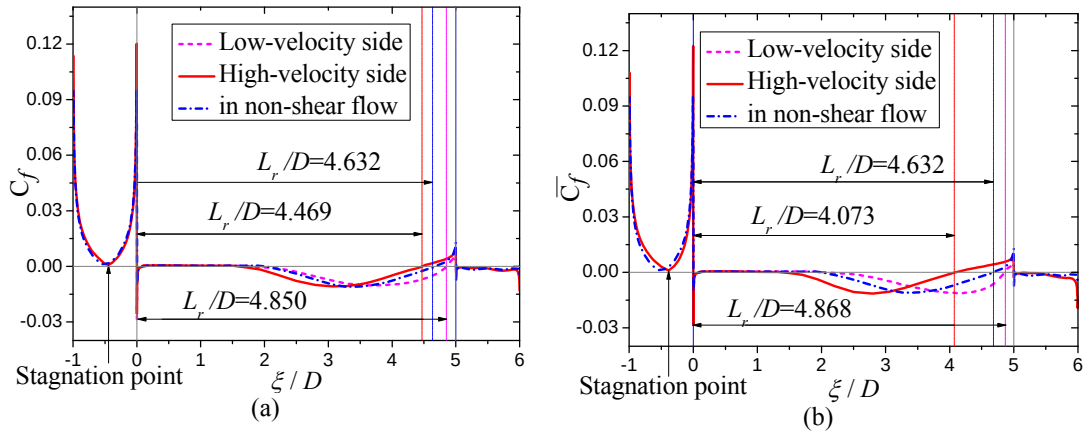


Fig. 9 Comparison on the distribution of the mean friction coefficient in shear flow and non-shear flow; (a) $\beta=0.05$; (b) $\beta=0.1$

Fig. 10 compares the mean pressure coefficient distribution around the rectangular cylinder surface in non-shear and shear flows. By the comparison in the region of $-1 < \xi/D < 0$ (the leading surface), it can be evidently observed that the stagnation point moves to the high-velocity side in shear flows.

As shown in Fig. 10, the value of mean pressure coefficients on the upper and lower sides in shear flows is almost the same in the region of $0 < \xi/D < 2.5$, in which the second vortex (the separate bubble) is formed. Thus it implies that the shear effects have little influence on determining the strength of separate bubble. However, for the region of $2.5 < \xi/D < 5$ where the first vortex (the main vortex) was formed, a significant deviation exists in the mean pressure distributions on high- and low-velocity sides as a result of the shear effects. Moreover, this deviation increases with the increase in shear parameter. The reason for this deviation is that the main vortex is quite different on the high- and low-velocity sides in shear flows as mentioned previously, and it suggests that the main role of shear effect plays on the large scale vortex.

For the region of $5 < \xi/D < 6$ (the trailing side), the mean base pressure coefficient increases with increase in shear parameter. This is similar to the finding of Onitsuka et al. (2000) in experimental study of the shear flow around rectangular cylinders with side ratio from 0.1 to 3.0.

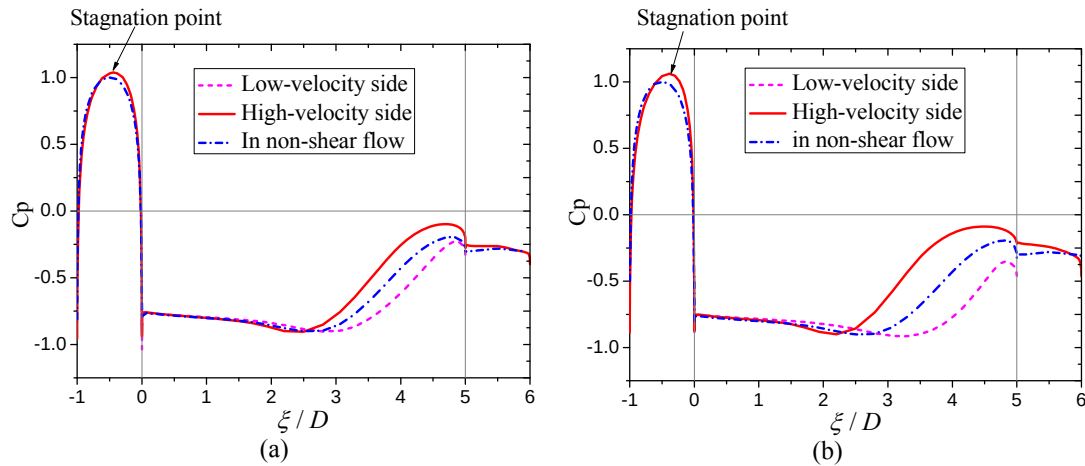


Fig. 10 Comparison of mean pressure coefficient over the rectangular cylinder surface in non-shear and shear flows, (a) $\beta=0.05$; (b) $\beta=0.1$.

Table 1 Comparison of mean and standard deviation values of aerodynamic force coefficients (normalized with respect to the body thickness D) with different shear parameters.

β	C_D	C_D'	C_L	C_L'	C_M	C_M'	T^*
0	1.100	0.082	--	0.890	--	0.773	279.5
0.05	1.128	0.090	-0.480	0.927	-0.717	0.817	213.4
0.1	1.142	0.111	-1.119	1.011	-1.405	0.867	262.1

As shown in Table 1, the direction of the lift force on a rectangular cylinder is the same to that on a circular cylinder, but opposite to that on a square cylinder. In the case of a circular cylinder, the movement of stagnation point generates a lift force from the high-velocity side to low-velocity side, while the velocity difference of the two sides has a contrary effect. The studies of Cao et al. (2007, 2010) showed that the contribution of the movement of stagnation point is dominant, and thus the lift force on a circular cylinder acts from the high-velocity side to low-velocity side. For the case of a square cylinder, the flow separates completely from leading side without flow reattachment, and the movement of stagnation point does not contribute to the lift force as shown in the simulation of Cao et al. (2012). Thus lift force direction is determined by the velocity difference and accompanying vortices with different strengths on two sides. For a rectangular cylinder with a side ratio $B/D=5$, the movement of stagnation point still has little influence on determining the direction of lift force. However for this kind of rectangular cylinder, the different movements of mean reattachment points on high- and low-velocity sides result in mean pressure distributions differing on two sides as shown in Fig. 15. Thus the lift force of a 5:1 rectangular cylinder acts from the high-velocity side to low-velocity side in shear flow.

5 CONCLUSIONS

In this paper, we carried out numerical simulations to investigate the aerodynamic characteristics of a 5:1 rectangular cylinder in shear flows at Reynolds numbers $Re=22,000$. The vortex shedding and aerodynamic forces accompanying with the complicated flow structures are studied.

The Strouhal number was found to have no significant variation in the shear parameter range of 0-0.1 at $Re=22,000$, which is similar to that in the case of a square cylinder and a circular cylinder. With increasing of shear parameter, the peak frequency of the drag coefficient becomes more identical to that of the lift coefficient due to the change of vortices forming behind

the back side. Although the Karmen vortex is not completely suppressed in the investigated ranges of shear parameter at $Re=22,000$, the Karmen vortex street in the wake is broken as a result of shear effects.

The intermittently-reattached flow becomes more stably-reattached and results in that the mean reattachment point moves upstream on the high-velocity side in shear flow, whereas it becomes more stably-separated on the low-velocity side. The stagnation point moves to the high-velocity side in shear flow, and the stagnation angle increases almost linearly with increasing of shear parameter.

Due to the movements of mean reattachment point, the mean pressure coefficient distribution over the rectangular cylinder surface is no longer symmetric. In the investigated range of shear parameter at $Re=22,000$, the mean value of drag force was observed to have unchanged while its standard-deviation increases with an increase in shear parameter. The mean lift force is no longer closing to zero resulting from the difference of the strength and location of the main vortices on two sides. Unlike the case of a square cylinder in a shear flow, the lift acts from high-velocity side to the low-velocity side for a 5:1 rectangular cylinder which is similar to that of a circular cylinder. However, the physical mechanism of the lift force direction of the rectangular cylinder, in which the movement of the mean reattachment point plays a dominating role to the lift force and the separation point is fixed, differs from that of the circular cylinder.

6 ACKNOWLEDGMENTS

This research was funded in part by Natural Science Foundation of China (NSFC) Grant No. 50978202 and Shanghai Pujiang Program number10PJ1409700.

7 REFERENCES

- 1 G. Bartoli, and M. Righi, Flutter Mechanism for Rectangular Prisms in Smooth and Turbulent Flow, *Journal of Wind Engineering and Industrial Aerodynamics*, **94** (2006), 275-91.
- 2 L. Bruno, D. Fransos, N. Coste, and A. Bosco, 3d Flow around a Rectangular Cylinder: A Computational Study, *Journal of Wind Engineering and Industrial Aerodynamics*, **98** (2010), 263-76.
- 3 S. Cao, S. Ozono, K. Hirano, and Y. Tamura, Vortex Shedding and Aerodynamic Forces on a Circular Cylinder in Linear Shear Flow at Subcritical Reynolds Number, *Journal of Fluids and Structures*, **23** (2007), 703-14.
- 4 S. Y. Cao, Y. J. Ge, and Y. Tamura, Shear Effects on Flow Past a Square Cylinder at Moderate Reynolds Numbers, *Journal of Engineering Mechanics-Asce*, **138** (2012), 116-23.
- 5 M. Kiya, H. Tamura, and M. Arie, Vortex Shedding from a Circular Cylinder in Moderate Reynolds Number Shear Flow, *Journal of Fluid Mechanics*, **101** (1980), 721-35.
- 6 A. G. Kravchenko, and P. Moin, Numerical Studies of Flow over a Circular Cylinder at $Re-D=3900$, *Physics of Fluids*, **12** (2000), 403-17.
- 7 B.P. Leonard, The Ultimate Conservative Difference Scheme Applied to Unsteady One-Dimensional Advection, *Computer Methods in Applied Mechanics and Engineering*, **88** (1991), 17-74.
- 8 C. Mannini, A. Soda, and G. Schewe, Numerical Investigation on the Three-Dimensional Unsteady Flow Past a 5:1 Rectangular Cylinder, *Journal of Wind Engineering and Industrial Aerodynamics*, **99** (2011), 469-82.
- 9 C. Mannini, Unsteady Rans Modelling of Flow Past a Rectangular Cylinder: Investigation of Reynolds Number Effects, *Computers & Fluids*, **39** (2010), 1609-24.
- 10 M. Matsumoto, Aerodynamic Damping of Prisms, *Journal of Wind Engineering and Industrial Aerodynamics*, **59** (1996), 159-75.
- 11 M. Matsumoto, H. Shirato, K. Araki, T. Haramura, and T. Hashimoto, Spanwise Coherence Characteristics of Surface Pressure Field on 2-D Bluff Bodies, *Journal of Wind Engineering and Industrial Aerodynamics*, **91** (2003), 155-63.
- 12 R. Mittal, and Moin P., Suitability of Upwind-Biased Finite Difference Schemes for Large-Eddy Simulation of Turbulent Flows, *AIAA Journal*, **35** (1997), 1415-17.

- 13 C. Norberg, An Experimental Investigation of the Flow around a Circular-Cylinder - Influence of Aspect Ratio, *Journal of Fluid Mechanics*, **258** (1994), 287-316.
- 14 C. Norberg, Flow around Rectangular Cylinders: Pressure Forces and Wake Frequencies, *Journal of Wind Engineering and Industrial Aerodynamics*, **49** (1993), 187-96.
- 15 A. Okajima, Strouhal Numbers of Rectangular Cylinders, *Journal of Fluid Mechanics* **123** (1980), 379-98.
- 16 L. Ong, and J. Wallace, The Velocity Field of the Turbulent Very near Wake of a Circular Cylinder, *Experiments in fluids*, **20** (1996), 441-53.
- 17 S. Onitsuka, Ozono, S., Cao, S., Wakasugi, Y., Flow around Rectangular Cylinders in Linear Shear Flow, in *The 16th Japanese National Wind Engineering Symp.* (Tokyo, 2000), pp. 279-84.
- 18 P. Parnaudeau, J. Carlier, D. Heitz, and E. Lamballais, Experimental and Numerical Studies of the Flow over a Circular Cylinder at Reynolds Number 3900, *Physics of Fluids*, **20** (2008).
- 19 S.V. Patankar, and D.B. Spalding, A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows, *Journal of Heat and Mass Transfer*, **15** (1972), 1787-806.
- 20 G. Schewe, Reynolds-Number-Effects in Flow around a Rectangular Cylinder with Aspect Ratio 1:5, in *Proceedings of the Fifth European and African Conference on Wind Engineering*, ed. by C. Borri, Augusti, G., Bartoli, G., Facchini, L. (Florence, Italy.: Firenze University Press, 2009).
- 21 K. Shimada, and T. Ishihara, Application of a Modified K-Epsilon Model to the Prediction of Aerodynamic Characteristics of Rectangular Cross-Section Cylinders, *Journal of Fluids and Structures*, **16** (2002), 465-85.
- 22 J. Smagorinsky, General Circulation Experiments with the Primitive Equations, *Monthly Weather Review*, **91** (1963), 99-164.
- 23 A. Sohankar, L. Davidson, and C. Norberg, Large Eddy Simulation of Flow Past a Square Cylinder: Comparison of Different Subgrid Scale Models, *Journal of Fluids Engineering-Transactions of the Asme*, **122** (2000), 39-47.
- 24 D. Sumner, and O. O. Akosile, On Uniform Planar Shear Flow around a Circular Cylinder at Subcritical Reynolds Number, *Journal of Fluids and Structures*, **18** (2003), 441-54.
- 25 T. Tamura, and Y. Ito, Aerodynamic Characteristics and Flow Structures around a Rectangular Cylinder with a Section of Various Depth/Breadth Ratios, *Journal of Structural and Construction Engineering* **486** (1996), 153-62.
- 26 T. Tamura, T. Miyagi, and T. Kitagishi, Numerical Prediction of Unsteady Pressures on a Square Cylinder with Various Corner Shapes, *Journal of Wind Engineering and Industrial Aerodynamics*, **74-76** (1998), 531-42.
- 27 D. Yu, and A. Kareem, Two-Dimensional Simulation of Flow around Rectangular Prism, *Journal of Wind Engineering and Industrial Aerodynamics*, **62** (1996), 131-61.