VELOCITY DIFFERENCE CHANGES BETWEEN FLUID AND SAND PARTICLES IN BOUNDARY-LAYER AND VERY NEAR WAKE AREA

by

Sha SHA[a,b], Xiantang ZHANG[a,b*], ZhiAng WANG[b], Han LIU[b], and Huiyao ZHANG[b]

[a]Shandong Provincial Key Laboratory of Civil Engineering Disaster Prevention and Mitigation, Shandong University of Science and Technology, Qingdao, China
[b]College of Civil Engineering and Architecture, Shandong University of Science and Technology, Qingdao, China

Original scientific paper
https://doi.org/10.2298/TSCI2106217S

Fluent simulates the water-sand flow around a cylinder. Monitoring lines are set up at different positions in the cylindrical surface and the very near wake area behind the cylinder, in order to explore the speed difference of fluid and sand in the water-sand two-phase flow in the boundary-layer and the very near wake area. The results show that the sand particles stay for the longest time on the back of the cylindrical surface and in the very near wake area, and a small part of the sand particles are sticky on the back of the cylindrical pier. When the height of the cylinder is $z/D \in (1.57, 3.14)$, the turbulent flow on the cylindrical surface is fully developed. The dynamic pressure of the flow field in the very near wake area behind the cylinder fluctuates greatly, and the water-sand flow is extremely unstable. At the monitoring position of the cylinder, there is a sudden decrease in the velocity of the fluid, while the velocity of the sand particles changes little and remains finally at about –0.02 m/s. The water-sand flow field near the wall changes drastically, but the velocity change of sand particles has obvious hysteresis compared with fluid. When leaving the near-wall position but still in the cylindrical wake area ($x/D \leq 3$), the changes in the water-sand flow field are more intense and and the velocity of the sand particles is still slightly larger than the fluid velocity.

Keywords*: fluent, water-sand flow, boundary-layer, velocity difference*

Introduction

There are many rivers and multiple bridges in China. The water-sand flow around a cylinder is common. Therefore, the study of the solid-liquid two-phase flow around a cylinder has practical engineering significance. In view of this, many scholars have carried out extensive research on the problem of solid-liquid two-phase flow around a cylinder. Righetti and Romano [1] found that the closer to the wall, the interaction of the solid particles and the fluid will be more affected through experiment. Ni and Huang [2] studied the motion characteristics of non-cohesive particles in the water flow by changing the particle concentration. Gore

*Corresponding author, e-mail: zzxhtm@sdust.edu.cn

and Crowe [3] found that the particle size would affect the turbulence intensity of fluid. Small size particles inhibited fluid turbulence intensity and the large one enhanced the intensity. In the same incoming flow velocity, Zhao *et al*. [4] used particle image velocimetry technology to carry out the influence of particles in the flow field on the turbulence and the law of interaction between particles and the fluid. Lei and Tan [5] obtained the 2-D flow characteristics near the wall in the wake area of the flow around the cylinder under different Reynolds numbers by Transition SST model. Cui *et al*. [6] found that the cross-sectional form of the submerged part of the composite bridge pier had a great influence on the average drag coefficient under different water depths. Zhang *et al*. [7] found that the free end of the finite-length cylinder had a significant effect on the wake area. The boundary-layer separation of the finite-length cylinder was unstable. Duan and Wan [8] used LES to analyzed the instantaneous velocity in the wake region behind the cylinders with four different slenderness ratios. It was concluded that the instantaneous velocity profile always surrounded the time-average velocity. Huang and Wu [9] found that the particle diffusion distribution in the wake was closely related to the Strouhal number of the particles and the structure of the wake vortex based on the discrete vortex method. Qiao *et al*. [10] studied the influence of mixed flow under different sand content on the force characteristics of cylinders with different surface roughness.

It can be seen that in the study of two-phase flow around a cylinder, most of the two-phase flow is studied as a mixed flow, or only as the particle flow in the flow field. In reality, when the mixed flow passes around the cylinder, there will be differences in the velocity of fluid and sand particles in the boundary-layer of the cylindrical wall and the wake region behind the cylinder [11]. In particular, the separation of the boundary-layer on the wall often causes a large increase in resistance and fluid kinetic energy loss, while the presence of sand particles in the flow field can regulate the separation of the boundary-layer, and enhance fluid pulsation and the transportation capacity of energy and momentum in the flow process [4, 12]. Therefore, if the velocity changes of the fluid and sand particles in the boundary-layer and wake area can be accurately estimated, it is of great significance to effectively control the boundary-layer flow in the project.

At present, most of the numerical simulation research on the boundary-layer is a 2-D model, which ignores the velocity change of the mixed water-sand flow on the wall surface of the cylindrical bridge pier in the span direction. Therefore, based on the common typical high Reynolds number Re = 1×10^6, a 3-D model is made to simulate the single cylinder of water-sand flow field by setting monitoring positions on the cylindrical surface and the very near wake area behind the cylinder, to explore the difference in velocity changes of fluid and sand particles in the boundary-layer and in the very near wake region.

Turbulence model and numerical model

Turbulence model

The movement of the fluid in the boundary-layer is affected by the normal velocity gradient. Therefore, RANS SST k-ω turbulence model based on turbulence theory [13] is chosen to solve the turbulent flow field. Among them, the SST k-ω turbulence model adopts the k-ω model in the near-wall region. The k-ω model can be expressed as the transport equation of turbulent kinetic energy, k, and specific dissipation rate, ω:

$$\rho \frac{\mathrm{d}k}{\mathrm{d}t} + \rho \frac{\partial (k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(T_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k \tag{1}$$

$$\rho\frac{\mathrm{d}\omega}{\mathrm{d}t}+\rho\frac{\partial(\omega u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(T_\omega\frac{\partial\omega}{\partial x_j}\right)+G_\omega-Y_\omega+D_\omega \tag{2}$$

where G_k is the turbulent kinetic energy generated by the laminar flow velocity gradient, G_ω – the turbulent kinetic energy generated by the ω equation, T_k and T_ω are the diffusion rate of k and ω, Y_k and Y_ω are the turbulence generated by diffusion, and D_ω – the orthogonal divergence.

Numerical model

The 3-D calculation model [14-17] established by ANSYS Workbench takes the cylinder diameter, D, as the characteristic scale, and the specific dimensions are shown in fig. 1. The origin of the co-ordinates is at the center of the bottom surface of the cylindrical pier, and the *x*-, *y*-, and *z*-directions are positive.

Setting the initial and boundary conditions [14-17] according to the actual situation:

– The entrance is the velocity inlet, the steady velocity of the water-sand two-phase flow U_0 = 1 m/s, water flows vertically into the velocity inlet [18], and the exit is free outflow.
– The left and right sides and upper and lower walls of the calculation domain are symmetrical boundaries, and the surface of the cylindrical bridge pier is a non-slip wall.
– The density of water $\rho = 1 \times 10^3$ kg/m^3, the dynamic viscosity coefficient $\mu = 0.001003$, the diameter of the cylinder D = 1.0 m, and the Reynolds number Re = 1.0×10^6.
– The particle size is uniformly set to 5×10^{-4} m, it belongs to sand according to the silt particle classification standard.
– The density of sand is 2500 kg/m^3, and the average sand content is 1.6 kg/m^3, which belongs to the river with large sand content [19].

The ICEM is used to mesh the 3-D model, and fig. 2 shows the dense grid division near the cylinder.

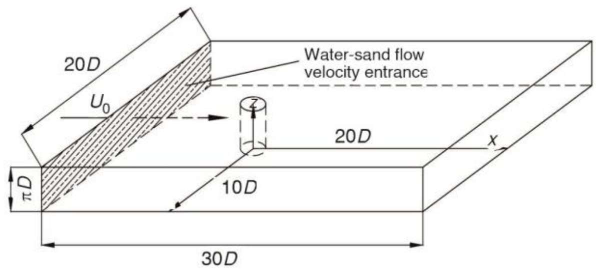

Figure 1. Model of the computational domain

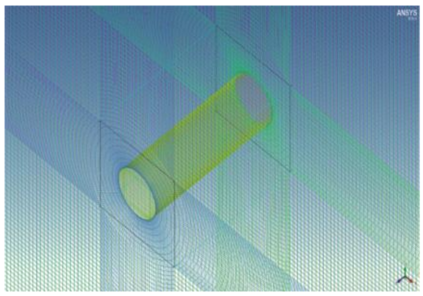

Figure 2. The grid division diagram

Combining the symmetry of the cylinder and the transition position of the boundary-layer, the monitoring position of the cylinder is shown in fig. 3(a). The monitoring position in the very near wake area ($x/D \leq 3$) [20] is shown in fig. 3.

Result analysis and discussion

Fluid and sand velocity changes at different monitoring positions of the cylindrical surface

As shown in fig. 4(a), the sand particles stay for a long time on the back of the cylindrical surface. Figure 4(b) shows that a small part of sand particles mainly sticks to the side and back of the cylindrical surface. This phenomenon is consistent with the argument that particles moving near the wall are easily deposited on the wall due to the Saffman force mentioned in [11].

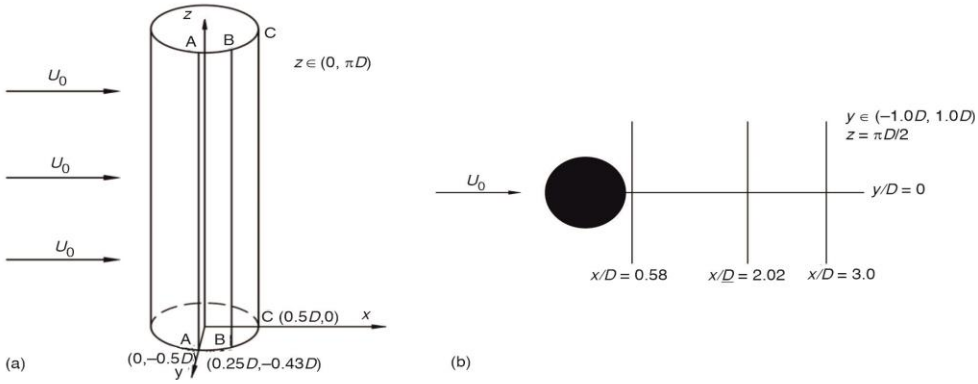

Figure 3. Schematic diagram of the monitoring line positions; (a) monitoring position of the cylinder and (b) monitoring location of very near wake area

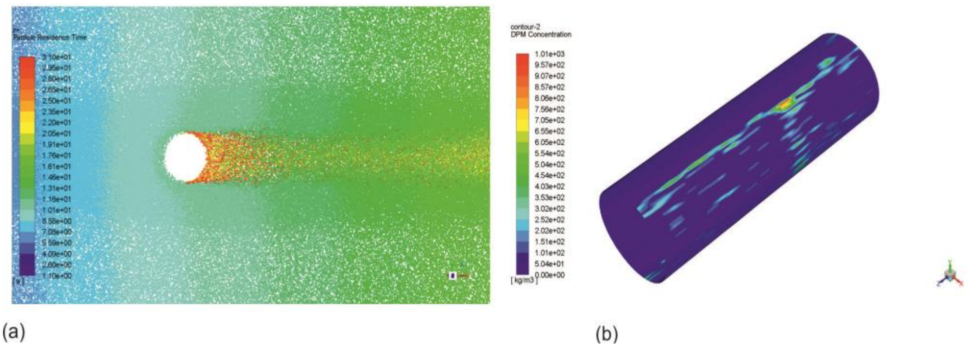

Figure 4. Distribution of sand particles in the cylinder; (a) sand particles residence time distribution and (b) the distribution of sand particles sticking to the wall

Analyzing the output discrete phase model (DPM) file data after the FLUENT transient calculation is completed, and the velocity changes of fluid and sand particles at different monitoring positions are obtained as shown in fig. 5.

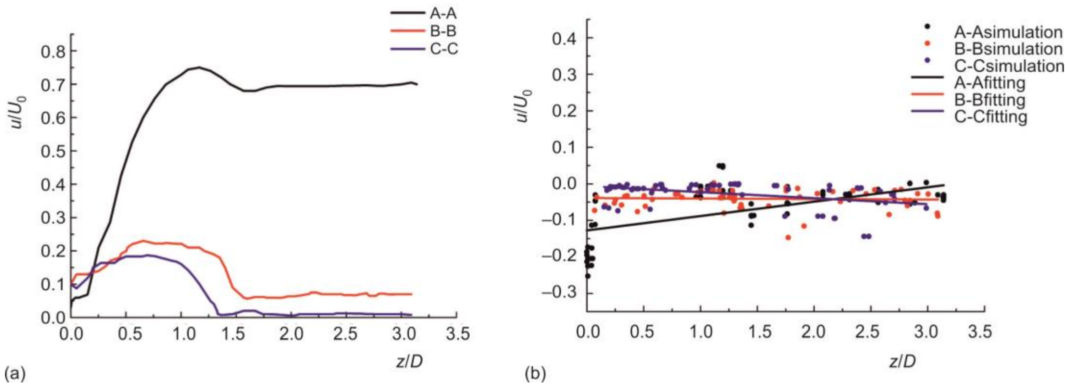

Figure 5. Velocity curves of fluid and sand particles at different monitoring positions of the cylinder; (a) fluid velocity changes and (b) sand velocity changes

From bottom to top along the cylinder A-A, the fluid velocity increases to 0.75 m/s, then decreases to 0.68 m/s and finally stabilizes at 0.70 m/s. From bottom to top along the cylinder B-B, the fluid velocity first increases to 0.23 m/s, then decreases to 0.06 m/s and finally stabilizes at 0.07 m/s. From bottom to top along the cylinder C-C, the fluid velocity increases to 0.19 m/s, then decreases to 0.01 m/s and remains stable. It can be seen that the fluid velocity tends to be stable from the position of the cylinder $z/D = 1.57$, and the turbulent flow on the cylindrical bridge pier surface is fully developed when $z/D \in (1.57, 3.14)$. During the mixed water-sand flow flows from A-A to B-B, the larger velocity gradient [21] in the boundary-layer makes the fluid velocity drop from 0.70 m/s to 0.07 m/s in a shorter path.

Due to the existence of negative pressure, the velocity of the sand particles is relatively unstable along the cylinder A-A from bottom to top. When the water-sand flow flows from the cylinder B-B to C-C, the sand velocity gradually stabilizes at about –0.02 m/s, and a small part of the sand moving near the wall is stuck on the cylindrical surface by the Saffman force.

Fluid and sand particles velocity changes at different monitoring positions in the very near wake area behind the cylinder

From the fluid velocity change curve at different monitoring positions of the cylinder, it is found that when $z/D \in (1.57, 3.14)$, the fluid velocity remains stable and the turbulent flow is fully developed on the cylindrical surface. Therefore, this article analyzes the flow field plane at $z/D = 1.57$:

– Figure 6(a) shows that the mixed water-sand flow velocity in the very near wake area basically does not exceed 0.2 m/s.
– Figure 6(b) shows that at $z/D = 1.57$, $y/D = 0$, the dynamic pressure fluctuates relatively large in the very near wake area $0.5 \le x/D \le 3$.

It shows that the movement of mixed water-sand flow is extremely unstable by the large-scale vortices in the very near wake area of the cylindrical bridge.

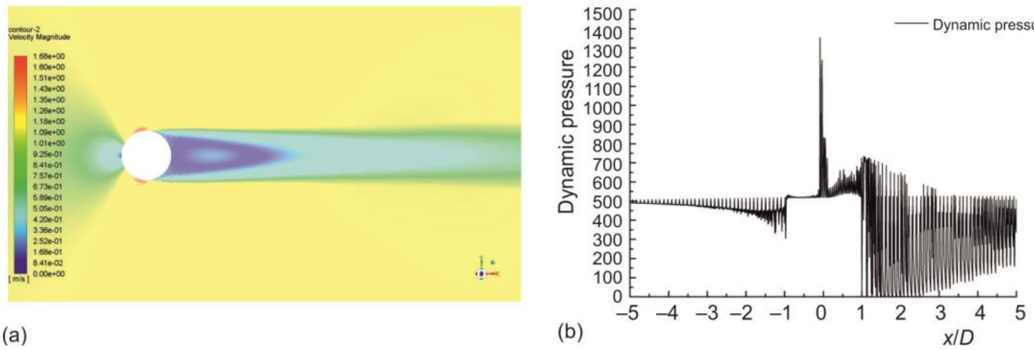

Figure 6. Velocity and dynamic pressure diagram of the flow field plane at *z/D* = 1.57; (a) velocity distribution of mixed water-sand flow and (b) dynamic pressure diagram

According to the analysis in fig. 6, this paper extracts the DPM data of three locations at $z/D = 1.57$, $y/D \in (–1, 1)$ in the very near wake area to analysis. The velocity changes of fluid and sand particles at different monitoring positions in the very near wake area are shown in fig. 7.

When $x/D = 0.58$, the dynamic pressure begins to fluctuate, and the flow field changes begins to become violent. At $y/D \in (–0.5, –0.4)$ and $y/D \in (0.4, 0.5)$, although both

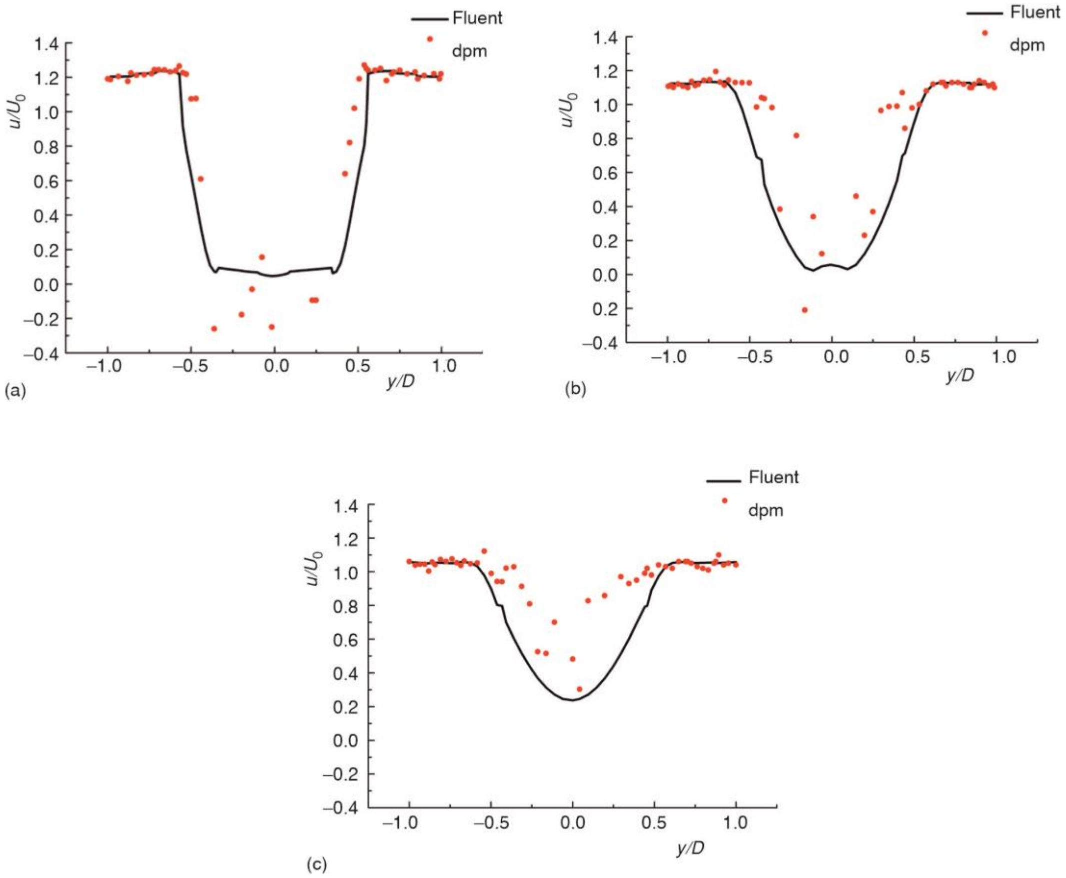

Figure 7. Velocity distribution curves of sand particles and fluids at different positions in the very near wake area; (a) *x/D* = 0.58, (b) *x/D* = 2.02, and (c) *x/D* = 3.0

the velocity of fluid and sand drop from 1.22 m/s due to the obstruction of the cylindrical bridge piers, the velocity of the sand particles is always slightly greater than that of the fluid. It can be seen that the velocity change of sand particles has an obvious hysteresis than that of the fluid. In the same way, the fluid velocity basically drops to zero at $y/D \in (-0.4, 0.4)$, but the post-cylinder sand velocity is not zero due to the negative pressure. At the same time, the Saffman force also helps the sand particles moving near the wall stick to the wall.

When $x/D = 2.02$, the dynamic pressure fluctuates greatly and the flow field changes relatively sharply. Although the change of fluid velocity decreases rapidly from 1.12 m/s in the area $y/D \in (-0.5, -0.15)$ and $y/D \in (0.15, 0.5)$, the area where the fluid velocity is basically zero has been reduced to (–0.15, 0.15), and the sand velocity is still slightly larger than the fluid velocity. However, the sand movement is dominated by inertial force at this time. It can be seen that the movement of sand particles responds slowly to changes in turbulence, and is greatly affected by the impact on the wall of the cylindrical bridge pier and the velocity is greater than that of the fluid.

When $x/D = 3.0$, the water-sand flow is far away from the cylinder but still in the very near wake area. The minimum velocity of the fluid behind the cylinder is no longer zero and has a tendency to gradually increase at this time. In the area affected by the diameter of the cylinder $y/D \in (-0.5, 0.5)$, the velocity of sand particles is still slightly larger than that of fluid. It can be seen that the movement of sand particles is still mainly controlled by inertia at this time and responds to turbulence changes slowly. The sand particles are greatly affected by the impact on the wall.

Conclusion

By analyzing the velocity changes of fluid and sand particles at different monitoring positions on the cylindrical surface and in the very near wake area behind the cylinder, the conclusions are obtained as follows.

- On the back of the cylindrical surface and in the very near wake area behind the cylinder, the sand particles stay for the longest time. A small part of the sand particles moving near the wall are stuck to the back of the cylindrical surface due to the Saffman force. In the process of flowing around the cylinder, the velocity of the water-sand flow at the cylinder $z/D = 1.57$ begins to remain stable. When the cylinder height is $z/D \in (1.57, 3.14)$, the turbulent flow on the cylindrical bridge pier surface is fully developed.
- The fluid on the cylindrical surface is subjected to viscous shear stress. Due to the large velocity gradient in the boundary-layer, the fluid velocity has a sudden drop to about 0.07 m/s during the movement of the fluid from A-A to B-B on the cylinder. The sand velocity on the back of the cylinder is basically stable at about –0.02 m/s. At this time, it is more likely to stick to the wall due to the Saffman force in the boundary-layer.
- In the very near wake area behind the cylindrical bridge pier, the mixed water-sand flow velocity does not exceed 0.02 m/s. The dynamic pressure fluctuates greatly due to the existence of large-scale vortices.
- In the very near wake area behind the cylindrical pier, the velocity of fluid and sand particles decreases rapidly in the area blocked by the cylindrical pier, but the velocity of the sand particles is always slightly greater than that of the fluid. The dynamic pressure of the water-sand flow in the wake area behind the cylinder fluctuates greatly and the flow is extremely unstable. The inertial force plays a role in leading the movement of sand particles in the very near wake area. Compared with the velocity change of the fluid, the velocity change of sand particles has obvious hysteresis. It can be seen that the movement of sand particles responds to turbulence changes slowly, and it is greatly affected by the impact on the wall.
- The study of the speed difference phenomenon of fluid and sand in the boundary-layer and the very near wake area has a certain significance for the effective control of boundary-layer flow in engineering. But there are still some shortcomings in this paper, such as temperature change can significantly affect the energy transmission in the boundary-layer, but this simulation is conducted at room temperature, and further research can be done on this.

Acknowledgment

This work was supported by the National Natural Science Foundation of China (51874189), Shandong Province Graduate Education Innovation Project (SDYY15086, SDYY15088).

References

[1] Righetti, M., Romano, G. P., Particle-Fluid Interactions in a Plane Near-Wall Turbulent Flow, *Journal of Fluid Mechanics, 505* (2004), Apr., pp. 93-121

[2] Ni, J. R., Huang, X. J. Some Aspects of Hyper-Concentrated Sediment-Laden Flows (in Chinese), *Journal of Hydraulic Engineering, 7* (2002), pp. 8-15
[3] Gore, R. A., Crowe, C. T., Effect of Particle Size on Modulating Turbulent Intensity. *International Journal of Multiphase Flow, 15* (1989), 2, pp. 279-285
[4] Zhao, H. L., *et al*., Experimental study of coherent structures in a solid-liquid turbulent boundary-layer (in Chinese), *Journal of Experiments in Fluid Mechanics, 31* (2017), 6, pp. 29-36
[5] Lei, J. M., Tan, Z. M., Numerical Simulation for Flow Around Circular Cylinder at high Reynolds Number based on Transition SST Model (in Chinese), *Journal of Beijing University of Aeronautics and Astronautics, 43* (2017), 2, pp. 207-217
[6] Cui, W. Z., *et al.*, Three-Dimensional Numerical Simulation of Flow Around Combined Pier based on Detached Eddy Simulation at High Reynolds Numbers, *International Journal of Heat and Technology, 35* (2017), 1, pp. 91-96
[7] Zhang, H., *et al*., Large-Eddy Simulation of the Flow Past Both Finite and Infinite Circular Cylinders at Re = 3900, *Journal of Hydrodynamics, 27* (2015), 2, pp. 195-203
[8] Duan, M. Y., Wan, D. C., Large Eddy Simulation of Flow Around the Cylinders with Different Aspects (in Chinese), *Chinese Journal of Hydrodynamics, 31* (2016), 3, pp. 295-302
[9] Huang, Y. D., Wu, W. Q., Numerical Study of Particle Distribution in the Wake of Liquid-Particle Flows Past a Circular Cylinder Using Discrete Vortex Method (in Chinese), *Applied Mathematics and Mechanics, 27* (2006), 4, pp. 477-483
[10] Qiao, Z. W., *et al.*, Effect of Roughness on the Force and Erosion of Cylindrical Structures Under Water-Sand Two Phase Flow (in Chinese), *Journal of Shangdong University of Science and Technology (Natural Science), 40* (2021), 3, pp. 44-49
[11] Liu, X. B., Cheng, L. J., A Numerical Simulation of the Motion of Solid Particles in Turbulent Boundary-Layers (in Chinese), *Shanghai Journal of Mechanics, 17* (1996), 1, pp. 54-60
[12] Chen, M. Z., *Fundamentals of Viscous Fluid Dynamics*, Higher Education Press, Beijing, China, 1993
[13] Menter, F. R., Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, *Aiaa Journal*, *32* (1994), 8, pp. 1558-1605
[14] Shen, Y.-B., *et al*., Effect of Roughness on the Flow of Parallel Double Cylinder Cylindrical Under Different Spacing Ratios (in Chinese), *Water Resources and Hydropower Engineering, 50* (2019), 07, pp. 131-136
[15] Moin, P., Verzicco, R., On the Suitability of Second-Order Accurate Discretizations for Turbulent Flow Simulations, *European Journal of Mechanics / B Fluids*, *55* (2016), Part 2, pp. 242-245
[16] Qiao, Y. L., *et al*., Analysis of Three-Dimensional Numerical Simulation Methods for Turbulent Flow Past Circular Cylinder (in Chinese), *Hydro-Science and Engineering*, (2016), 3, pp. 119-125
[17] Cui, W. Z., *et al*., Study on Influence Factors of Flow Around Twin Tandem-Circular Cylinders Under High Reynolds Number (in Chinese), *Water Resources and Hydropower Engineering, 49* (2018), 2, pp. 92-98
[18] Zhang, X. T., *et al*., Study of Correlation Between Lift and Drag Forces with Wave Attacking Height of Cross-Sea Bridge Based on FLUENT (in Chinese), *Journal of Shangdong University of Science and Technology (Natural Science), 32* (2013), 4, pp. 57-61
[19] Yu, W. C., Yue, H. Y., Position of Runoff and Sediment of Yangtze River in World Rivers (in Chinese), *Journal of Yangtze River Scientific Research Institute*, *19* (2002), 6, pp. 13-16
[20] Xia, M., Karniadakis, G., Dynamics and Low-Dimensionality of a Turbulent Near Wake, *J. Fluid Mech*., *410* (2000), May, pp. 29-65
[21] Ji, L. Y., Study on the Collision Rule and Erosion Characteristic of Solid-Liquid Two-Phase Flow with Large Particle Diameter, Ph. D. thesis, Zhejiang Sci-Tech University, Hangzhou, Zhejiang, China, 2017

Paper submitted: May 11, 2021
Paper revised: July 10, 2021
Paper accepted: July 13, 2021