The visualization of flow field around circular cylinders by Fluent standard k–\(\varepsilon\) turbulence model

J Wang\(^1\), S G Weng\(^{1,2}\), S S Wu\(^1\), X D Hu\(^1\) and X Yang\(^1\)

\(^1\)Jiangsu Hydraulic Research Institute, 97 Nanhu Road, Nanjing 210017, China

E-mail: wengsonggan@sina.com

Abstract. An array of four circular cylinders was designed with different layouts (different value of H/Z), to study the flow field characteristics between cylinders, and the shading coefficient between circular cylinders also was explored. The Fluent standard k–\(\varepsilon\) turbulence model was employed in the flow field visualization and drag coefficient. In this paper, it is shown that the shading coefficient between circular cylinders decreases, with the value of H/Z increasing. The drag coefficient of the front circular cylinders presents symmetrical distribution. The drag coefficients are between 0.98 to 1.10. The drag coefficient of the back circular cylinders presents the same characteristic, the drag coefficients are between 0.38 to 0.47. The formation of vortex, vortex shedding, the separation of flow are formed between cylinders with value of H/Z increasing. When H/Z=8, the phenomenon of vortex shedding turns up, and the shading effect of the front cylinders is very weak.

1. Introduction

Flows past circular cylinders, have been applied in various engineering design. In particularly, there are some complex phenomena that refer to reattachment points and multiple flow separation, unsteady vortex shedding, and turbulent structures. On account of the need of engineering design, the prediction of flow structures in flow fields must be efficiently and accurately. The hydrodynamics gives an example, that prediction of the pressure distribution and drag forces, wake structures as well as pollution dispersion, are considerable [1].

Flow visualization is a novelty method to study the fluid flow structures, and the flow structures can be observed and monitored. The major methods include adding a pollutant such as a smoke or a dye to the fluid. The low impact on the flow dynamics is the main advantage to flow visualization methods, which bases on mostly non-contact. Such advantageous also are used in harsh measurement conditions, where other measurement methods are few [2]. A few applications of flow visualization in various fields are described below.

The particle image velocimetry (PIV) system is used as a method of flow visualization in various applications, such as laminar flow behavior of water, flow field in a sphere-packed pipe, the flow fields in the high-speed water tunnel, and the acoustic cavitation patterns in the flow velocity and vorticity fields [3-6]. Hydrogen-bubble technique as a specific form of the particle image velocimetry (PIV), is applied in the internal-combustion engine intake port, mechanism of technique of flow field display, and the spray structures under the effect of hydrodynamic cavitation [7-9]. Laser induced fluorescence technique (LIF) is a high-efficiency method in studying the fluid mixing in a fuel rod bundle geometry [10-12] and the temperature distribution of the flow fields [13]. Computational fluid dynamics (CFD) as a direct, convenient method close to the actual situation always used to simulate...
the working flow field, and acquired the distribution of different sections of the particle velocity, particle volume fraction distribution, axial and radial velocity distribution, velocity distribution of the liquid, simulations were carried out for the rotation speed, the inlet velocity of the liquid, and the viscosity of the liquid all have a certain effect on the flow in annular space [14-16].

MCH River is a tributary of the Huaihe River, with the impregnation of Yangtze River. There are 9 long-span bridges across the MCH River. The effect of backwater is serious, due to the superimposed effect of bridge’s pile foundation.

To the point, we focus on flow visualization of flow around the circular cylinders, to study the flow field characteristics between cylinders, and the shading coefficient between circular cylinders also was explored. The framework of the paper is showed as follows. In Section 1, we introduce the development and status of the flow visualization. In Section 2, we present the numerical method and modeling. In Section 3, we discuss the results. We conclude this paper in Section 4 and provide future insights to our research.

2. Numerical method and modelling

In the paper, we utilize Ansys (16.0) Fluent, a commercial CFD solver, to solve the equations of motion in 2-D numerical analysis. An unsteady implicit coupled pressure has been employed, based double precision solver. The standard k-ε turbulence model is employed in flow variables of the circular cylinders, a bounded central difference discretization scheme and second order upwind-based discretization scheme. The cell-face pressures are calculated by the standard interpolation scheme [1].

2.1. Flow domain

The study of flow around four circular cylinders corresponds to an experiment performed in a water tunnel by Pietro Catalano [17]. The experiment conditions was Re = 10000 in this case. The computational domain is 150 (meter) × 60 (meter), showed in figure 1(a). The vertical spacing of circular cylinders is Z (Z=1, 2, 3,…), the transverse spacing of circular cylinders is H (H=1, 2, 3,…).

![Flow domain](image)

**Figure 1.** The cylinder computational domain dimensions (meter). (a) The computational domain (meter) and (b) The circular cylinders layout (meter).

2.2. Grid generation

![Grid generation](image)
A grid generator ICEM CFD of the Navier-Stokes Equations code Ansys Fluent was used for meshing the system with unstructured grid as shown in figure 2.

2.3. Governing equation

In the study, it is assumed that the fluid is incompressible and Newtonian with temperature-dependent fluid properties.

Continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0$$  \hspace{1cm} (1)

Momentum equation:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = g_i - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$  \hspace{1cm} (2)

The standard k–ε turbulence model is a major method applied in engineering process calculation. It possesses the advantage of wide range, economics, and reasonable accuracy. The model is semi-rational relation, based on abundant experimental phenomena. It can be used in simulating the flow around circular cylinders in the engineering application.

Standard k–ε turbulence model equation:

$$\rho \frac{dk}{dt} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M$$

$$\rho \frac{d\varepsilon}{dt} = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}$$  \hspace{1cm} (3)

Where $u_i$ denotes Cartesian coordinate velocity component; $x_i$ denotes Cartesian coordinate; $p$ denotes intensity of pressure; $\rho$ and $\mu$ respectively denote density and coefficient of dynamic viscosity of liquid; $G_k$ denotes the turbulent kinetic energy generated by the mean velocity gradient, $G_b$ denotes the turbulent kinetic energy generated by buoyancy, $Y_M$ denotes the effect of compressible turbulent fluctuation on the total dissipation rate, $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ are constant, $\sigma_k$, $\sigma_\varepsilon$ denote turbulence’s prandtl number of equation k and $\varepsilon$, $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, $C_{3\varepsilon}$=0.09, $\sigma_k$=1.0, $\sigma_\varepsilon$=1.3 [1].
3. Results and discussion

The flow over circular cylinders is investigated in this section. The results obtained by using standard k-ε turbulence model are compared with different circular cylinders layouts (Z denotes the vertical spacing of circular cylinders, Z=1, 2, 3,…; H denotes the transverse spacing of circular cylinders, H=1, 2, 3,…). Moreover, the flow characteristics are analyzed using current standard k-ε turbulence model.

![Flow around circular cylinders](image1.png)

**Figure 3.** The velocity vectors showing the flow around the circular cylinders (m/s). (a) H/Z=1, (b) H/Z=2, (c) H/Z=3, (d) H/Z=4, (e) H/Z=5, (f) H/Z=6, (g) H/Z=7 and (h) H/Z=8.

Figure 3 shows vectors colored by flow velocity around the circular cylinders. The drag coefficients calculate based on the drag of single pile in same water channel. From the figure, it can be seen that, the vortices are formed in the region behind the cylinders with pronounced separation in the cylinder wake region and the flow is highly complex. When passing the cylinder, the velocity of flow is decelerated in the front. And then, the velocity of the flow is accelerated at the cylinder tips, producing jet flow. The flow begins to separate behind the cylinder causing vortex shedding with the
value of $H/Z$ increasing, which is an unsteady phenomenon ($Z$ denotes the vertical spacing of circular cylinders, $H$ denotes the transverse spacing of circular cylinders). When $H/Z=3$, the separation of flow is beginning, and the vortices are formed in the region between the cylinders. When $H/Z=8$, the phenomenon of vortex shedding turns up, and the influence of the front cylinders decreases.

The flow around circular cylinders is a complex flow case, therefore, drag coefficients measurement is an important factor to be considered in the current case. The drag coefficients calculate based on the drag of single pile in same water channel. The figure 4 gives the drag coefficients with different value of $H/Z$. It can be seen that, with the value of $H/Z$ increasing, the shading coefficient of the front cylinders are diminishing. When $H/Z=8$, the drag coefficient is becoming steady.

![Figure 4](image)

**Figure 4.** The drag coefficient showing the flow over the circular cylinders.

### 4. Conclusion

The aim of this paper is to study the flow field characteristics between cylinders with different layouts (different value of $H/Z$), to provide initial value for long-span bridge construction. The standard $k$--$\varepsilon$ turbulence model with unstructured grid has been presented in this study. Comparison to different value of $H/Z$ proved that the results obtained with the standard $k$--$\varepsilon$ turbulence model predicted efficiently the main characteristics of the flow around the cylinders. The results showed that the formation of vortex, vortex shedding, the separation of flow are formed between cylinders with value of $H/Z$ increasing.

In future, we will test the flow field characteristics between cylinders with different layout, in order to evaluate the shadowing effect between circular cylinders.
Acknowledgments

The paper is funded by the Science and Technology Project of Jiangsu Province (Grant No. BM2018028, BZ2017056), the Water Resources Science and Technology Project of Jiangsu Province (Grant No. 2015032, 2016072), the Technology Demonstration Project of MWR (Grant No. SF-201816).

References

[1] Elkhoury M 2016 Assessment of turbulence models for the simulation of turbulent flows past bluff bodies J. Wind Eng. Ind. Aerodyn. 154 10-22
[2] Benjamin B, Alen O, Brane Š et al 2014 A computer-aided visualization method for flow analysis Flow Meas. Instrum. 38 1-8
[3] Xia G D, Chen Z, Cheng L X et al 2017 Micro-PIV visualization and numerical simulation of flow and heat transfer in three micro pin-fin heat sinks Int. J. Therm. Sci. 119 9-23
[4] Shinji E, Mohammad R N and Hidetoshi H 2014 Visualization experiment of complex flow field in a sphere-packed pipe by detailed PIV measurement Fusion Eng. Des. 89 1251-6
[5] Hu H B, Bao L Y and Ruan C 2013 Laser PIV system used for flow field displaying and measuring Fire Control & Command Control 38 111-5
[6] Ma X J, Huang B, Wang G Y et al 2017 Experimental investigation of conical bubble structure and acoustic flow structure in ultrasonic field Ultrason. Sonochem. 34 164-72
[7] Ma H W, He X, Zhang J H et al 2013 Experimental investigation on engine intake port flow-field visualization Chin. Inter. Combust. Eng. Eng. 34 46-50
[8] Fu X G and Yu F F 2016 Technique of flow field display with hydrogen bubble and application Eng. Test. 56 21-4
[9] Morteza G, Gokhan A, Mustafa U et al 2016 Visualization of microscale cavitating flow regimes via particle shadow sizing imaging and vision based estimation of the cone angle Exp. Therm Fluid Sci. 78 322-33
[10] Wang X Y, Wang R Q, Du S J et al 2016 Flow visualization and mixing quantification in a rod bundle using laser induced fluorescence Nucl. Eng. Des. 305 1-8
[11] Marsel V Z, Alexey P T, Alexander A C et al 2015 Gas flow visualization using laser-induced fluorescence Dyn. vibroacoust. mach.(DVM2014) 106 92-6
[12] Arne T, Kai R, Marc R et al 2017 Induced infrared thermography: Flow visualizations under the extreme conditions of an open volumetric receiver of a solar tower Int. J. Heat Fluid Flow 65 105-13
[13] Chen Y Y, Yu Y, Zhong X et al 2017 Influence of flow velocity on flow field's optical tomography diagnosis Opt. Commun. 382 386-91
[14] Wang X X, Liao Z, Wang Y H et al 2017 Study on cuttings transport behavior in annular space of wellbore based on fluent Chin. Energy Environ. Prot. 39 217-22
[15] Ke X Y, Prahl J M, Alexander J J D et al 2017 Mathematical modeling of electrolyte flow in a segment of flow channel over porous electrode layered system in vanadium flow battery with flow field design Electrochim. Acta 233 124-34
[16] Zhu Y, Gao X R and Li X R 2011 Numerical simulation and visualization of flow field in a special piloted directional valve for marine steering gear 2011 Int. conf. adv. Eng. 24 539-45
[17] Catalano P, Wang M, Iaccarino G et al 2003 Numerical simulation of the flow around a circular cylinder at high Reynolds numbers Int. J. Heat Fluid Flow 24 463-9