Numerical Analysis of the Effect of the Cylinders with Different Cross Sections on the Flow Field Characteristics

Haixia Wang¹, Yufei Liu¹, Xing Cao*¹ and Jusheng Chen¹

¹College of Electromechanical engineering, Qingdao University of Science & Technology, Qingdao, Shandong, 266061, China
*Corresponding author’s e-mail: qustcaox@163.com

Abstract: To test the effect of the cylinder on the flow field characteristics, four cylinders of different sections were used in this study. The vorticity characteristics, lift and drag, spectral characteristics of non-circular cross sections in two different placements were investigated. According to the results, for the effect of placement method of non-circular cross-section cylinders, the cylinders receiving force from fluid in the flow field under the vertex-windward placement were more stable, and the vibration was smaller.

1. Introduction
Flow around cylinder is common in nature. Also, cylinders are a common structure for its engineering application as well as academic importance, in bridge piers, chimneys, towers, heat exchangers, etc.[1]. To investigate the dynamics characteristics of cylinders, the numerical studies have been conducted frequently[2].

The circular cross-section cylinder is commonly selected as fundamental structure in a wide range of multiple domains. The factors affecting the flow and the structure of the flow field of cylinder of circular cross-section have high practical implication. Up to now the most widely studied cylinder is circular cross-section cylinder[3]. Non-circular cross-section cylinders are unexplored relative to the study of circular cross-section cylinders. The complex phenomena in flow distribution are usually induced when flow goes across the non-circular cylinder[4]. Flow around non-circular section cylinder is being extensively studied. Iungo and Bursetti[5] reported that the direction and scale of flow affected the flow around the triangular prism. The force of the triangular prism would be obviously different with the variation of the flow direction. Trias et al.[6] studies the flow separation at the leading edge of flow around squared cross-section cylinder at Re=22000 using the DNS method. Moreover, the less research compared the differences between non-circular cross-section cylinder and circular cross-section. Accordingly, in this study, the flow characteristics and the flow field structure of three types of non-circular cross-section cylinders were studied. Subsequently, the comparison of circular cross-section and non-circular cross-section cylinders is reported.

2. Numerical approaches
2.1. Computational models
For numerical simulation calculations, the calculation domain will to some extent affect the accuracy of the simulation results. The distances that ensure the numerical simulation results relatively stable between the inlet and cylinder center, between the upper border and column center and between the
lower border and column center should be not less than 10D (D denotes the characteristic length of the cylinder). From the outlet to column center, the distance should not be less than 20D[7].

In this study, the circular cross section, the Squared cross section, the regular triangular cross section and regular hexagon section cylinders with the same diameter of the Circumscribed circle were selected. The numerical domain of the cylinders is shown in figure 1.

![Figure 1. Computational model of triangular cylinder](image)

Non-circular cross-section cylinders have the two placements, the vertex windward placement and the side windward placement. The domain and placement of the flow field around the cylinder of the regular triangular are shown in figure 1 (a) and (b). The repeating details of the rest of the two cases are not to be provided.

2.2. Governing equation
The computational model of two-dimensional incompressible viscous fluid was employed in this study. Since the temperature change was not considered here, the energy equation was not considered as well. The continuity and momentum equations are shown by the following equation:

\[
\begin{align*}
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \\
\frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} + \frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) &= 0 \\
\frac{\partial v}{\partial x} + u \frac{\partial v}{\partial y} + \frac{\partial v}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
\end{align*}
\]

Where \( u \) denotes the speed in the x direction, \( v \) is the speed in the y direction, \( p \) is the pressure, \( \nu \) is the kinematic viscosity, and \( \rho \) is the fluid density.

2.3. Boundary conditions
With the continuous development of computer technology, many engineering complex problems can be solved by using regional discrete numerical calculation methods combined with computer solutions [8-12]. FLUENT was used for simulation calculations in this study, and a laminar model was used at \( Re = 200 \) [10].

The finite volume method was used in the simulation calculation. And the boundary conditions were as follows in FLUENT: The entrance adopts velocity inlet boundary conditions. Fluid medium was water. The outlet adopted the pressure outlet boundary conditions and the average static pressure was equal to zero. The two sides perpendicular to the spanwise boundary were set to symmetric boundary conditions. The no slip wall boundary condition was adopted on the cylinder surface.

2.4. Model validation

2.4.1. Mesh independence verification. In this model four sets of mesh of different mesh elements (8250, 15058, 19110, 23483) were selected to demonstrate the flow around circular section cylinder with \( Re = 200 \). The calculation results of mesh independence verification are listed in table 1.
Table 1. Calculation results of mesh independence verification

| Group   | $C_d$ | $C_l$ | $S_t$ |
|---------|-------|-------|-------|
| Mesh 1  | 1.20  | 0.48  | 0.19  |
| Mesh 2  | 1.224 | 0.50  | 0.202 |
| Mesh 3  | 1.240 | 0.510 | 0.210 |
| Mesh 4  | 1.245 | 0.515 | 0.212 |

It could be seen that the differences of all the calculated results were little. The deviation of the results of Mesh 3 and Mesh 4 were less than 1.0%. It was indicating that the grids have already met the requirements for mesh independence. Considering the calculation speed and calculation accuracy, Mesh 3 was used for numerical calculation in all numerical simulation in this study.

2.4.2. Numerical methods verification. Circular section cylinders were selected to demonstrate the reliability of numerical models and calculation methods by comparison with existing literature, because of the extensive research. The lift coefficient, drag coefficient, and $S_t$ computed here and the reference calculation which compared with literature results are listed in table 2.

Table 2. Comparison and analysis of parameters

| Reference | Time-average $C_d$ | Amplitude of $C_l$ | $S_t$ |
|-----------|------------------|------------------|-------|
| Braza M.[12] (1985) | 1.31            | 0.65             | 0.19  |
| Wei Z.L.[1] (2006) | 1.29             | 0.74             | 0.18  |
| Shu P.C.[13] (2008) | 1.34             | 0.66             | 0.20  |
| The results of this study | 1.24           | 0.51             | 0.21  |

In table 2, the deviations were confined into a reasonable range, which proved its reliability of the simulation method in general.

3. Numerical results

The vortices contour of the regular triangular section cylinder under vortexes windward placement and the sides windward placement at $t=0$, $t=T/2$, $t=T/4$ and $t=3/4T$ exhibited a symmetric relationship. Figure 2 (a) reveals that vortexes 1, the leading vortexes in the wake region (the lower vortexes of the $t=0$ moment), gradually moved downstream with the wake and deviates from the cylinder. Subsequently, under the action of the vortexes 2, the shear layer connecting the vortexes 1 and the cylinder was continuously stretched until it broke down. The vortex 1 was separated from the cylinder into independent vortexes to form vortexes that alternately fall off. Figure 3 shows that there was small vortex at the vertex of the regular triangular section under the side-windward condition, and the small vortexes were separated from the dominant vortexes. Due to the symmetrically distributed in the remaining non-circular section cylinders, only a half-cycle vorticity contour map is shown afterwards.

![Figure 2. Vorticity contours of triangular cylinder when the vertex faced the flow](image1)

![Figure 3. Vorticity contours of triangular](image2)
Figure 4. Time curves of lift and drag coefficients of triangular cylinder

$C_l$ and $C_d$ of the regular triangular section cylinder under vertex-windward and the side-windward are shown in figure 4 when Re=200. The abscissa represents the dimensionless time, and the ordinate represents the lift coefficient and the drag coefficient. The amplitude of the $C_l$ under the side-windward placement of the regular triangular section was larger than the amplitude of the $C_l$ under the vertex-windward placement mode. Besides, the time-average value of the $C_d$ under the side-windward placement was greater than that of the $C_d$ placed at the vertex-windward placement. In other words, the placement of the regular triangular section cylinder had a certain effect on the values of $C_l$ and $C_d$. The side-windward placement had larger $C_l$ and $C_d$, and the vertex-windward placement helped to reduce $C_l$ and $C_d$.

Figure 5. Spectral analysis of lift coefficient of triangular cylinder

The $C_l$ spectrums obtained by FFT (Fast Fourier Transformation) of the regular triangular section cylinder under vertex-windward placement and side-windward placement are shown in figure 5. The abscissa represents the frequency, and the ordinate represents the amplitude. Figure 5 suggests that $C_l$ frequency of the regular triangular section cylinder under the side-windward placement was smaller than that of the vertex-windward placement. This was because the vortex shedding frequency $f_v$ of the vertex-windward placement was greater than that of side-windward placement. The time for the vortex shedding in the vertex-windward placement was less than that in the side-windward placement. The vortex shedding was delayed by side-windward placement. The number of $S_t$ under the vertex-windward placement was larger than that under the side of the regular triangular section when frequency and speed were under the identical condition.
Figure 6. Vorticity contours of rectangle cylinder when the vertex faced the flow

(a) $t=0$  
(b) $t=T/4$

Figure 7. Vorticity contours of rectangle cylinder when sides facing the flow

(a) $t=0$  
(b) $t=T/4$

The instantaneous vorticity contour map of Squared section cylinder under vertex-windward placement is given in figure 6. The vortexes separation points under the vortexes windward placement were fixed at the two end points of the upper and lower symmetry. Due to the viscosity of the fluid, there was a recirculation zone at the cusp of the downstream region during the vortexes shedding process. Comparing figure 6 with figure 7, it is found that the length of the wake vortex which fell off at alternation certain period forming region under the vertex-windward placement was shorter.

Figure 8. Time curves of lift and drag coefficients of rectangle cylinder

(a) vertex-windward placement  
(b) side-windward placement

Figure 8 shows the $C_l$ and $C_d$ of Squared cross section cylinder under side-windward placement and vertex-windward placement. This figure suggests that that the Squared section cylinders exhibit periodic pulsation. By comparison, the time-average value of $C_d$ under the vertex-windward placement was larger than that of side-windward placement. Furthermore, the amplitude of $C_l$ under side-windward placement was larger. It is noteworthy that under the vertex-windward placement, the larger drag and smaller lift were imposed on Squared section cylinders.

4. Conclusions

In this study, different shapes of section cylinders have been studied by using commercial numerical calculation software. Circular section and no-circular sections cylinders with the same diameter of the circumscribed circle were selected for research as $Re=200$. The computational models, boundary conditions and governing equation were firstly carried out, and mesh independence was verified, subsequently. The effect of no-circular sections cylinders placement method and side number and comparing of circular section cylinder had been researched. The comprehensive conclusions were drawn as following:

For the effect of placement method as non-circular section cylinders, the amplitude of $C_l$ and time-average value $C_d$ under the vertex-windward placement was smaller. The cylinders which were more stable under the vertex-windward placement received smaller force and minor vibration from fluid.
For the effect of side number as non-circular section cylinders, the amplitude in the spectrogram increased with the increase in sides. Main frequency decreased with the number of sides increasing. St has negative correlation with the number of sides expressed by $f_v$ decreasing. The amplitude increased with the increase in sides, suggesting that a stronger force was imposed to the cylinder with the increase in sides. In brief, increasing sides could prevent vortex shedding for non-circular section cylinders, whereas more fluid forces would be imposed to the cylinder.

Acknowledgments

This work was supported by the [Shandong Provincial Natural Science Foundation, China] under Grant [number ZR2018PEE017]; [the Application Foundation Research Program of Qingdao] under Grant [number 17-1-1-17-jch].

References

[1] Wei Z. L., Sun D.J., Yin X.Y. (2010) Numerical simulation of transverse oscillating airfoil flow in a cylindrical wake. Hydrodynamics research and progress: Series A, Chang.
[2] Zdravkovich M.M. (1997) Flow around circular cylinders. A comprehensive guide through flow phenomena, experiments, applications, mathematical models, and computer simulations, vol. 1. Oxford University Press.
[3] Coutanceau M, Defaye J R. (1991) Circular cylinder wake configuration: A flow visualization survey. Applied mechanics reviews, 44(6): 225-305.
[4] Huang, R. F., Lin, B. H., & Yen, S. C. (2010) Time-averaged topological flow patterns and their influence on vortex shedding of a Squared cylinder in crossflow at incidence. Journal of Fluids & Structures, 26(3), 406-429.
[5] Iungo G.V., Buresti G. (2009) Experimental investigation on the aerodynamic loads and wake flow features of low aspect-ratio triangular prisms at different wind directions. Journal of Fluids and Structures, 25(7): 1119-1135.
[6] Trias F. X., Gorobets A., Oliva A. (2015) Turbulent flow around a Squared cylinder at Reynolds number 22,000: A DNS study. Computers & Fluids, 123:87-98.
[7] Sohankar A., Norberg C., Davidson L. (1998) Low-Reynolds number flow around a Squared cylinder at incidence: Study of blockage, onset of vortex shedding and outlet boundary condition. Numerical Methods in Fluids, 26: 39-56.
[8] Hong J. (2015) Numerical simulation of flow field characteristics in the near-field region of pulsed circular jet. Shandong: East China University.
[9] Wang F.J. (2004) Computational Fluid Dynamics Analysis. Beijing: Tsinghua University Press.
[10] Yin D.C., You Y.X., Wei G. (2008) Three-Dimensional Numerical Simulation of Flow around Upright Cylinder in Two-Layer Viscous Fluid. Hydrodynamics Research and Progress, 23(1): 1-6.
[11] Zhang H. (2010) Electromagnetic optimization control of wake with the goal of drag reduction and vibration reduction. Nanjing: Nanjing University of Science and Technology.
[12] Braza M. (1989) Numerical Study and Physical Analysis of the Pressure and Velocity Fields in the Near Wake of a Circular Cylinder. Journal of fluids mechanics, 165: 79-130.
[13] Shu P., Wang Y.S., Pang Z. Y. (2009) Numerical simulation of flow around a cylinder using DES method based on SST model. Ship Science and Technology, 31 (2), 30-33.