Numerical study of flow characteristics of the blade with different structures

Guoping Li¹, Zhenlai Ma¹, Biwen Chen¹, Changsheng Chen¹, Laizuo Chen² and Bo Gao²

¹Shanghai Marine Equipment Research Institute, Shanghai, China
²Jiangsu University, Jiangsu, China

Email: mazhenl@163.com
Email: chenzhangsheng@shcbsbxjs.onexmail.com

Abstract. In order to study the effect of blade structures on the induced excitation characteristics, two different structures are investigated, which are the blade with special trailing edge and the staggered blade. Firstly, based on the steady numerical simulation, the time-averaged velocity and pressure distribution are obtained at different Re numbers (5×10³ and 1×10⁴). Secondly, the unsteady numerical simulation method LES is applied to acquire the transient and complex flow field around the blade. It is found that both two blade structures significantly change the flow characteristics around the blade, especially in the wake flow region. The evident frequency is observed from the frequency spectrums. It is clear that the dominant frequency is originated from the vortex shedding, which is similar to the Karmen vortex street. Finally, the results will be useful for the design of blade with low noise purpose.

1. Introduction

Flow around the bluff body is an important problem in engineering due to the complex flow induced excitations, and many researches have been carried out. For flow around the single cylinder, Coutanceau¹¹ and Williamson¹² studied the flow around a circular cylinder at different Reynolds numbers. It is found that the vortexes in the flow wake region are influenced by the Re number and the Re=3×10⁵ is defined as critical Re number. As for the double cylinders, Chen³⁴ and Zdravkovich⁵⁶ studied about its interference model with different arrangements. Zdravkovich⁷⁸ and Mederios⁹ provided the concrete flow field distribution of the flow around the double cylinders.

When it comes to the flow around the flat plate, Parker R¹⁰ and Nakamura Y¹¹ have carried out the experimental research about it. With the development of the CFD, the numerical simulation is applied to study the flow around the bluff body. Auteri¹², Leclercq¹³ and Dadmarzi¹⁴ have carried out the numerical simulation research about the flow around the flat plate. According to the numerical simulation, the vortexes in the flow wake region are all captured, which are consistent with the experimental data.

For now, the structures of the bluff body have been considered, which may be useful when utilized in vane pumps and other fluid machinery. J. Hu¹⁵ and Seung-Jae Lee¹⁶ have studied the NACA0009 hydrofoil with different trailing edge truncations and beveled trailing edges. It is found that modifying the trailing edge of blade can change the vortex structures in the flow wake region remarkably. Yao¹⁷ and Zeng¹⁸ studied the hydrofoil with Donaldson’s trailing edge. Results reveal that the amplitude of the vortex shedding is obvious decreased, which is caused by the energy dissipation of vortex

1. Introduction

Flow around the bluff body is an important problem in engineering due to the complex flow induced excitations, and many researches have been carried out. For flow around the single cylinder, Coutanceau and Williamson studied the flow around a circular cylinder at different Reynolds numbers. It is found that the vortexes in the flow wake region are influenced by the Re number and the Re=3×10⁵ is defined as critical Re number. As for the double cylinders, Chen and Zdravkovich studied about its interference model with different arrangements. Zdravkovich and Mederios provided the concrete flow field distribution of the flow around the double cylinders.

When it comes to the flow around the flat plate, Parker R and Nakamura Y have carried out the experimental research about it. With the development of the CFD, the numerical simulation is applied to study the flow around the bluff body. Auteri, Leclercq and Dadmarzi have carried out the numerical simulation research about the flow around the flat plate. According to the numerical simulation, the vortexes in the flow wake region are all captured, which are consistent with the experimental data.

For now, the structures of the bluff body have been considered, which may be useful when utilized in vane pumps and other fluid machinery. J. Hu and Seung-Jae Lee have studied the NACA0009 hydrofoil with different trailing edge truncations and beveled trailing edges. It is found that modifying the trailing edge of blade can change the vortex structures in the flow wake region remarkably. Yao and Zeng studied the hydrofoil with Donaldson’s trailing edge. Results reveal that the amplitude of the vortex shedding is obvious decreased, which is caused by the energy dissipation of vortex
collisions from the upper and lower trailing edges. Mulvany\cite{19} have studied a modified NACA16 hydrofoil via different numerical simulation methods, they compared the simulated results with the experimental data provided by Bourgoyne\cite{20}.

In this paper, two blade structures are designed and analyzed to clarify the corresponding effect on the flow field and induced excitations. The transient numerical simulation method is mainly adopted considering its high calculation accuracy.

2. Numerical simulation setup

2.1. Design schemes of the blade structure

In order to study the influence of different structures on the excitation characteristics of the flow around the blade, two typical blade structures are selected for numerical calculation and analysis. The two blade structures are named as Trailing edge and Staggered blade as shown in Fig.1 and Fig.2.

![Fig.1. Trailing edge](image1.png) ![Fig.2. Staggered blade](image2.png)

As is shown in Fig.1, the chord length \( L = 3D \), where \( D = 10mm \) is the blade thickness. The span width is 20mm. \( S \) is the extension length of trailing edge of 2D, and \( h \) is the thickness of trailing edge of 0.1D. As for Fig.2, \( T \) is flow direction spacing between the upstream and downstream blades of 1D. \( H \) is normal spacing of 2D. \( L \) is the same with Fig.1.

2.2. Mesh generation

It is believed that the quality of the grid has a great influence on the accuracy of calculation results. Firstly, to make sure that the grid is enough for the numerical simulation with expected accuracy, the structured grids are used for the models. Compared with the unstructured grids, convergent characteristic of the structured grid is better. And the structured grids are easier to control, especially around the solid wall. Secondly, the ANSYS-ICEM is utilized to generate the structured grids of two models. To obtain better numerical simulation results, the grids are refined around the solid wall, where large pressure gradient and flow separation are generated. Finally, the number of grid is determined to be \( 1 \times 10^6 \) approximately. The averaged \( y^+ \) value on the wall is about 0~2 to meet the requirements of the LES. Fig.3 shows the mesh grid of the calculated model.

![Fig.3. Detailed structured grids](image3.png)

2.3. Numerical scheme

Turbulent flow is quite complex and unsteady. Various physical parameters of fluid, such as velocity, pressure and temperature, change with time and space. Basically, the fluid follows the conservations of energy, mass and momentum. In the present paper, the flow around the blade does not involve heat transfer. So the control equations are as following.

The continuous equation:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \quad (i = 1,2,3)
\]  

(1)
When the fluid is not compressible, the above equation is:

\[
\frac{\partial (\rho u_i)}{\partial x_i} = 0 \quad (i = 1, 2, 3)
\]  

(2)

The momentum equation is:

\[
\frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \mu \frac{\partial u_i}{\partial x_j} \right) + S_i \quad (i = 1, 2, 3)
\]

(3)

The momentum equation is also known as the Navier-Stokes equation, where \( p \) is the average static pressure, \( u_i \) is the Reynolds time-average velocity in the \( i \) direction, \( \rho \) is the liquid density, and \( S_i \) is the generalized source term.

The discretization and solution of N-S equations are the core contents of computational fluid dynamics. Nowadays, the numerical calculation methods developed by researchers are mainly divided into two categories: direct numerical simulation and indirect numerical simulation. Direct numerical simulation is defined as the direct solution of the transient Navier-Stokes equation, which has extremely high requirements for the computer, and the calculation period is also very long. Indirect numerical simulation is to simplify the turbulent flow. In the present paper, considering the time spent and constraints of the computing resources, the LES is chosen for the numerical simulation.

As for LES, the large-scale flow structures are considered and calculated. As for the small-scale flow structure, its effect on large-scale motion is solved by establishing an appropriate model. The variables \( \phi_i \) can be separated by the filter function:

\[
\phi_i (x, t) = \phi_i^l (x, t) + \phi_i'' (x, t) \quad (i = 1, 2, 3)
\]

(4)

Where \( \phi_i^l \) is the large-scale part of the turbulent flow, and the \( \phi_i'' \) is the small-scale flow structure in the flow.

The N-S equation is:

\[
\frac{\partial}{\partial t} (\rho \bar{u}_i) + \frac{\partial}{\partial x_j} (\rho \bar{u}_i \cdot \bar{u}_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{u}_i}{\partial x_j} \right) - \frac{\partial \tau_{ij}}{\partial x_j} \quad (i, j = 1, 2, 3)
\]

(5)

Where \( \tau_{ij} \) is the subgrid-scale stress (SGS), which reflects the influence of small-scale vortices for the N-S equation, it is defined in Eq.(6):

\[
\tau_{ij} = (\bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j) \quad (i, j = 1, 2, 3)
\]

(6)

SGS is an unknown variable, in order to close the Eq.(5), a suitable calculation model should be used. In the present paper, the SGS model of Smagorinsky–Lilly is applied, which is believed that it has an adequate ability to deal with the turbulent flow.

To acquire the accurate flow field, the commercial CFD software ANSYS-Fluent is applied in the present paper. At first, the standard k-\( \omega \) model is utilized to calculate the steady numerical simulation. Then, the results from the steady numerical simulation are set as the initial condition in LES. The inlet boundary condition is set as velocity inlet, and the outlet boundary is outflow. The surface of blade is set as no slip wall. The SIMPLEC scheme is used to couple pressure and velocity. The second order upwind scheme is used for pressure spatial discretization, and the bounded central differencing is used for momentum spatial discretization.

In the LES unsteady numerical simulation, the time step is a significant parameter, which has a greater impact on the accuracy of the calculation results. The time step can be calculated as follows:

\[
\Delta t = 0.02 \frac{U}{U_\infty}
\]

(7)

Where \( U_\infty \) is the incoming velocity. Finally, time step is selected as \( \Delta t = 5 \times 10^{-5} \) s to ensure the high accuracy of calculation.

3. Results and discussions

3.1. Validation of numerical simulation method

The flow field characteristics of single and double flat blades are studied, and the numerical simulation method is verified by the Strouhal number (St):

\[
St = \frac{fD}{U_\infty}
\]

(8)
Where $f$ is vortex shedding frequency of the cylinder, also known as the Strouhal frequency.

The St number is a dimensionless parameter that reflects the relationship between the vortex shedding frequency, the size and flow velocity. It is a function of the structure and the Reynolds number ($Re$). The St number of the flow around a circular cylinder under different Reynolds numbers has been measured by Lourenco$^{211}$. In the present paper, four cases have been calculated. The comparison between the numerical simulation results and the data from the above literature is listed in Table 1. Where $T/D$ is the characteristic dimension.

### Table 1: Comparison of the Strouhal number (St) in different cases

| Case                      | Research method | $St$       | $Re$ |
|---------------------------|-----------------|------------|------|
| Lourenco (cylinder model) | Experiment      | 0.215±0.005| /    |
| Case1(cylinder model, $T/D$=1) | LES             | 0.21       | 3900 |
| Case2(single plate model, $T/D$=2) | LES             | 0.24       | 48000|
| Case3(single plate model, $T/D$=3.75) | LES           | 0.292      | 48000|
| Case4(double plate model, $T/D$=3.75) | LES           | 0.289      | 48000|

As shown in Table 1, the difference between Lourenco’s experimental data and result from LES is only 2.4%, which is acceptable. Compared with case2 and case3, the St number increases with the increase of $T/D$ ratio under the same $Re$ number. Similarly, compared with case3 and case4, the St number of single plate model is higher than the double plate model slightly.

As a result, it is accepted that the LES can capture complex flow around the blade accurately.

#### 3.2. Effect of different structures on flow around the blade

In the present paper, the $Re$ number is applied, which is defined in Eq.(9).

$$Re = \frac{\rho v L}{\mu}$$

(9)

Where $\mu$ is dynamic viscosity coefficient, $v$ and $L$ are characteristic parameters.

The time-averaged velocity and pressure distributions are shown from Fig.4 to Fig.7, and the distribution of the flow field can be obtained preliminarily.

According to Fig.4 and Fig.5, it is observed that backflow exists in both two blade structures. Besides, the backflow in the structure of trailing edge is more intense than the structure of staggered slightly. Similarly, both two blade structures have low velocity region at the blade leading edge. No
obvious difference of velocity distribution is observed between different Re numbers, which may be because the Re number does not reach the critical value.

Fig.6. Time-averaged pressure distribution at Re=5×10³

Fig.7. Time-averaged pressure distribution at Re=1×10⁴

According to Fig.6 and Fig.7, it is clear that both two blade structures have a high-pressure region at the leading edge. With the increase of Re number, the pressure at the corresponding position increases. However, a low-pressure region is observed at the lower surface of the staggered blade.

Fig.8. The time-averaged velocity in the region of the wake flow at Re=5×10³

Fig.8 and Fig.9 present the time-averaged velocity in the region of the wake flow at different Re numbers. It is observed that the time-averaged velocity in the wake flow shows semi-parabolic distribution characteristics. Due to the trailing edge structure, the length of the backflow region is extremely short. L_b is defined as the length of the backflow region, which is 0.2D approximately for the blade with trailing edge structure. After the backflow region, the time-average flow velocity of the wake flow increases rapidly showing approximately linear growth trend. It is defined as the "linear growth region", and the time-averaged velocity almost linearly increases from 0 to about 0.6 \( U_\infty \). Then, velocity will change into the "slow growth region", and the flow velocity increases from 0.6 \( U_\infty \) to 0.8 \( U_\infty \). In this area, the flow velocity changes from linear growth to non-linear growth, and the growth
rate gradually decreases along the flow direction. After that, the time-averaged velocity slowly increases until reaching the incoming flow velocity. As for staggered blade, the $L_b$ is about 0.8D. Then, the time-averaged velocity increases rapidly. Affected by the downstream blade, there is a sharp increase of flow velocity in the range of $x/D = 2 \sim 5$. Then, the growth rate of the time-averaged velocity tends to be unchanged. Finally it will approach the incoming flow velocity.

![Time-averaged velocity in the region of the wake flow at $Re=1 \times 10^4$](image.png)

Fig. 9. The time-averaged velocity in the region of the wake flow at $Re=1 \times 10^4$
Having investigated the distributions of time-averaged velocity and pressure, it is necessary to study the evolution of the wake flow. Fig.10 and Fig.11 show the evolution of the wake flow around the blade with different structures at different moments under two Re numbers. It can be seen that the
obvious periodic changes in the flow around the blade can be captured. As shown in Fig.10(a) and Fig.11(a), the large-scale vortex structure at the downstream of the blade with trailing edge is affected by the typical structure. It moves downstream and falls off from the trailing edge. At the same time, only two vertex structures are obvious observed in the wake flow, which form and alternately fall from the upper and lower surfaces of the blade. The Karmen vortex street is clearly observed in the wake flow region. As for the staggered blade shown in Fig.10(b) and Fig.11(b), the vortexes formed in the upstream are impacted and damaged by the downstream staggered blade, which is affected by the upstream blade. The wake flow of the downstream blade is more complicated as characterized by small-scale vortexes and flow separation phenomena.

In order to clarify the evolution of the vortex structure in the region of the flow wake, the vorticity distributions at different Re numbers are investigated.

According to Fig.12 and Fig.13, it can be observed that both blade structures have obvious periodic changes in the flow around the blade. The large-scale vortex structure downstream the blade is affected by the trailing edge. It moves downstream and falls off at the distance from the trailing edge. The vorticity is concentrated at the corners of the upper and lower surfaces. As for the staggered blade, affected by the upstream blade, the turbulence of the incoming flow changes, and the vortex formed in the upstream is impacted by the downstream blade. It causes that the flow in the wake region of the downstream blade is more complicated. Many small vortices in the wake region are generated around the upper and lower surface edges.
3.3. Effect of different structures on unsteady load of the blade

As shown in Fig.14 and Fig.15, the lift-drag coefficients are given at different Re numbers. For
different blade structures, the lift-drag coefficients continue to fluctuate with time. Similar to the flow around a cylinder, the lift coefficient curve fluctuates significantly over time. The drag coefficient fluctuation amplitude is relatively low, and its average value is greater than zero. Two different regions of high and low resistance exist for the lift-drag coefficients. The region of high resistance corresponds to a large fluctuation of the lift coefficient, and the region of low resistance corresponds to a small fluctuation of the lift coefficient.

The values of lift-drag coefficients are given in Table.2 and Table.3 at different Re numbers. From the above results, it is observed that the lift coefficient for the blade with trailing edge is more stable than the lift coefficient in staggered blade, while the staggered blade has the larger lift fluctuation amplitude due to the small-scale vortexes existing in the wake flow.

![Graphs showing lift-drag coefficients at different Re numbers.](Trailing edge) ![Graphs showing lift-drag coefficients at different Re numbers.](Staggered)

Fig.14. Lift-drag coefficients at $Re=5\times10^3$

Fig.15. Lift-drag coefficients at $Re=1\times10^4$

| Table.2 Lift-drag coefficients in $Re=5\times10^3$ |
|--------------------------------|
| **Trailing edge** | **Upstream blade** | **Downstream blade** |
| $C_D$ | 0.60 | 0.25 | 0.19 |
| $C_{L\text{rms}}$ | 0.40 | 0.54 | 0.22 |

| Table.3 Lift-drag coefficients in $Re=1\times10^4$ |
|--------------------------------|
| **Trailing edge** | **Upstream blade** | **Downstream blade** |
| $C_D$ | 0.55 | 0.94 | 0.69 |
| $C_{L\text{rms}}$ | 0.52 | 2.12 | 0.50 |
Fig. 16. Frequency spectrum

To obtain the frequency spectrum of the lift-drag coefficients at different Re numbers, the FFT method is applied, and the results are presented in Fig. 16. The abscissa represents the frequency in Hz, and the ordinate represents the amplitude, which essentially represents the magnitude of the vortex energy. In all cases, only one obvious characteristic frequency is captured in the velocity spectrum, and it is the dominant frequency in spectrum. It indicates that the flow field structure around the blade shows periodic characteristics, which is characterized by the dominant large-scale vortex structure in the flow field.

3.4. Discussions
In the present paper, two different blade structures are concentrated. The utilized numerical simulation method is validated via experiment data provided in the literature. Then, considering the different Re numbers, the time-averaged velocity and pressure distributions are obtained. To acquire the transient flow field, the LES method is applied to capture the complex flow structures and time-frequency signals. The analysis suggests that the trailing edge structure will increase the contacting area of the fluid around the blade. The flow resistance downstream of the separation point is small, and the overall averaged resistance is reduced. As for the staggered blade, the upstream velocity gradient becomes larger due to the hindrance of the downstream blade. The overall averaged resistance increases. For the frequency spectrums, it is clearly observed that only one characteristic frequency caused by the vortex shedding occurs. Besides, the amplitude of the blade with trailing edge is stronger than the staggered blade slightly.

4. Conclusions
In the present paper, the blade with trailing edge and staggered blade are investigated by the numerical simulation method to clarify the flow pattern and induced excitations at different Re numbers. Some conclusions are as following.
1. Two blade structures significantly change the flow characteristics of the blade, especially in the wake flow region.

2. The blade with trailing edge has smaller averaged velocity, while the staggered blade has the shorter back flow region.

3. The amplitude of the staggered blade is larger than the blade with trailing edge, and it is due to the stable wake flow of the blade with trailing edge. The large amplitude of staggered blade is caused by the small-scale vortexes in the wake flow region.

In the further study, the experiments will be carried by the PIV and LDA to obtain the effect of the blade on flow field and excitations. The obtained results in the present paper will provide some useful reference for the design of blade with the low excitation purpose.

References

[1] Coutanceau M, Defaye J R. Circular cylinder wake configurations: A flow visualization survey[J]. Applied Mechanics Reviews, 1991, 44:255-305.

[2] Williamson C H K. Vortex dynamics in the cylinder wake[J]. Annual Review of Fluid Mechanics, 1996, 28:477-539.

[3] Chen S S, Paidoussis M P, Au-Yang M K. Flow-Induced Vibration of Circular Cylindrical Structures[C]. Hemisphere Publishing, Washington, 1978.

[4] Chen S S. A general theory for dynamic instability of tube arrays in cross flow[J]. Journal of Fluids and Structures, 1987, 1:35-53.

[5] Zdravkovich M M, Pridden D L. Interference between two circular cylinders; series of unexpected discontinuities[J]. Journal of Wind Engineering and Industrial Aerodynamics, 1977, 2:255-270.

[6] Zdravkovich M M. Review of flow interference between two circular cylinders in various arrangements[J]. Fluids Engineering, 1977, 99:618-633.

[7] Zdravkovich M M. Classification of flow-induced oscillations of two parallel circular cylinders in various arrangements[C]. ASME Winter Annual Meeting: Symposium on Flow-Induced Vibrations, 1984, 2:1-18.

[8] Zdravkovich M M. Review of interference-induced oscillations in flow past two parallel circular cylinder in various arrangements[J]. Journal of Wind Engineering and Industrial Aerodynamics, 1988, 28:183-200.

[9] Mederios E B, Zdravkovich M M. Interference-induced oscillation of two un-equal cylinders[J]. Journal of Wind Engineering and Industrial Aerodynamics, 1992, 41-44, 753-762.

[10] Parker R, Welsh M. Effects of sound on flow separation from blunt flat plates[J]. International Journal of Heat and Fluid Flow, 1983, 4(2): 113-127.

[11] Nakamura Y, Ohy A, Tsuruta H. Experiments on vortex shedding from flat plates with square leading and trailing edges[J]. Journal of Fluid Mechanics, 1991, 222: 437-447.

[12] Auteri F, Belan M, Gibertini G, et al. Normal flat plates in tandem: An experimental investigation [J]. Journal of Wind Engineering and Industrial Aerodynamics, 2008, 96:872-879.

[13] Leclercq D J J, Doolan C J. The interaction of a bluff body with a vortex wake[J]. Journal of fluids and Structures, 2009, 25:867-888.

[14] Dadmarzi F H, Narasimhamurthy V D, Andersson H I, et al. The turbulent wake behind side-by-side plates[C]. 13th European Turbulence Conference (ETC 13), Journal of Physics Conference Series, 2011,318: 1-6.

[15] Hu, J., Wang, Z.B., Zhao, W., Sun, C., Sun, S.L., Guo, C.Y. Numerical Simulation on Vortex Shedding from a Hydrofoil in Steady Flow. 2020 https://doi.org/10.3390/jmse8030195.

[16] Lee, S.J., Lee, J.H., Suh, J.C., 2015. Numerical investigation on vortex shedding from a hydrofoil with a beveled trailing edge. Model. Simulat. Eng. 2015, 9.

[17] Yao, Z., Wang, F., Dreyer, M., Farhat, M. Effect of trailing edge shape on hydrodynamic damping for a hydrofoil. J. Fluid Struct, 2014, 51:189–198.
[18] Zeng, Y.S., Yao, Z.F., Zhou, P.J., Wang, F.J., Hong, Y.P. Numerical investigation into the effect of the trailing edge shape on added mass and hydrodynamic damping for a hydrofoil. J. Fluid Struct, 2019, 167–184.

[19] N. Mulvany, J. Tu, L. Chen, and B. Anderson. Assessment of two-equation turbulence modelling for high Reynolds number hydrofoil flows. International Journal for Numerical Methods in Fluids, 2004, 45(3): 275–299.

[20] D. A. Bourgoyne, J. M. Hamel, S. L. Ceccio, and D. R. Dowling. Time-averaged flow over a hydrofoil at high reynolds number. Journal of Fluid Mechanics, 2003, 496:365–404.

[21] Lourenco L M, Shih C. Characteristics of the plan turbulent near wake of a circular cylinder. A particle image velocimetry study[R]. Private communication, 1993.