Advances in Machinery, Materials Science and Engineering Application IX
M. Chen et al. (Eds.)

doi:10.3233/ATDE230543

Study of the Influence of Rotor Shape on the Aerodynamic Characteristics of Flettner Rotor

Lianzhong HUANG [a], Qiushi SONG [a], Rui ZHANG [a], Kai WANG [a,1] and Ranqi MA [a]
[a] *Marine Engineering College, Dalian Maritime University, Dalian, Liaoning, China*

Abstract. The research and application of sail-assisted ships are of great significance for promoting the green development of shipping industry. The optimization design of sails is crucial for improving the efficiency of wind energy utilization. This study investigates the effect of rotor shape on the aerodynamic characteristics of the Flettner rotor by using the computational fluid dynamics (CFD) method. New geometric model of the multi-circular arc Flettner rotor using SolidWorks software, followed by meshing and setting boundary conditions, is established. On this basis, the aerodynamic characteristics of different shapes of Flettner rotor are analyzed. The study results show that the lift and drag coefficients increase as the velocity ratio increases, and the closer the rotor shape is to a circle at a certain velocity ratio, the larger the lift coefficient is. In addition, the lift coefficient of the multi-arc Flettner rotor is close to that of the circular Flettner rotor at high wind speeds. The study can provide some guidance for improving the aerodynamic performance of the Flettner rotor.

Keywords. Sail-assisted ships, Flettner rotor, computational fluid dynamics, rotor shape, green shipping

1. Introduction

In response to the increasingly stringent global environmental requirements and the implementation of mandatory emission reduction regulations, energy conservation and emission reduction have gradually become the main development direction of the shipping industry [1]. Therefore, the application of new energy sources on ships has also gained widespread attention. As a clean renewable energy source, the use of wind energy to assist ship sailing is one of the important means of energy saving and emission reduction [2]. Flettner rotors are getting more and more attention as a widely used wind assisted propulsion method for its efficiency and reliability. Talluril et al [3] conducted a case study of a merchant vessel with Flettner rotors and the results showed that the adoption of a Flettner rotor can save up to 20% of fuel costs and reduced CO2 and NOx emission levels. Craftt et al [4] studied the effect of end plates on rotor performance and the results showed that without end plates, pressure variations at the top of the rotor can cause strong disturbances in the vicinity of the rotor, which would reduce the performance of the Flettner rotor. Bordognag et al [5] studied the

[1] Kai WANG, Corresponding author, Marine Engineering College, Dalian Maritime University, Dalian, Liaoning, China; E-mail: kwang@dlmu.edu.cn.

interactions between the Flettner rotors and the results showed that the aerodynamic performance of the Flettner rotors was affected by their interactions and that the arrangement of the Flettner rotors on the deck had a significant effect on the driving force coefficient and the heeling force coefficient as the wind direction changed. In addition, Mittals et al. [6] studied the effect of Reynolds number on the lift and drag characteristics, and the results showed that the velocity ratio is between 0 and 2, and the lift coefficient is proportional to the flow time at low Reynolds number. Lelea et al. [7] found that wind speed and wind direction also caused some effect on the net output power of the Flettner rotor, and the net output power reached a maximum at a true wind angle of about 90°. From the analysis of the above study, it is clear that the shapes of the rotor and the velocity ratio have an important influence on the lift and drag characteristics. In this paper, the models of conventional Flettner rotor, 3-arc Flettner rotor, 6-arc Flettner rotor, 15-arc Flettner rotor and 27-arc Flettner rotor are established, and the effects of different rotor shapes on the aerodynamic characteristics of Flettner rotor are studied and analyzed by using computational fluid dynamics methods to provide guidance for the design and optimization of Flettner rotor.

2. Numerical Simulation

2.1. Flettner Rotor Geometry Model

An accurate geometric model is a prerequisite for accurate numerical simulations. Therefore, this paper uses SolidWorks to model the multi-arc Flettner rotor. In order to further compare and analyze the contribution of multi-arc Flettner rotors to the auxiliary thrust for the ship, and study the Magnus effect of multi-arc Flettner rotors, the 2D simulation models of five Flettner rotors were established, which are conventional Flettner rotor, 3-arc Flettner rotor, 6-arc Flettner rotor, 15-arc Flettner rotor, and 27-arc Flettner rotor, respectively, as shown in figure 1.

Conventional Flettner rotor | 3 arc Flettner rotor | 6 arc Flettner rotor | 15 arc Flettner rotor | 27 arc Flettner rotor

Figure 1. 2D geometry model of Flettner rotor.

2.2. Numerical Simulation Calculation

The computational domain created in this paper is built from Fluent's own geometry and mesh components. In order to better simulate the application of the Flettner rotor in real navigational conditions, the size of the computational domain should be as large as possible. But in practical application, the infinite flow field of the computing domain must be transformed into a finite flow field for calculation. In addition, the size of the computational domain is reasonably set with reference to the size of the geometric model, and combined with the discussion of the computational domain in the literature [8]. Above all, the finally determined computational domain in this paper is a rectangle with a length of 30 m and a width of 20 m, with four boundary surfaces, including the inlet, outlet, Cylinder, and wall.

After the computational domain is determined, the flow field region needs to be discretized, namely, replacing the original continuous space with a finite set of discrete points [9]. Usually, the computational domain is replaced by a finite number of non-overlapping sub-regions, and then the node positions of each sub-region and the control body they represent are determined, and the nodes and positions of each sub-region are used as representatives of the control body and contain all the physical quantities of the control body. The discretized cells can be either volume cells (e.g., cubes, tetrahedra, etc.) or surface cells (e.g., rectangles, triangles, etc.). Each cell has physical quantities associated with the fluid flow, such as velocity, pressure, etc. By discretizing the physical quantities on the grid, the hydrodynamic equations can be solved numerically to simulate and predict the behavior of fluid flow.

Generating a suitable mesh is crucial to the accuracy and computational efficiency of numerical simulations. The degree of mesh refinement has a direct impact on the accuracy of the results. A fine mesh can better capture the detailed flow, but the computational cost also increases. Therefore, in practical applications, the appropriate mesh type and refinement level need to be selected according to the requirements of the specific problem and the limitations of computational resources. The grid of the geometric model built in this paper is a structured grid. The advantage is that the quality of the generated quadrilateral mesh is good, the fit to the region boundary is better, and it is suitable for hydrodynamic calculations, as shown in figure 2.

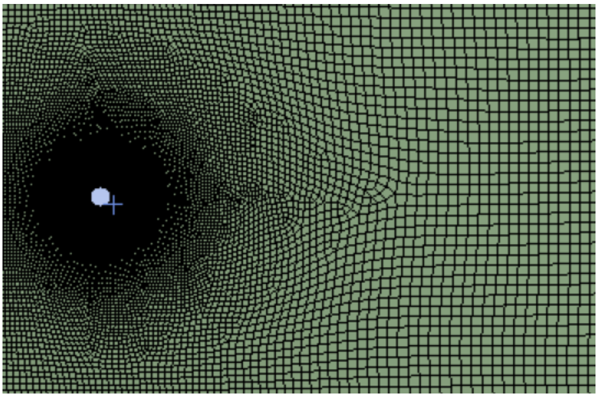

Figure 2. Mesh generation.

The Fluid simulation calculations are performed by using the Fluent software in a well meshed computational domain [9]. The wind speed is 5.388 m/s, the flow of air is taken as an incompressible fluid, the two-dimensional incompressible N-S equation and the two-dimensional continuity equation are selected as the controlling equations, and the Realizable k-ε model is adopted as the turbulence model. The initial parameters are set as pressure p=101325 Pa, air density ρ=1.205 kg/m^3, temperature T=288.16 K, and the boundary conditions of each boundary layer are defined, and the boundary types are shown in table 1.

Table 1. Boundary Condition.

Name	Boundary Condition
Inlet	Velocity-inlet
Outlet	Pressure-outlet
Cylinder	No-slip wall
Wall	No-slip wall

The variation of the lift and drag coefficients with the velocity ratio was calculated using 2D numerical simulations in the range of $0 \leq k \leq 3.19777$, keeping the wind direction and velocity constant, as shown in figure 3. The calculated results are recorded in table 2 and the comparison results with the wind tunnel experiments carried out by Bordogna et al [10] are shown in figure 4. From the comparison of the lift coefficient in figure 5 and the drag coefficient in figure 6, it can be seen that the calculated values from the 2D simulation fit well with the test values with small errors. Therefore, the simulation scheme is feasible to be used in this paper.

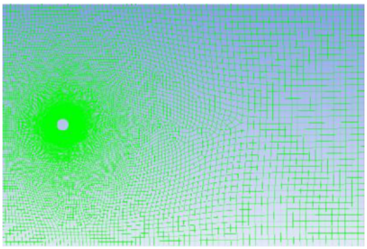

Figure 3. 2-D simulation

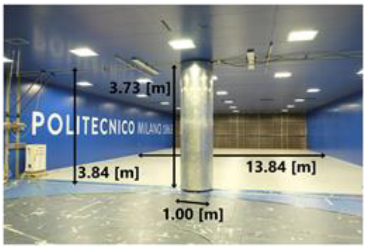

Figure 4. Wind tunnel test [10].

Table 2. Numerical calculation results of the Flettner rotor at different velocity ratio.

Velocity ratio (k)	Lift Coefficient	Drag Coefficient
0	0	0.305
0.47344	1.621	0.421
0.9663	3.023	0.732
1.20853	3.712	0.803
1.45181	4.311	0.925
1.75172	5.433	1.334
2.04429	5.912	1.622
2.28757	6.632	1.934
2.53085	7.521	1.992
2.73638	7.332	2.566
3.19777	7.821	2.960

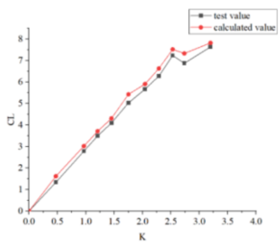

Figure 5. Comparison Chart of lift coefficient.

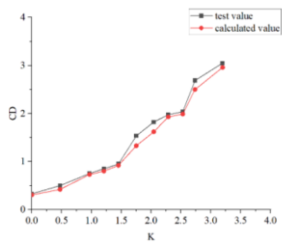

Figure 6. Comparison Chart of drag coefficient.

3. Comparison on the Lift and Drag Coefficient of different Flettner Rotors

Different shapes of Flettner rotors would generate different lift and drag forces when driven by the wind. Lift and drag forces are important factors affecting the efficiency of wind energy utilization, and the lift and drag coefficients are one of the parameters used to describe the extent to which rotor shape affects lift and drag.

However, the accurate calculation of lift and drag coefficients for different rotor shapes is a complex issue involving expertise in aerodynamics and hydrodynamics. The exact value of lift and drag coefficients is also influenced by a variety of factors, such as wind speed, wind direction, and material properties. In addition, different rotor shapes will exhibit different lift and drag characteristics at different wind speeds and angles. Therefore, the detailed aerodynamic experiments and numerical simulations are needed to accurately compare the lift and drag coefficients of different rotor shapes, to find the optimal rotor shapes.

In this paper, in order to compare the aerodynamic characteristics of the multi-arc Flettner rotor, the lift and drag coefficients were calculated under the wind speeds of 3 m/s, 6 m/s, 9 m/s and 12 m/s at k=1.5. The calculated lift and drag coefficients are illustrated in figure 7 and figure 8.

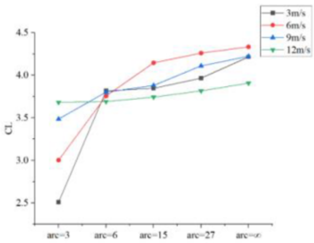

Figure 7. Lift coefficient comparison.

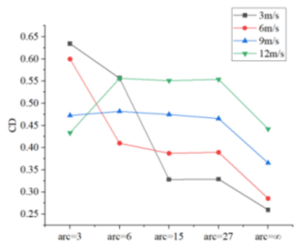

Figure 8. Drag coefficient comparison.

From the comparison results of the lift and drag coefficients, it can be seen that the increase in the number of arc sides, the rotor tends to be circular and the lift coefficient would gradually increase, and the drag coefficient would gradually decrease. In addition, with the increase of wind speed, the lift coefficient does not necessarily increase in v = 6 m/s when the lift coefficient reaches its maximum, and the drag coefficient would not increase with the increase of wind speed, and the drag coefficient of the 3-arc Flettner rotor in v = 3 m/s reaches its maximum. Furthermore, in the case of high wind speed, the effect of rotor shape have little influence on the lift and drag coefficient.

Velocity contour represent the flow of air over the surface of a Flettner rotor and thus can be used to analyze the lift and drag distribution. Wind speed is usually represented using colors or lengths of arrows, namely, the higher wind speeds are represented by darker colors or longer arrows, and the lower wind speeds are represented by lighter colors or shorter arrows. Pressure contour are used to describe the pressure distribution on the Flettner rotor surface at different positions. In a pressure contour, isobars (contours) are usually used to represent different pressure regions, and the distance between adjacent isobars indicates the pressure difference.

The simulated velocity and pressure contours are shown in figure 9. The velocity and pressure contours show that the shape of the rotor has an effect on the velocity and pressure distribution, which in turn produces the high pressure area marked in red in the figure, and the low pressure area marked in green in the figure. The pressure difference can provide lift for the ship.

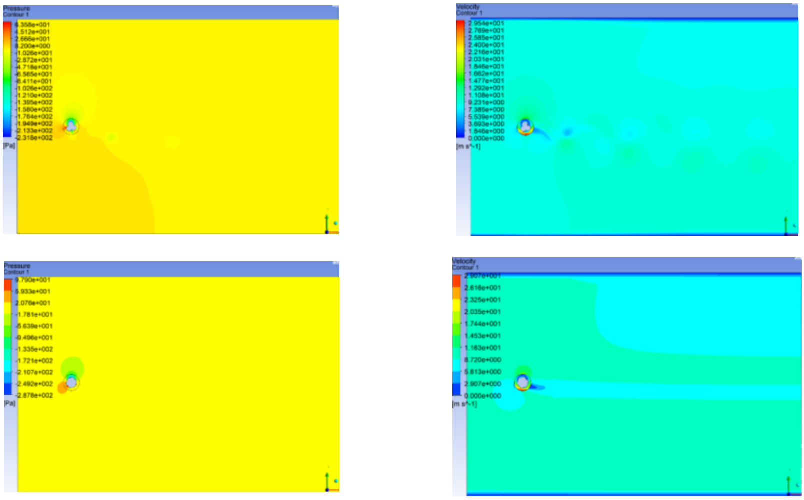

Figure 9. Pressure and velocity contours of the 3-arc Flettner rotor and 6-arc Flettner rotor when v=9m/s velocity ratio k=1.5.

4. Conclusions

(1) The Flettner rotor was simulated numerically by using Fluent software to obtain the corresponding lift and drag coefficients. The calculated results are compared with the experimental results, which show that the calculated results are valid and both the lift and drag coefficients increase with the velocity ratio increases. By numerical simulation of the Flettner rotor, the flow field situation of the Flettner rotor is obtained, which can provide references for the structural design and aerodynamic performance optimization of the Flettner rotors.

(2) By numerical simulation of each shape of Flettner rotor, the results show that the lift coefficient does not necessarily increase with the increase of wind speed at the velocity ratio k=1.5. the closer the shape of the rotor is to a circle, the larger the lift coefficient is. In addition, the difference between the lift coefficient of the multi-arc Flettner rotor and the conventional Flettner rotor decreases with the increase of wind speed.

Acknowledgements

The authors are grateful to the support of the National Key R&D Program of China (2022YFB4300803), the National Natural Science Foundation of China (52271305, 52071045), and Fundamental Research Funds for Universities (LJKQZ2021009, LJKMZ20220360), the Fundamental Research Funds for the Central Universities (3132023211, 3132023525).

References

[1] Jie X, Zhou L, Tong J, et al. Current status and prospects of new energy application on ships. Ship & Boat. 2021; 32(5).
[2] Clodic G, Babarit A, Gilloteaux JC. Wind propulsion options for energy ships. ASME 2018 1st International Offshore Wind Technical Conference (San Francisco), 2018; V001T01A002.
[3] Talluri L, Nalianda DK, Giuliani E. Techno economic and environmental assessment of Flettner rotors for marine propulsion. Ocean Engineering. 2018; 154: 1-15.
[4] Craft T, Johnson N, Launder B. Back to the future? A re-examination of the aerodynamics of Flettner-Thom rotors for maritime propulsion. Flow, Turbulence and Combustion. 2014; 92: 413-427.
[5] Bordogna G, Muggiasca S, Giappino S, et al. The effects of the aerodynamic interaction on the performance of two Flettner rotors. Journal of Wind Engineering and Industrial Aerodynamics. 2020; 196: 104024.
[6] Mittal S, Kumar B. Flow past a rotating cylinder. Journal of Fluid Mechanics. 2003; 476: 303-334.
[7] Lele A, Rao KVS. Effect of skin friction coefficient on power developed by Flettner rotor system for ship propulsion. Applied Mechanics and Materials. Trans Tech Publications Ltd, 2018; 877: 378-383.
[8] Ma Y. Numerical simulation and experimental study on the aerodynamic performance of the sail. Wuhan University of Technology, 2009; pp. 1-133.
[9] Yang L. Study on sail performance and engine-propeller matching of large ocean-going bulk carrier. Shanghai JiaoTong University, 2013; pp. 1-96.
[10] Bordogna G, Muggiasca S, Giappino S, et al. Experiments on a Flettner rotor at critical and supercritical Reynolds numbers. Journal of Wind Engineering and Industrial Aerodynamics. 2019; 188: 19-29.