Proceedings of the 3rd International Conference on Computing Innovation and Applied Physics
DOI: 10.54254/2753-8818/9/20240740

Application of magnus effect on wings

Qin Zhang

Xingzhi College, Zhejiang Normal University, Lanxi, Jinhua, Zhejiang, China

2689923242@qq.com

Abstract. As an important part of the aircraft, the wing provides enough lift for the aircraft, but also creates additional drag. An effective wing design guarantees sufficient lift while minimizing the impact of drag on the aircraft. Aerodynamic analysis is an interdisciplinary research in the course of wing design. In this paper, the design of Magnus type is discussed by means of computational fluid dynamics. First, the airfoil with Magnus rollers was designed. Then, the key parameters such as rotational speed and Angle of attack are studied. Through the analysis of the line chart and calculation domain image of the calculation results, the results show that the installation of Magnus roller on the wing can effectively increase the lift and reduce the drag. With the increase of the rotational speed of Magnus Roller, the lift generated is larger. In the process of increasing the rotational speed, the reflux area behind the airfoil keeps decreasing until it disappears. It is proved that the Magnus airfoil can effectively avoid the phenomenon of airflow separation and avoid the stall of the aircraft. The conclusion can provide theoretical basis for the optimization design of the new generation aircraft airfoil.

Keywords: Wing Design, Magnus Effect, Computational Fluid Dynamicsm, Aerodynamic Analysis.

1. Introduction

1.1. Background

Since the Wright brothers made the first aircraft, the wing has experienced the evolution of rectangular wings, straight wings, swept wings, delta wings, trapezoidal wings, variable swept wings, strake wings, etc. By now, wing technology has been relatively mature. However, engineers and scientists hope to find new wings with lower drag and higher lift. As a promising design scheme, the Magnus effect, has attracted wide attention. The basic principle of the Magnus effect is to use the rotation of a cylinder to make the speed difference between the movement of the cylinder surface and the direction of wind. According to Bernoulli's principle, the upper surface and lower surface of the cylinder will generate the required lift. Compared with modern aviation equipment flying based on wing shape, the potential benefits of the Magnus effect, such as high lift coefficient, increased lift and reduced drag, have not been fully demonstrated..

1.2. Literature review

The study of the Magnus Effect can be traced back to the 17th century. After watching the tennis match at Cambridge College in 1672, Newton discovered and deduced the phenomenon and reason for tennis deviation in the process of fast rotation. In 1852, the German physicist Gustav Magnus explained this

DOI: 10.54254/2753-8818/9/20240740

effect in detail, so it was called "Magnus effect" [1]. In 1918, Herman Fottinger placed a rotating cylinder in the ocean current and conducted an experiment related to the transverse force. His conclusion is that in terms of force, the rotating cylinder acts like an inclined flat plate. In 1919, according to Fottinger's suggestion, Gum bel made a propeller with rotatable cylindrical blades. The experiment was a great success, but two scientists found that this technology still has great challenges in the actual industry [2]. After the U.S. oil crisis in the 1970s, based on the consideration of energy issues, all countries in the world ignited research enthusiasm for Magnus
wind turbines [3, 4].

Magnus has also made considerable progress in its application. For example, Zhuang Yue-qing and Huang Dian-gui [5] studied the aerodynamic performance of the wing with a rotating leading edge through numerical simulation. The influence of Magnus cylinder diameter changes and double Magnus cylinder configuration on the aerodynamic characteristics of airfoil was studied, and the optimal speed ratio configuration of double Magnus cylinder configuration was analyzed by Wang Ze et al [6]. On the basis of the generalized Magnus effect, Zheng Huankui [7] put forward a local moving airfoil, compared with conventional airfoils, a higher lift-drag ratio can be obtained at the same flow speed, which can slow down or prevent the flow separation phenomenon on the airfoil surface to some extent. Based on the Magnus effect theory and traditional wing, a new blade structure with a rotating cylinder at the leading edge is proposed. It is found that adding a cylinder with the Magnus effect at the leading edge of the airfoil can inhibit the fluid flow separation around the airfoil and improve the aerodynamic performance of the airfoil [8]. However, the application of Magnus' theory in aircraft wing design has not been yet fully understood, and its key parameters in wing design have not been determined.

1.3. The scope of this research

The traditional airfoil is sensitive to the angle of attack, and the process to make it is very complicated. Although the cylindrical blade has low cost and high lift coefficient, its lift to drag ratio is relatively low. In view of the shortcomings of these two types of airfoils, this paper attempts to study a new type of hybrid blade, while the roller/cylinder is placed on the front side of the traditional airfoil, and the Magnus effect is used to reduce the angle of attack sensitivity of the airfoil blade and improve its lift-drag ratio. In this study, computational fluid dynamics approach is used to investigate how to improve the aerodynamic performance of an air foil with Magnus cylinder. And the key parameters which may have great effects on the performance are simulated and discussed. The following of the paper is organized as follows, Section 2 introduces the methodology of CFD, Section 3 details the results and discussions, and finally, the conclusions of the paper are presented in Section 4.

2. Methodology

2.1. Governing Equations

Fluid mechanics is a discipline that studies the laws of motion and applications of gases and liquids, usually the state of the fluid itself under the action of various forces; while computational fluid dynamics, its basic definition is the numerical calculation, simulation of various related physical phenomena during flow, including flow, heat transfer, chemical reaction, etc. [9]. It integrates various disciplines such as computational mathematics, computer science, fluid mechanics, and scientific visualization. The basic idea of Computational Fluid Dynamics is to replace the original continuous physical quantities in the time and space domains with a set of variable values at a series of finite discrete points. Discrete, establish the algebraic equation system between the variable values at discrete points, and then solve the algebraic equation system to obtain the approximate value of the variable [10]. The governing equation of computational fluid dynamics includes the continuity equation [11],

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

and the Navier-Stokes equations [11],

$$\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}=-\frac{1}{p}\frac{\partial p}{\partial x_j}+(v+v_T)\frac{\partial^2 u_i}{\partial x_j^{\ 2}} \tag{2}$$

Here, u_i is the velocity components in the streamwise (i= 1), spanwise (i=2), and wall-normal (i=3) direction, respectively. p is the air density, p is the pressure, V is the kinetic viscosity VT is the eddy viscosity, which is determined by the k-ε model [12] in the simulation. The aerodynamic performance of the Magnus airfoil could be evaluated by the drag and lift coefficients. Drag coefficient (usually expressed as cd) is a dimensionless quantity, which is used to quantify the resistance of an object in a fluid. The lower drag coefficient means that the object will have less aerodynamic or hydrodynamic resistance. The drag coefficient can be expressed as [11],

$$C_d=\frac{F_d}{\frac{1}{2}pU^2A} \tag{3}$$

where F_d is the drag force, p is the mass density of the fluid, U is the fluid relative velocity, and A is the area perpendicular to the direction of fluid flow. Likewise, the lift coefficient can be expressed as [11],

$$C_l=\frac{F_l}{\frac{1}{2}pu^2A} \tag{4}$$

with F_l is the lift force and C_l is the lift coefficient.

2.2. Flow domain, boundary conditions and grid generation

Geometry development and meshing are important aspects of computational fluid dynamics numerical calculations. Geometry development transfers realistic problems into mathematical problems that are easy to describe, and meshing is the link between the geometric model and numerical calculation. Only when the geometric model is divided into a certain standard mesh can it be solved numerically [13]. Usually 80%-90% of the workload required to complete a project is occupied by model establishment and grid generation, and the numerical calculation part is mainly completed by computer. At present, the geometry development software includes Solid Works, Pro/E, CATIA, etc., and the commonly used meshing software includes ANSYS ICEM CFD, GAMIT, etc. It is noted that ANSYS has developed rapidly in recent years, and it is more and more widely used because of its friendly operation interface, rich geometric interface, perfect geometric function, and flexible topology structure [10]. The developed geometry model is presented in Figure 1. It can be observed in the figure that a roller/cylinder with a diameter of 40 mm is placed at the leading edge of the airfoil. When the airfoil is moving, the roller/cylinder could rotate and thus produce the Magnus force which could be used to improve the aerodynamic performances. As the airfoil need to generate a less lift, in this scenario, the cylinder needs to rotate clockwise. Figure 2 shows the mesh of the flow field around the airfoil with Magnus roller/cylinder. Here, the mesh is generated with different sections, and we make sure the mesh is fine enough to capture the flow features near the surface and the gap region between the airfoil and the magnus cylinder.

DOI: 10.54254/2753-8818/9/20240740

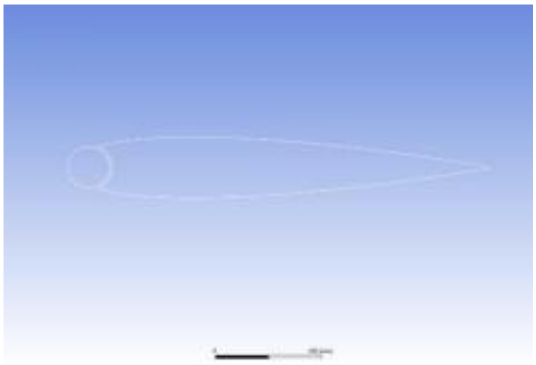

Figure 1. Cross-section of the airfoil with Magnus roller/cylinder at the leading edge.

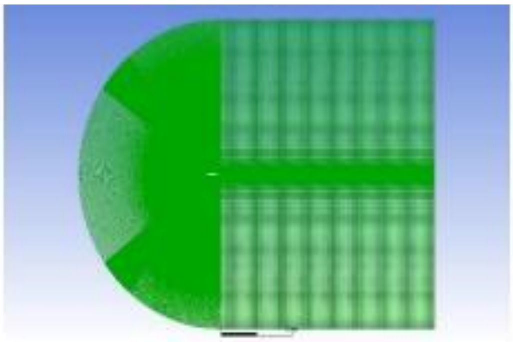

Figure 2. Meshes of the flow field around the airfoil with Magnus roller/cylinder.

3. Result and discussion

3.1. Velocity field around Magnus airfoils

Figure 3 shows the velocity contours for the Magnus airfoilswith an attack angle of 10°, and the rotation speeds of the cylinder are respectively 15000 rad/s, 1000 rad/s, 100 rad/s, and 10 rad/s. It can be observed that the velocity distribution on the upper and lower surfaces of the leading edge of the Magnus airfoil is uneven. After the airflow touches the rotating cylinder at the leading edge of the airfoil, it separates, and the velocity at the separation point of the airflow is almost zero, which is called the stagnation point. The airflow velocity of the airfoil part above the stagnation point is higher and gradually decreases outward in a circular ring. With the increase of the rotation speed, the velocity contours gradually move closer to the inside, and the high-speed airflow area becomes more concentrated. It can also be seen from the velocity distribution in the figure that there is a large low-speed separation region on the upper surface of the rear end of the rotating cylinder in contact with the airfoil. This low-speed separation region further extends backward, and gradually expands at the rear end of the airfoil. As the rotation speed increases, the low-speed airflow area at the front end of the airfoil gradually increases. The reason is that due to the effect of viscosity, the clockwise rotating cylinder makes the airflow move downward. The airfoil surface is thus attached, thereby reducing the area of the low-velocity airflow area. It is speculated that the lift coefficient of the airfoil will increase significantly due to the existence of the rotating cylinder at the leading edge. As regarding to the lift coefficient, it will gradually increase with the increase of the rotational speed.

DOI: 10.54254/2753-8818/9/20240740

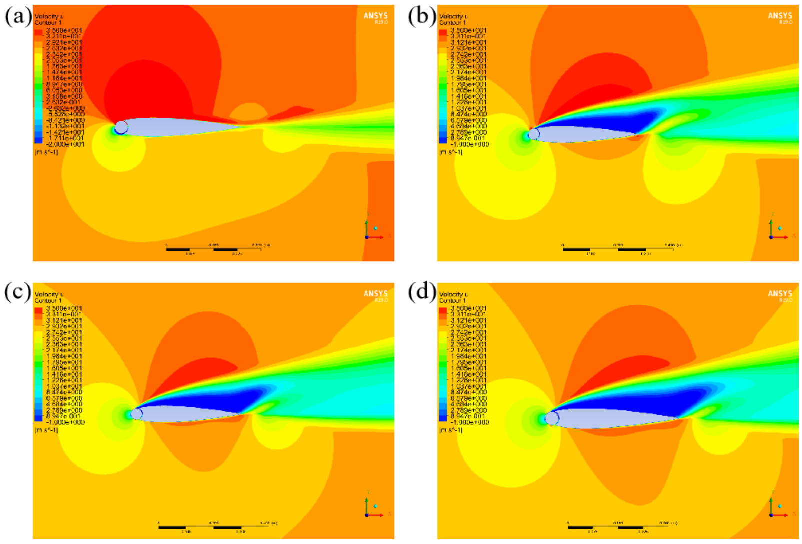

Figure 3. Velocity contours for the flow past Magus airfoils with an attack angle of 10°, and the rotation speeds of the cylinder are: (a) 15000 rad/s, (b) 1000 rad/s, (c) 100 rad/s, and (d) 10 rad/s.

Figure 3 shows the streamlines around Magnus airfoil with an attack angle of 10°, and the rotation speeds are 15000 rad/s, 1000 rad/s, 100 rad/s, and 10 rad/s. Combining the velocity contour s(Figure 3) and the streamlines (Figure 4), it can be observed that when the rotational speed increases, a circular area existing at the upper end of the rear, which has a tendency to become smaller until it disappears. That means that the velocity near the airfoil is getting smaller with the increase of rotation speed. Further examine the figure, it is found that the backflow generated by the rear end of the airfoil is getting smaller as well, which could generate positive effects. It is obvious that the Magnus airfoil can effectively reorganize the fluid flow separation around the airfoil. So, it can be concluded that the Magnus roller/cylinder can effectively improve the aerodynamic performance of the aircraft, i.e., it could effectively increase the lift and reduce the drag without changing other parameters.

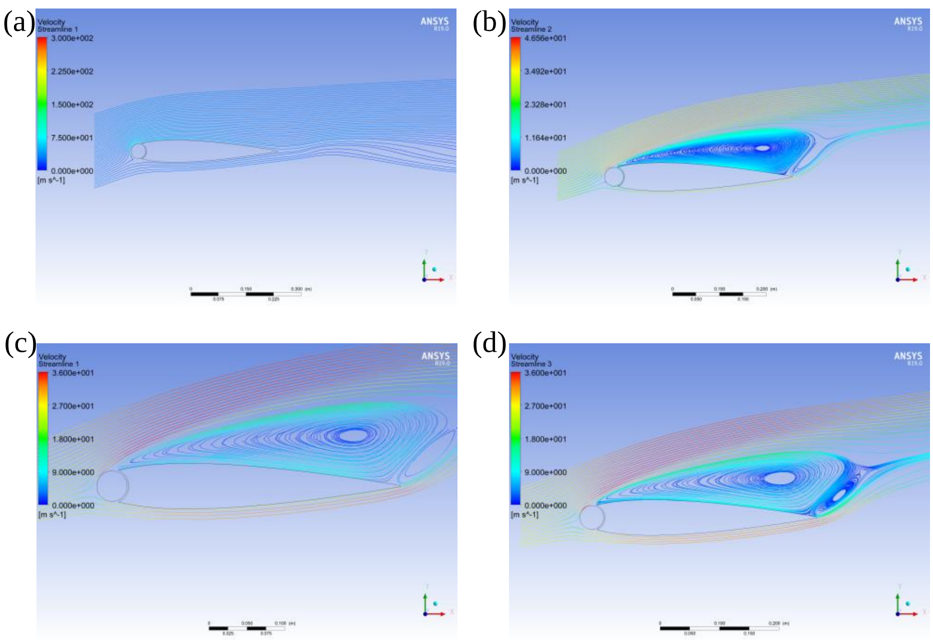

Figure 4. Streamlines for the flow past Magus airfoils with an attack angle of 10°, and the rotationspeeds of the cylinder are: (a) 15000 rad/s, (b) 1000 rad/s, (c) 100 rad/s, and (d) 10 rad/s.

DOI: 10.54254/2753-8818/9/20240740

3.2. *Pressure field around Magnus airfoils*

Figure 5 shows pressure contours for the Magnus airfoils with an attack angle of 10°, and the rotation speeds are respectively 15000 rad/s, 1000 rad/s, 100 rad/s, and 10 rad/s. Here, the pressure contours could provide a more intuitive understanding of the aerodynamic behavior. From the figure 5, it is obvious that the airfoil with the rotational speed of 15000rad/s has a larger pressure difference, while all the other cases, the pressure difference is small or even close to zero. Taking Bernoulli's equation for analysis, it can be seen that the airfoil with rotational speed of 15000rad/s has a capability to generate more lift than other cases, which thus is the optimal design.

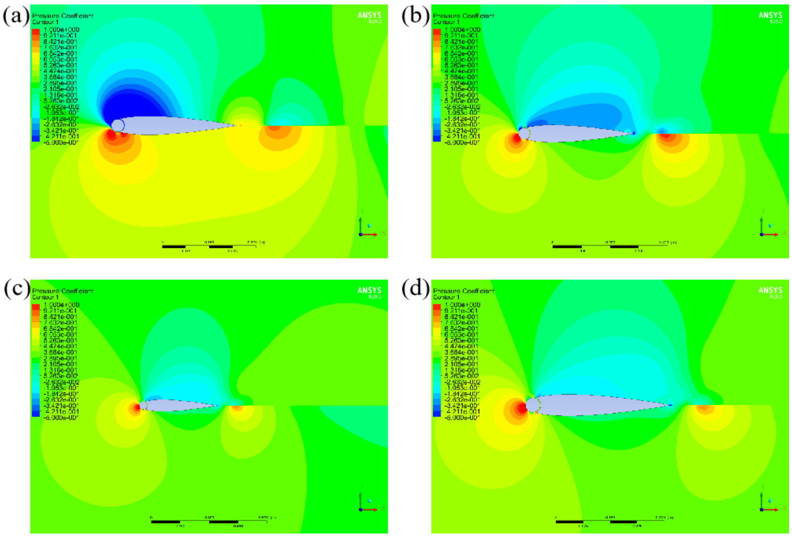

Figure 5. Pressure contours for the flow past Magnus airfoils with an attack angle of 10°, and the rotation speeds are (a) 15000 rad/s, (b) 1000 rad/s, (c) 100 rad/s, and (d) 10 rad/s.

3.3. *Aerodynamic performance*

In this part, the lift, drag performances of Magnus airfoils at different angles of attack and rotating speeds of Magnus cylinder are displayed. Figure 6 shows the lift coefficient and drag coefficient with changing rotation speed. It is observed that, with the increase of rotation speed, the lift coefficient is improved for the angle of 5o and 10o, but it has limited effects when the angle of 0o. For the drag coefficient, the rotation speed could effectively reduce the drag for the angle of 5o and 10o. But again, for the angle of 0o, the effects from the rotation speed are marginal.

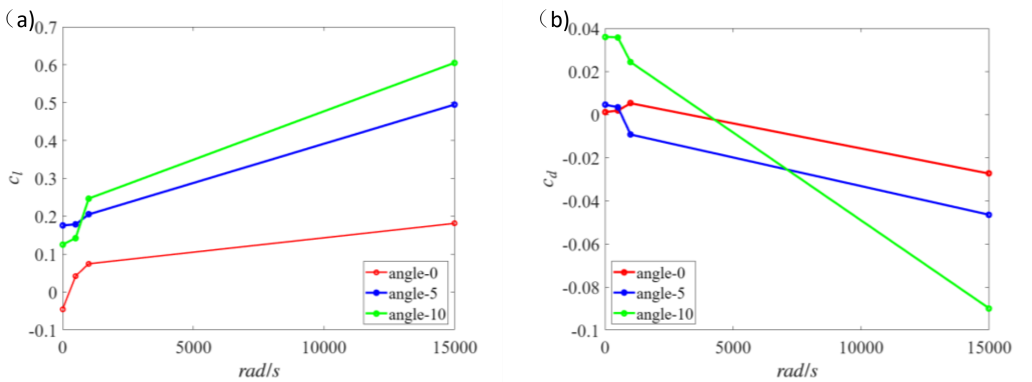

Figure 6. Drawing of direct view of airfoil calculation results. (a) Lift coefficient and (b) Drag coefficient with changing rotation speed.

DOI: 10.54254/2753-8818/9/20240740

4. Conclusion

By using computational fluid dynamic approach, this paper studies the influence of Magnus wing design on the aerodynamic performance of an airfoil. Here, the proposed Magnus airfoil is designed by adding a rotating cylinder at the leading edge of the traditional airfoil. The effects from the key parameters, such as the rotation speed and angle of attack, are systematically investigated. Results shows that at the angle of 5o and 10o, both the lift and drag performances are improved, but with the angle of 0o, the effects of rotation speed are marginal for lift and drag performances. Considering the truth that the angle of attack is usually not zero, it thus means that increasing the speed of the Magnus roller/cylinder can improve the lift and reduce the drag of the airfoil.

References

[1] Seifert, Jost. A review of the Magnus effect in aeronautics. Progress in Aerospace Sciences, 55 (2012): 17-45.
[2] Föttinger, Hermann. Neue Grundlagen für die theoretische und experimentelle Behandlung des Propellerproblems. Jahrbuch STG, 19, no. 1 (1918): 385-472.
[3] Zhao J X. Application of composite materials in wind power generation. High Technology Fiber and Application. 2003, 28 (4): 1~4.
[4] Luo Y F. Development and trend of wind energy and leaf materials in the world. High technologyfiber and application, 2003, 28 (5): 1~7.
[5] Zhuang Yue-qing, Huang Dian-gui. Study on aerodynamic performance of wind airfoil with leading edge rotation. Journal of engineering thermophysics, 2011, 32 (01):43-46.
[6] Wang Ze, Chen Guanhua, Xu Xin, Xue Xiaopeng. Magnus cylindrical numerical study of mechanism of fixed-wing rising. Journal offlight mechanics, 2021, 33 (6) 6:15 to 21.
[7] Zheng Huankui. Research on Aerodynamic Characteristics of New Local Wing Surface Moving airfoil of Micro Air Vehicle. PhD Dissertation. East China University of Science and Technology, 2015.
[8] Zhang Ling, Chen Kaixin, GAO Sheng-qiang, ZHANG Li-dong. Numerical Simulation of Magnus Leading Edge Rotating Cylindrical Wing. Science technology and engineering, 2018, 018(007):252-258.
[9] Zhu Hong. Numerical simulation of wind turbine blade flow field based on Magnus effect. PhD Dissertation. Harbin Institute of Technology, 2010.
[10] Liu Guoyong. Fluid Simulation and Application: Simulation and Application Examples Based on Fluent. Beijing: Metallurgical Industry Press, 2017.
[11] Munson, Bruce Roy, Theodore Hisao Okiishi, Wade W. Huebsch, and Alric P. Rothmayer. Fluid mechanics. Singapore: Wiley, 2013.
[12] Zhou, LiXing. Second-order moment modeling of dispersed two-phase turbulence—Part 1: USM, k-ε-kp, and non-linear k-ε-kp two-phase turbulence models. Science China Physics, Mechanics and Astronomy 54, no. 6 (2011): 1098- 1107.
[13] Shao Xin, Han Siqi, Gao Lubao. ANSYS FLUENT 17.0 Fluid Analysis Examples. Beijing: Beijing University of Aeronautics and Astronautics Press, 2018.10.