Exploring the Influence of NACA0018 Airfoil Attack Angle on the Airflow Characteristics Based on CFD

Hechong Chen[1][a], Hao Mei[2][b] and Guyu Yi[3][c]

[1]School of Mechanical Science and Engineering, Huazhong University of Science and Technology, Wuhan, 430074, China
[2]International Business School, Henan University, Kaifeng, 475004, China
[3]Air Traffic Management, Civil Aviation University of China, Tianjin, 300300, China

Keywords: NACA0018, CFD, Turbulence Model, Attack Angle, Aerodynamic Characteristics.

Abstract: In order to explore the influence of aerodynamic characteristics on aircraft flight performance and analyze the effect of attack angle on the aerodynamic performance of NACA0018 airfoil, numerical simulation of NACA0018 airfoil has been established by using FLUENT software. Firstly, by comparing the results of computational models and experimental values, the numerical boundary conditions and turbulence models applicable to airfoil calculations were determined. Secondly, combined with the principle of lift improvement, the pressure and velocity field of NACA0018 at 0° and 4° two attack angles were analyzed. The experimental results show that increasing the attack angle can significantly improve the lift, that is, the lift of the airfoil at a 4° attack angle is greater than that 0°. This study provides a reference for wing design and aerodynamic performance analysis, which can optimize wing shape and structure based on changes in attack angles under different flying conditions. In the future, further research will be conducted on the matching between wing attack angles and wing types, flap lengths, flap deflection angles during different flight stages, in order to optimize the aircraft performance comprehensively.

1 INTRODUCTION

In recent years, with the advancement of high-performance computers, computational fluid dynamics (CFD) has become a core technology in the field of aerospace research. Researchers have studied the impact of aircraft lift on the aerodynamic performance of wing profiles, as well as the design and failure issues of aircraft lift devices, through numerical simulation methods. The NACA0018 airfoil, as a classic airfoil, has become a standard model for wing aerodynamics research due to its simple geometric shape and wide application. Attack angle(AoA) can improve wing lift within a certain range to meet various flight requirements by increasing lift to varying degrees through attack angle, especially during take-off and climb phases (Fan, Pang, and Liu, 2004). Therefore, it is very significant to understand the NACA0018 airfoil aerodynamic characteristics on aircraft controling and wing designing.

As the attack angle increases, the air velocity flowing through the wing will increase, the air pressure will decrease, and the lift coefficient (CL) will increase. When flying at a constant altitude, the increase on CL indicates the decrease in required ground speed. During take-off, the aircraft must reach sufficient speed and attack angle conditions to balance its lift and gravity. At the end of the ground acceleration phase, the aircraft begins to lift its front wheels. During this phase, it is necessary to maintain acceleration and increase the attack angle to obtain greater lift. The ground effect gradually decreases until it leaves the ground (Airbus, 2002).

During level flight, lift and drag are balanced, and the lift limit is reached when CL equals CLmax. At this point, if the attack angle increases, stall will occur. At high attack angles, the airflow separates from the upper surface of the wing. If the attack angle

[a] https://orcid.org/0009-0001-8965-1592
[b] https://orcid.org/0009-0007-9317-4390
[c] https://orcid.org/0009-0005-3566-0070

Chen, H., Mei, H. and Yi, G.
Exploring the Influence of NACA0018 Airfoil Attack Angle on the Airflow Characteristics Based on CFD.
DOI: 10.5220/0013444600004558

In *Proceedings of the 1st International Conference on Modern Logistics and Supply Chain Management (MLSCM 2024)*, pages 497-504
ISBN: 978-989-758-738-2

continues to increase, the separation point of the airflow becomes unstable and rapidly fluctuates back and forth. As a result, the pressure distribution constantly changes and alters the position and magnitude of lift. This effect is called chattering, which manifests as intense vibrations. When AoA reaches its maximum value, the separation point moves further forward, and the airflow on the upper surface achieves total separation. This phenomenon leads to a significant loss of lift, known as stall (Airbus, 2002).

Although attack angle technology has been widely applied in aviation, simulation research on the influence of different geometric parameters, especially attack angle, on the aerodynamic performance of airfoils is still relatively scarce. The daily operation of large aircraft is related to the safety of personnel and property, as well as the efficiency of airlines. The refined design of aircraft components is of great significance in ensuring that aircraft manufacturing meets design and airworthiness requirements. With the continuous development of aviation technology, higher requirements have been put forward for wing design (Wang Chunyan, 2023).

This study used the NACA0018 airfoil as the research object. In the simulation environment, visual flow fields were obtained under different configurations by changing the attack angle. The changes in lift, drag, and speed coefficients were analyzed based on the exported data. This method allows for the evaluation of the overall aerodynamic performance of the wing under dynamic conditions, providing a reference for aircraft design.

2 METHOD

2.1 Establishment of Geometric Structure of Flow Field

The basic airfoil studied in this paper is NACA 0018, its chord length is set to 1000mm, and it is modeled in ug, as shown in Figure 1.

To create the flow field model, use a benchmark equal to 15 times the airfoil's chord length, that is, the diameter of the C-type area and the side length of the square area are 15000mm. From the front center of the airfoil section, the center line extends 100mm inward as the circle center of the C-type flow field, and the line extends 15000mm towards the rear edge as the axis to draw the square flow field area,as shown in the Figure 2.

According to Figure 3, the hollow airfoil section

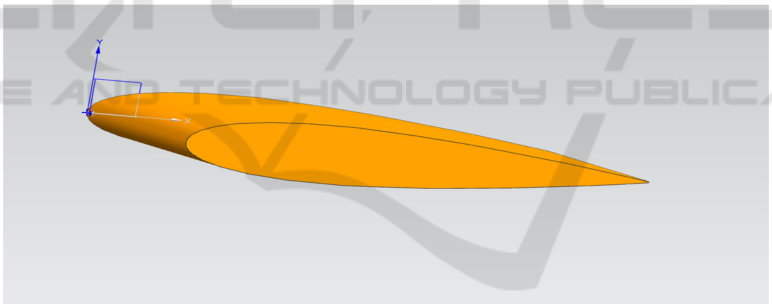

Figure 1: NACA 0018 3D model.

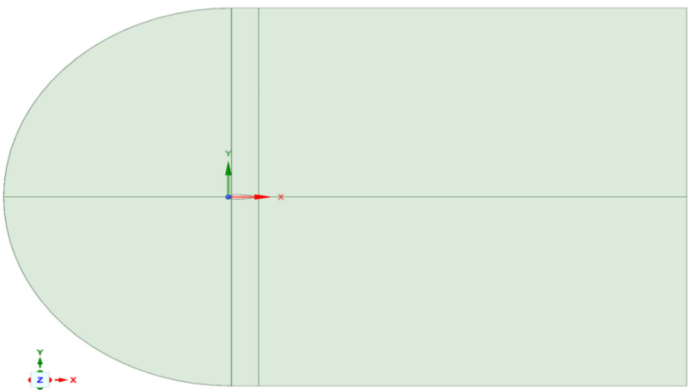

Figure 2: Flow field model.

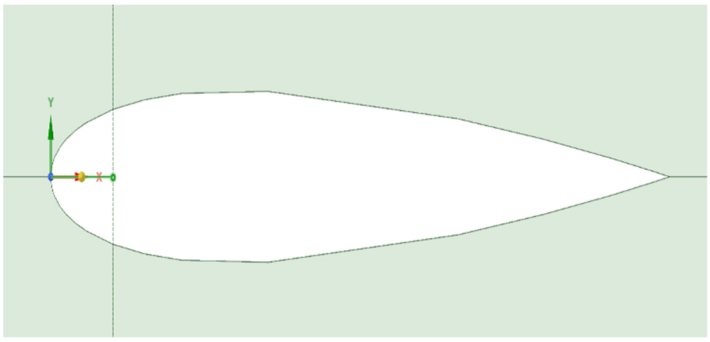

Figure 3: Flow field model details.

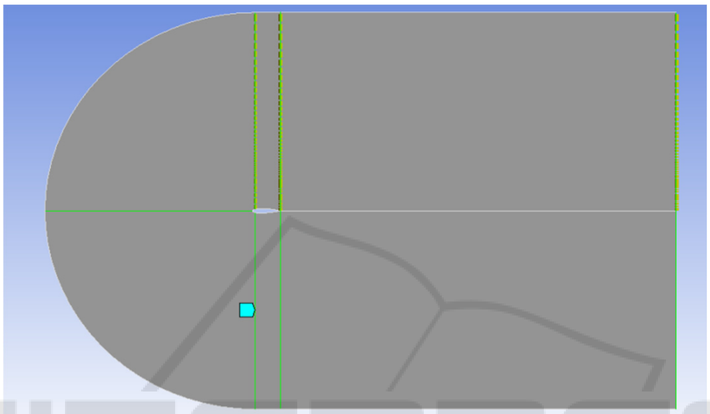

Figure 4: The first set of edge size adjustment.

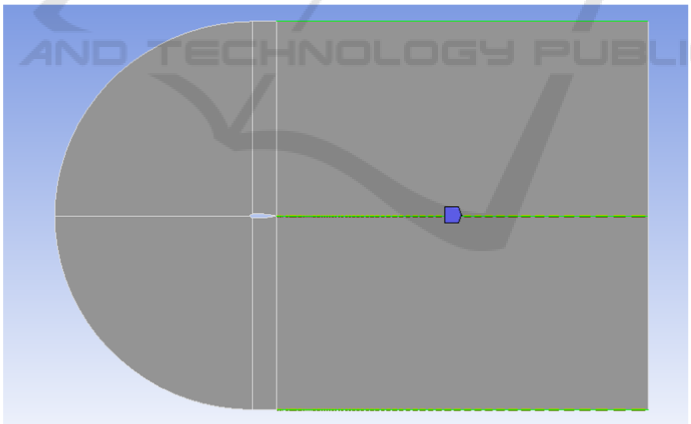

Figure 5: Second set of edge sizing adjustments.

area serves as the boundary between the airfoil and the flow field area. The whole flow field is divided into six regions with the center of the C-shaped flow field and the apex of the tail edge of the airfoil as the boundary (Apostol E I, Țîțu A M, 2023).

2.2 Grid Division

The boundary of each region of the flow field is dimensioned in order to achieve a finer meshing of the area near the airfoil surface. The boundary of the region is named after the overall direction of the diverting field.

The six vertical lines and the horizontal lines that meet the airfoil's leading edge are taken as the first group for edge size adjustment,as shown in Figure 4. As seen in Figure 5, there are 200 partitions, a 40000 bias factor, and bias in the vicinity of the airfoil's surface. The second set of lines for adjusting edge sizing is the three long horizontal lines that follow the airfoil's rear edge, with the number of partitions of 200 and the bias factor of 40000, also biased towards the area close to the airfoil surface,as shown in the Figure 6. According to the Figure 7, the two trailing edge curves of the airfoil along with the short horizontal lines on either side are used as the third set for edge sizing, with a number of partitions of 300 and no bias. The fourth group for edge dimensional adjustment consists of the two airfoil leading edge curves and the two quarter arcs, with the number of partitions being 300 and without bias. Finally, six areas are selected for face grid division, and quadrilateral grid structure is adopted (Kaya M N, Kok A R, Kurt H, 2021).

Generate the final mesh division and specfic details shown in the Figure 8 and Figure 9.

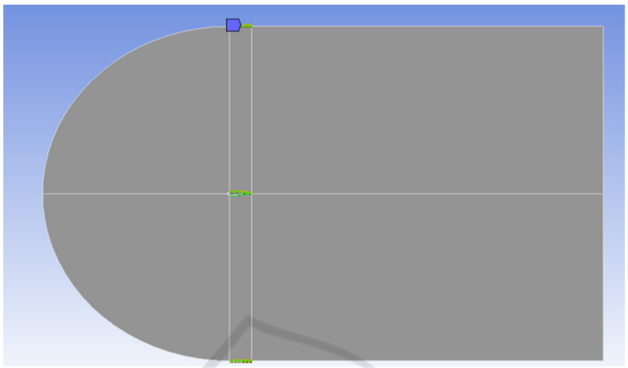

Figure 6: Third set of edge size adjustments.

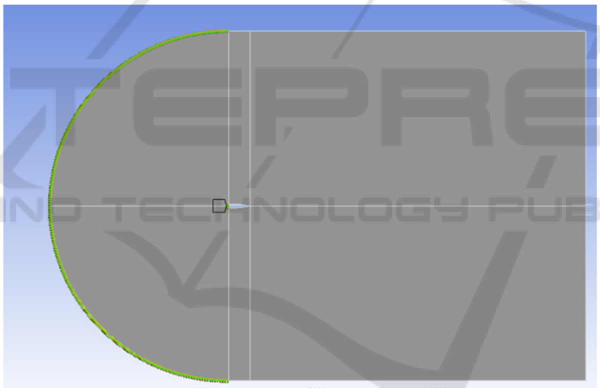

Figure 7: Fourth set of edge size adjustments.

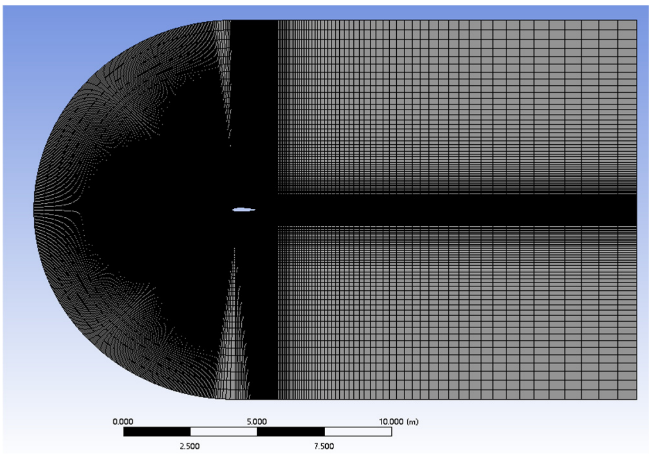

Figure 8: Grid division results.

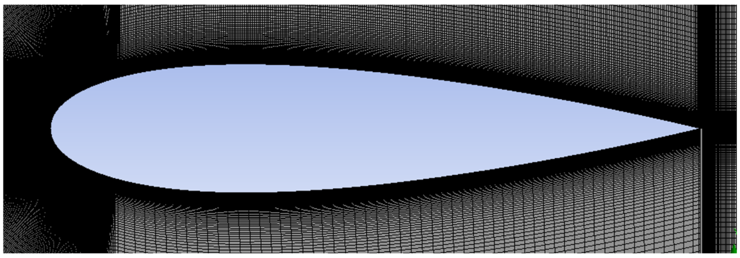

Figure 9: Grid division details.

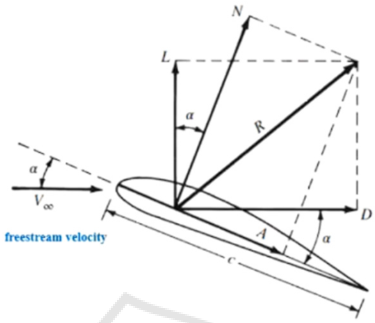

Figure 10: The relationship between the Angle of attack and the lift and drag coefficients.

2.3 Experimental Setup

The solver used in the experiment adopts the pressure base type, the velocity format is absolute velocity, and the plane transient measurement does not consider the influence of gravity. The viscous SST k-omega model is used for the flow field model (Molaa A A, Abdulwahid M A, 2024). Second-order upwind format was used for the discrete equation format, and the COUPLED method was chosen for the pressure-velocity coupling. The fluid used to fill the flow field is air with a density of 1.225kg/m^3 and a viscosity of 1.7894*10-5kg/(m • s). Working conditions Atmospheric pressure is 101325Pa. Set the residual to 10-6 and the number of iterations to 1000.

Experiment with 0° Angle of attack first. In the velocity inlet, choose the velocity definition method of the Components, because the Angle of attack is 0, so do not set the speed in the Y direction, set the speed in the X direction to 33m/s, the concentration of turbulence is 5% and the ratio of turbulence viscosity is 10. The gauge pressure of the pressure outlet is 0. Set the force vector directions for drag to 1 and 0 (corresponding vectors for X and Y, same below), and the force vector directions for lift to 0 and 1, respectively. Initialize and iterate the calculation to get the result of the response.

Then experiment with 4°angle of attack. Similar setup, but due to the change of the Angle of attack, the direction of the velocity and the direction of the force vector of the lift resistance must be changed in response. According to the relevant theoretical knowledge, the relationship between the Angle of attack and the lift force and the drag coefficient can be obtained as seen in the Figure 10.

First of all, for the inlet velocity is changed.,as shown in the figure 10, the velocity in X direction is and the velocity in Y direction is (Li S, Li Y, Yang C, et al, 2018).Therefore, the inlet velocity is changed to 32.93m/s in X direction and 2.31m/s in Y direction in the setting of inlet velocity. Accordingly, according to the results in the Figure 11, the direction of the force vector of the resistance is set to 0.998 and 0.07, and the direction of the force vector of the lift is set to -0.07 and 0.998. The final result is obtained by iterative calculation under such conditions.

3 EXPERIMENTAL RESULTS

The lift force, drag coefficient and corresponding pressure and velocity cloud maps at 0° and 4° attack angles were obtained through the experiment.

The first is the result of 0° angle of attack. As shown in Figure 11, in the static pressure diagram, the pressure distribution around the airfoil is basically gentle and uniform. Two blue areas on the upper and lower parts of the airfoil represent two low pressure areas. Since they are symmetrical airfoil, the pressure in the both parts of the airfoil is basically the same.

The leading edge's stagnation point has a high-pressure area, and a slightly pressurized area is formed in the back edge of the wing. In the velocity distribution cloud diagram, as shown in Figure 12, the velocity at the stagnation point of the leading edge is very low, and a long wake is generated at the trailing edge, while the velocity above and below the airfoil is high (Sun X, Zhou D, 2022). At the position close to the wall, the velocity is basically zero due to the absence of external environmental conditions. The distribution of the two images at 0°angle of attack corresponds closely to the theoretical distribution, and the results of simulation are satisfactory.

The next is the simulation results at 4°angle of attack. According to Figure 13, the static pressure diagram shows that the high pressure stagnation point area moves down due to the presence of the attack Angle.what' s more, an obvious low pressure area is generated above the airfoil. Such results meet our experimental expectations, that is to say, the lift force of the airfoil is significantly improved under this condition (compared with the condition of 0° attack angle). Similarly, in the velocity distribution diagram, as shown in Figure 14, it can be seen that the velocity of the area above the wing is larger, while the stagnation point area moves down, and the wake rises slightly. The simulation results at an Angle of attack of four degree show that the airfoil's lift increases, which is in line with the experimental expectation, and verifies the important influence of the attack angle on the aerodynamic characteristics of the airfoil. The experimental results show that different angles of attack have significant effects on the aerodynamic performance of airfoil, especially the generation and distribution of lift. Designers can optimize the wing shape and structure according to the change of the angle of attack under different flight conditions to improve aircraft performance in different flight stages (Liu et al, 2022). For example, a wing with a larger Angle of attack can be designed during takeoff and landing phases to enhance lift, while it can be optimized for a smaller angle of attack to reduce drag during cruise phases. Studying the pressure distribution and velocity field at different attack angles can help design and optimize the shape of the wing and improve the lift and aerodynamic efficiency of the aircraft (Huang S, Hu Y, Wang Y, 2021). Through simulation, the performance of the wing in different flight states can be predicted, so as to optimize its shape and structural design.

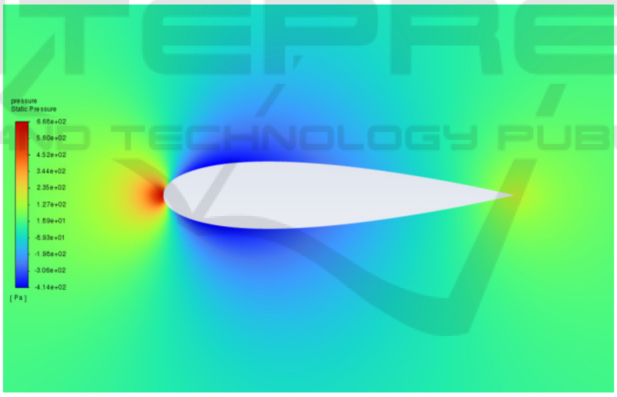

Figure 11: 0°pressure cloud image.

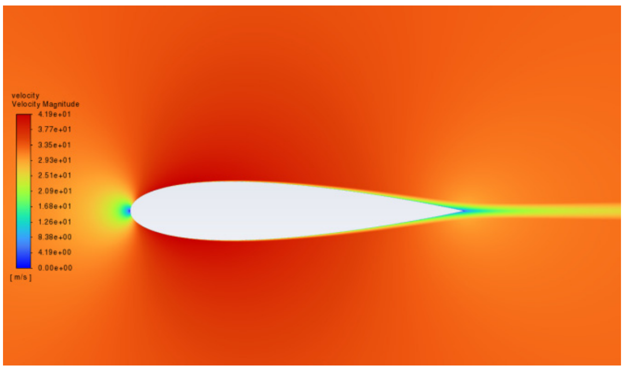

Figure 12: 0°velocity cloud image.

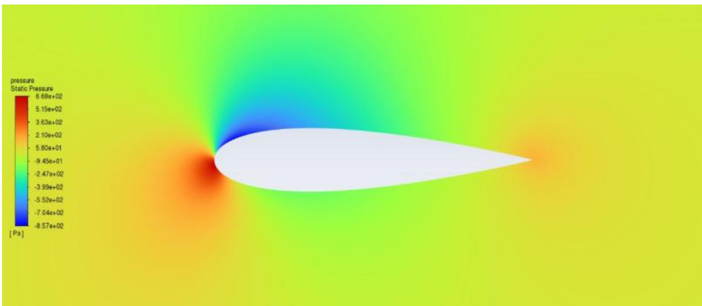

Figure 13: 0°pressure cloud image.

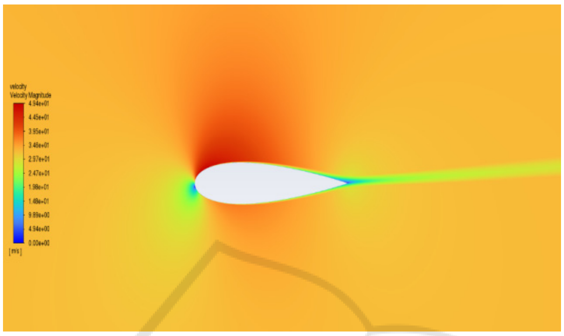

Figure 14: 0°velocity cloud image.

4 CONCLUSION

In this paper,The NACA0018 airfoil's aerodynamic performance at different angles of attack is analyzed by means of computational fluid dynamics (CFD). The experimental results show that, compared with 0° attack angle, the lift of the wing at 4° attack angle is improved, the velocity in the lower part of the wing is reduced, and the pressure is increased. Such analysis is not only a key theoretical understanding of airfoil behavior, but also a practical design that helps to optimize the performance of the wing in different stages of flight, such as take-off, cruise and landing. The results of this study will contribute to the wider field of fluid dynamics and aerodynamics by validating theoretical predictions and providing data that can be used to improve computational models. In addition, the findings will be directly applied to improving aircraft performance through better wing design, ultimately enabling more efficient and safer flight operations.It is hoped that in the future, more accurate CFD simulations will help designers better understand the attack angle's effect on lift and optimize the design of the wing. Furthermore, It is considered that Lift force and attack angle are non - linear. In the future, the research on nonlinear dynamics will be strengthened to improve wing performance and achieve safer and more efficient flight operations.

AUTHORS CONTRIBUTION

All the authors contributed equally and their names were listed in alphabetical order.

REFERENCES

Airbus. 2002. Mastering aircraft performance. 43, 142. 35-36.

Apostol, E., I., Țîțu, A., M. 2023. CFD Simulation of the Aerodynamic Characteristics of the NACA 0018 Symmetrical Profile. *The International Symposium for Production Research.*, 423-431.

Fan, J., Pang, Y., Liu, Zh. 1994-2024. NACA0012 airfoil Aerodynamic characteristics under different flap deflection angles. *Chinese Academic Journal Electronic Press.*

Huang, S., Hu, Y., Wang, Y. 2021. Research on aerodynamic performance of a novel dolphin head-shaped bionic airfoil. *Energy*, 214, 118179.

Kaya, M., N., Kok, A, R., Kurt, H. 2021. Comparison of aerodynamic performances of various airfoils

from different airfoil families using CFD. *Wind and Structures*, 32(3), 239-248.
Liu, Q., Miao, W., Bashir, M., et al. 2022. Aerodynamic and aeroacoustic performance assessment of a vertical axis wind turbine by synergistic effect of blowing and suction. *Energy Conversion and Management,* 271, 116289.
Li, S., Li, Y., Yang, C., et al. 2018. Design and testing of a LUT airfoil for straight-bladed vertical axis wind turbines. *Applied Sciences,* 8(11), 2266.
Molaa, A., A., Abdulwahid, M., A. 2024. Numerical and experimental study of the impact on aerodynamic characteristics of the NACA0012 airfoil. *Open Engineering*, 14(1), 20220506.
Sun, X., Zhou, D., 2022. Review of numerical and experimental studies on flow characteristics around a straight-bladed vertical axis wind turbine and its performance enhancement strategies. *Archives of Computational Methods in Engineering*, 29(3),1839-1874.
Wang, Ch. 2023. Structural analysis and assembly design of flaps for a certain type of aircraft. *China Science and Technology Information*, 14, 38-41.