Using numerical simulation methods to analyze the aerodynamic performance of racing tail fins under multiple factor conditions

Weiyuan Liu

the High School Attached to Hunan Normal University, No. 48 Taozihu Road, Yuelu District, Changsha, Hunan,China

34646545@qq.com

Abstract. In high-speed ground racing cars, the aerodynamics of the tail wing are crucial as they directly affect the overall performance and results of the car. In the aerodynamic performance of the tail wing, down force and drag are the two most important parameters. In order to study the specific effects of factors such as racing speed, tail wing profile, and tail angle of attack on the aerodynamic performance of the tail wing, fluid dynamics numerical simulation methods were used for research.The research results indicate that within the parameter range of 160km/h to 240 km/h, both resistance and down force continuously increase with the increase of speed, and show a typical linear variation pattern. Within the range of 0 ° to 35 ° angle of attack, both drag and down force increase, but within 20 °, the increase in down force is faster, and once it exceeds 20 °, the increase in drag becomes more pronounced.At the same time, four different airfoil structures were compared and analyzed, and it was concluded that the NACA airfoil is more suitable for low-speed operating conditions, as it can generate higher down force at lower speeds. The research results of this article provide certain reference value for the design and finalization of racing tail fins.

Keywords: Numerical simulation, down force, airfoil, Ansys Fluent.

1. Introduction

The application of tail wing technology in racing has been very extensive. The tail wing is one of the most important components of the aerodynamic package in racing cars, responsible for providing down force to the car, ensuring that the car can have sufficient ground grip, and preventing risks during high-speed movement. And during the rapid movement of the racing car, due to the sudden appearance of an empty area behind the car, the airflow will show a turbulent vortex shape and form a low-pressure area, while the front of the racing car is under normal pressure, which will create a large air resistance. The tail wing (and diffuser) is used to quickly smooth out the turbulent flow.

In addition, in the field of racing, in recent years, civilian ordinary cars have also attached increasing importance to the use of tail wing components. Currently, automobile manufacturers around the world are committed to researching and developing new technologies for automotive drag, and a more rational, perfect, and scientific era of tail fins has arrived. In the past, China neglected the installation of the rear wing in the use of sedans because the speed was not high at that time, and the drag reduction and fuel

© 2024 The Authors. This is an open access article distributed under the terms of the Creative Commons Attribution License 4.0 (https://creativecommons.org/licenses/by/4.0/).

saving effects of the rear wing were not very obvious. If driving at a speed of 120km/h on the highway, it can save 14% fuel, and the role of the car's rear wing is very obvious. A rear wing made by precise calculation based on the width of the vehicle body can not only make the vehicle more stable and safe during high-speed driving, but also reduce air resistance, increase speed, and save fuel consumption. This has certain positive significance for energy conservation and environmental protection in the current situation of high oil prices and energy shortages.

2. Relevant Academic Research

There have been some researchers who have carried out related work on the tail wing of racing cars, such as Zhejiang University of Technology conducting wind tunnel and numerical simulation studies on FSAE racing cars. Aerodynamic analysis and comparison were conducted on airfoils that have a significant impact on the aerodynamic performance of the tail wing under different curvature and curvature position conditions. The effects of wing curvature and thickness on its drag and negative lift were discovered, and suitable airfoils for FSAE racing tail wings were determined; At the same time, key parameters such as the wing gap, end plate shape, and number of fins of the tail wing were studied, and the optimal solution was determined based on the variation laws and matching principles of its pressure cloud map and streamline map. Researchers at Harbin Institute of Technology used computational fluid dynamics (CFD) method and boundary element method based on potential flow theory to solve the flow around a two-dimensional airfoil profile NACA0015. [1]Researchers from Chongqing University combined the wing shape analysis software Profili and Xfoil to conduct detailed airfoil selection and angle of attack determination. Three dimensional flow field numerical simulation based on computational fluid dynamics was used to optimize the front and rear wings of the racing car[2].After comparing various styling strategies, a new type of drag reducing curved wing design was determined, using an aerodynamic kit consisting of a "straight main flap" tail wing and an arrow shaped curved front wing. The optimized racing car has an increased negative lift coefficient of 1.68 and a negative lift to drag ratio of 1.91.[3]

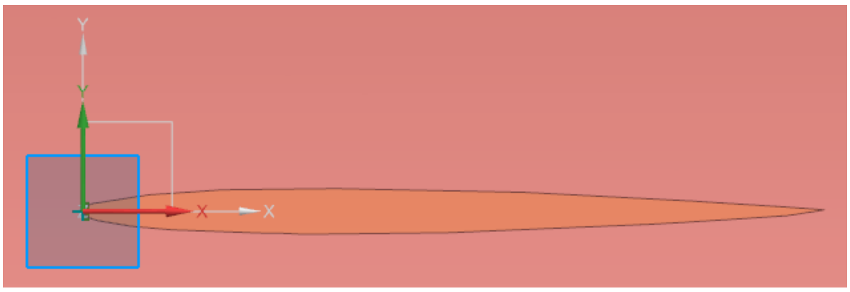
3. Methods

In research, both experimental and simulation methods are commonly used to carry out related research work. This article uses Ansys Fluent commercial numerical simulation software as the main research tool to design the tail structure and calculate the aerodynamic performance of four different airfoils: NACA4412, NACA63A210, NACA23012, and NACA23015. By analyzing the simulation results, the optimal tail wing structure was obtained, and the laws of downforce and drag changing with velocity and angle of attack were given.

3.1. Detailed Approach

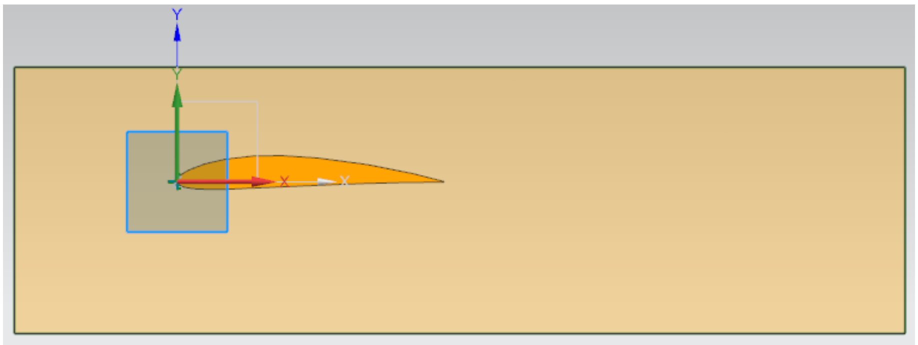
The four types of airfoils used in this article are NACA4412, NACA63a210, NACA23012, and NACA23015. The NACA airfoil is a series of low-speed airfoil models introduced by NASA in the United States. The cross-sections of the four selected airfoils in this article are shown in Figure 1.

(a) NACA.airfoil.63a210

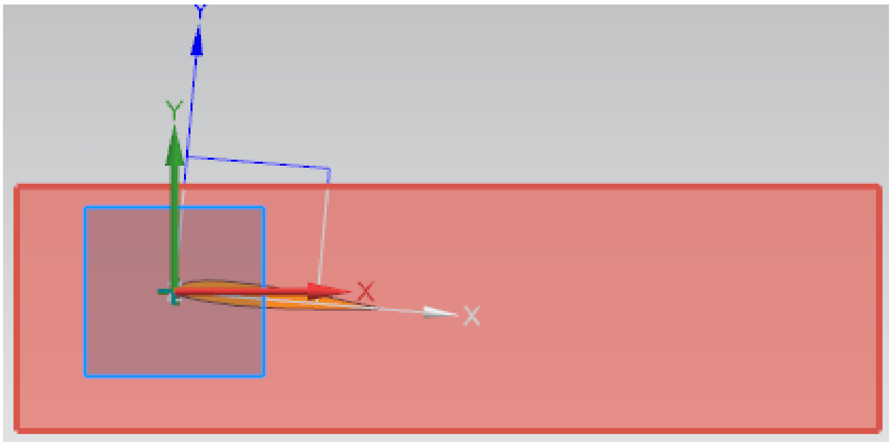


DOI: 10.54254/2753-8818/55/20240180

(b)NACA.airfoil.4412



(c)NACA.airfoil.23012



(d)NACA.airfoil.23015

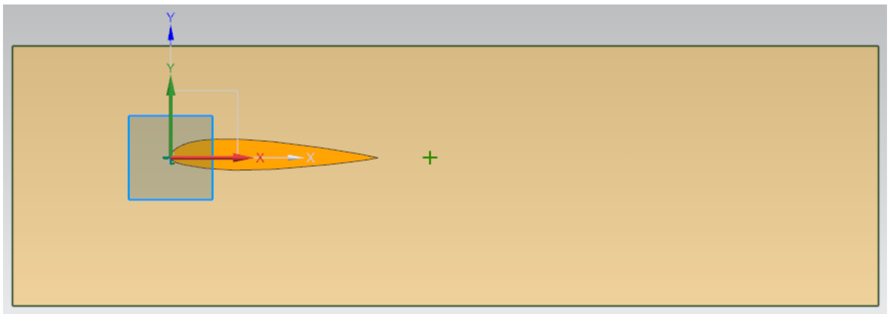
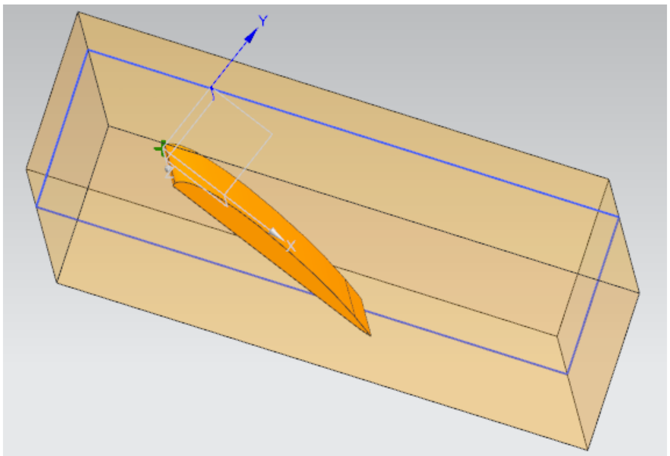


Figure 1. The cross-sections of the four selected airfoils'Drawing

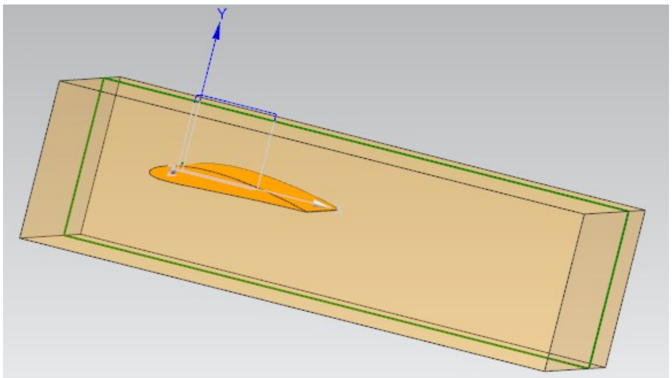
Using cross-sectional point cloud data from the airfoil database for modeling, polynomial fitting curves were applied to the upper and lower wing surfaces to obtain actual wing surface curves. Then, a three-dimensional model of the actual tail wing was obtained by stretching. By raising the middle position of the tail wing, a three-dimensional shaped tail wing can be formed, which can improve the pressure distribution in its middle section and increase the downforce of the tail wing[4],as shown in Figure 2.

DOI: 10.54254/2753-8818/55/20240180

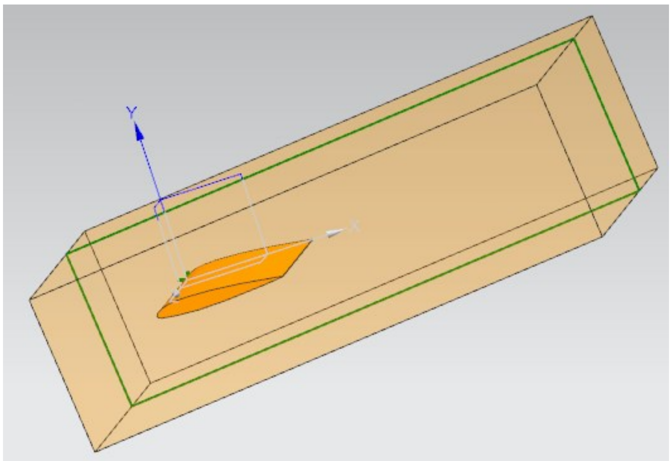
(a)NACA.airfoil.63a210



(b)NACA.airfoil.4412



(c)NACA.airfoil.23012



(d)NACA.airfoil.23015

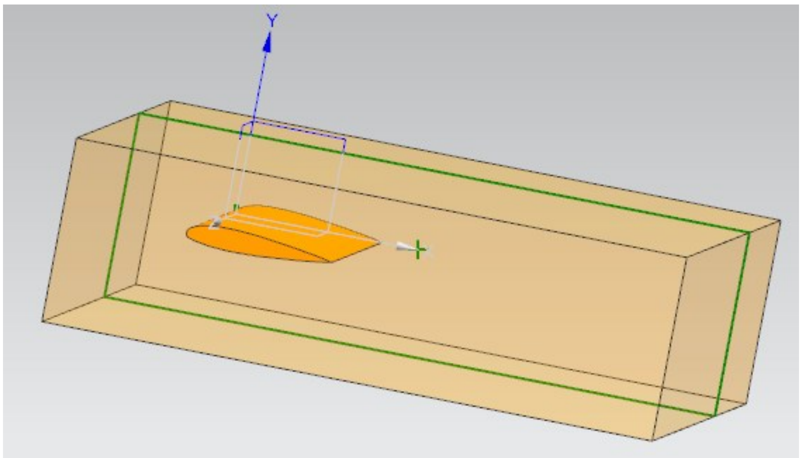


Figure 2. Three dimensional image obtained after stretching the airfoil

For different 3D models of airfoils, the same numerical calculation domain is used, and the setting of the numerical calculation domain generally follows the principle of "two in the front and five in the back, three in the top and three in the bottom". The overall calculation domain size is 80cm * 40cm * 120cm, as shown in Figure 3.

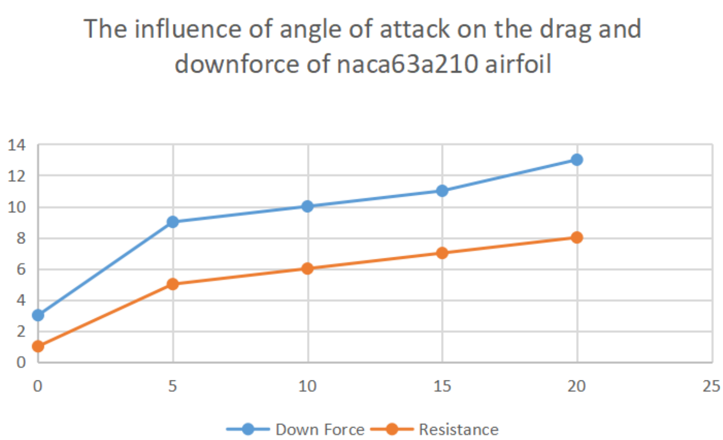
The calculation method for numerical simulation of flow fields generally adopts the method of solving N-S equations. However, in the actual solving process, due to the strong nonlinearity of N-S equations, direct numerical solution is difficult. Therefore, commercial numerical simulation software Ansys Fluent is used as the solver, and the Reynolds stress averaging method is used to solve turbulence, thereby obtaining more reliable flow field simulation results.

Use the steady-state pressure based solver in Fluent to solve the problem, with velocity boundary conditions at the inlet and pressure boundary conditions (atmospheric pressure) at the outlet. The total number of grids is 2.5 million, and the coupled algorithm is used. The turbulence model uses the SST k-omega method. Convergence criterion 1e-6

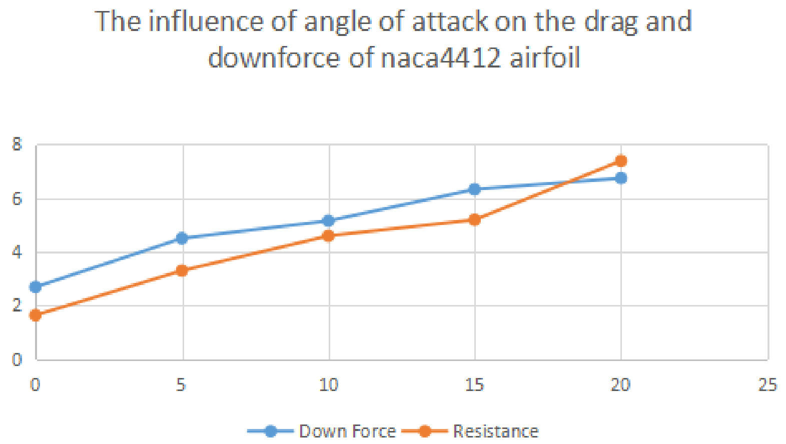
4. Experimental Result

Calculation results and analysis. Firstly, specify the benchmark operating conditions for 4 types of airfoils, plan to calculate 50m/s, and then calculate 5 attack angles of 0 °, 5 °, 10 °, 15 °, and 20 °.[5] A total of 20 types of data need to be calculated. Statistical cloud map for each operating condition, including pressure and resistance, can be compiled into a table. Draw a graph based on the obtained data.

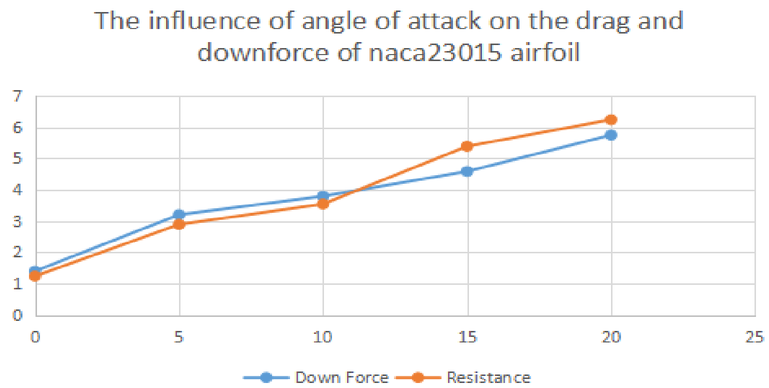
[a]NACA.airfoil.63a210



[b]NACA.airfoil.4412



[c]NACA.airfoil.23015



[d]NACA.airfoil.23012

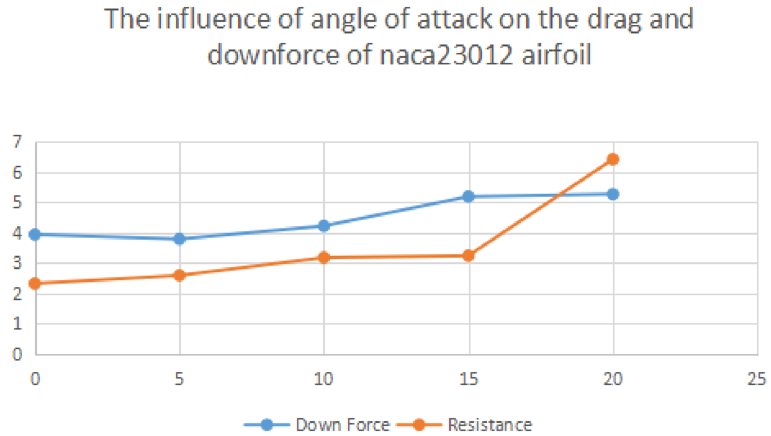


Figure 3. Down force and Resistance

As can be seen from Fig. 3, the downforce and resistance of the four wings increase with the angle of attack. But the trends are different for each. Detailed results please see the sixth point of this paper: Data Analysis

5. Conclusion

After modeling and numerical simulation of different wing structures, the following conclusions can be obtained:

(1) Quantitative conclusions should be used as much as possible. Within the speed range studied in this article, both downforce and drag increase with increasing speed, while drag and downforce increase with increasing angle of attack.

(2) The comparison results between different airfoils show that at a constant speed, the NACA23012 airfoil has the highest lift to drag ratio within 0 to 15 °, which can provide greater downforce while minimizing drag. Therefore, it is believed that the airfoil structure selection within 0 to 15 ° is the best. The NACA63A210 airfoil has the highest lift to drag ratio at 0 °, which is the optimal structural choice for the same airfoil with different angles of attack. The NACA4412 airfoil has the highest lift to drag ratio at 10 °, so the optimal structural choice for this wing is 10 °. When the angle of attack of NACA23012 is 15 °, there is a maximum lift to drag ratio. When the angle of attack of the NACA23015 airfoil is 5 °, there is a maximum lift to drag ratio.

References

[1] Yu Kainan, Xie Shibin. Research on CFD based FSAE racing tail design and optimization [J]. Mechanical and Electrical Engineering, 2018, 35 (01): 16-21.

[2] Cheng Jie, Fang Zheng, Zhang Wei. Comparative study of CFD and potential flow numerical simulation methods for the flow field around wing profiles [J]. Hydrodynamics Research and Progress Series A, 2023, 38 (05): 677-682. DOI: 10.16076/j.cnki. cjhd.2023.05.004

[3] Zhou Tao, Zeng Zhong Aerodynamic Optimization Design of FSAE Racing New Curved Front Wing and Tail Wing [J]. Journal of Chongqing University, 2017, 40 (10): 40-52

[4] Tian Tian, Li Gang, Zhang Zhiqiang, etc FSAC racing car body empty sleeve design and simulation analysis [J]. Journal of Liaoning University of Technology (Natural Science Edition), 2023, 43 (04): 221-227. DOI: 10.15916/j.issn1674-3261.2021.04.003

[5] Zhang Yingchao, Jiang Qingwen, Gong Hongyu, et al. The mutual influence between the ta il wing and aerodynamic characteristics of a racing car [C]//China Society of Automotiv e Engineers.Automotive Aerodynamics Committee of China SAE. Proceedings of the 20 22 Academic Annual Meeting of the Automotive Aerodynamics Committee of the Chine se Society of Automotive Engineers - Aerodynamics Venue. State Key Laboratory of A utomotive Simulation and Control, Jilin University;, 2022:13.DOI:10.26914/c.cnkihy.2022. 035139.