Numerical Study on Aerodynamic Characteristics of Wings of Passenger Aircraft under High Subsonic Conditions

Xiao Wang *

School of Optical Information and Energy Engineering, Wuhan Institute of Technology, 430079, China

* Corresponding author: 2110120217@stu.wit.edu.cn

Abstract. In aerospace engineering, optimizing aerodynamic efficiency under high subsonic flight conditions is paramount. Traditional airfoil research and design methodologies have exhibited limitations, particularly in their efficacy at speeds approaching high subsonic speed, where drag reduction and lift enhancement become critical for performance optimization. These conventional approaches often fall short of addressing the nuanced aerodynamic challenges presented by high subsonic regimes. Addressing these limitations, the present research adopts a computational fluid dynamics (CFD) approach, utilizing ANSYS Fluent 2024 R1 to conduct a detailed numerical analysis of the NACA 0012 airfoil's performance. The study leverages Reynolds-averaged Navier-Stokes (RANS) equations to explore the aerodynamic characteristics of the airfoil, focusing on lift and drag behaviors and flow separation under various flight conditions. Therefore, this methodological framework enables a comprehensive evaluation of the airfoil’s aerodynamic efficiency in high subsonic flight. The findings reveal significant insights into the NACA 0012 airfoil’s aerodynamic properties, demonstrating its potential for drag reduction and lift optimization at critical flight speeds. Notably, the research elucidates the airfoil's ability to maintain aerodynamic stability and efficiency, highlighting its suitability for applications in modern aircraft designs aimed at high subsonic operations. The implications of this study extend beyond the specific aerodynamic performance of the NACA 0012 airfoil. By advancing the understanding of airfoil behavior in near-sonic conditions, this research can inform future airfoil design and optimization efforts, fostering the development of more efficient and environmentally sustainable aviation technologies.

Keywords: CFD, High subsonic conditions, NACA0012.

1. Introduction

The quest for enhancing aerodynamic efficiency has perennially driven advancements in aircraft wing design. Traditional wing aerodynamic design methods have encountered limitations in the domain of civil aviation, particularly in optimizing performance under high subsonic conditions [1]. This paper delves into the numerical study of wings on passenger aircraft, a design that emerges as a solution to the constraints faced by conventional wing designs.

Traditional wing designs are challenged by the increase in drag at speeds approaching the speed of sound, which limits their efficiency in high subsonic flight regimes. Some specific wing shapes (NACA 0012), however, offer a breakthrough in this aspect. They are engineered to delay the onset of shock waves and reduce drag at these critical speeds. Unlike traditional designs, these wings maintain their thickness ratio and sweep angle but enhance the drag divergence Mach number [2]. For a given drag divergence Mach number and sweep angle, it allows for a thicker wing profile, increasing wing volume and potentially decreasing wing weight or enhancing aspect ratio. Furthermore, with a specified drag divergence Mach number and thickness ratio, the sweep angle can be reduced, augmenting maximum lift and improving lift-to-drag ratio during takeoff and landing. This adjustment leads to an optimized design cruise lift coefficient and, for a given aspect ratio, a lighter wing structure. The comparative advantages are evident in improved aerodynamic performance and operational efficiency.

Recent studies have underscored the significance of this kind of wings in addressing the aerodynamic challenges of modern aviation. A well-designed wing must be evaluated against many design objectives and constraints, such as lift-to-drag ratio, stall performance, jitter boundaries, and fuel tank capacity [3]. NACA 0012 wing experiences lower drag and produces less intense shock waves due to its wing shape, which affects the maneuverability of the aircraft and the amount of lift it can experience in flight [4].

The practical implementation of this type of wings in commercial aviation is a testament to their advantages. Major aircraft manufacturers have integrated these wing designs into their latest models, aiming to achieve higher efficiency, reduced fuel consumption, and improved overall performance [5]. This adoption signifies a shift towards more sustainable and economically viable aviation practices.

This paper proposes a numerical approach to studying the aerodynamic characteristics of these wings on passenger aircraft under high subsonic conditions. Through CFD simulations, this paper aims to analyze the performance of these wings in comparison to traditional designs, focusing on aspects such as drag reduction, lift enhancement, and overall aerodynamic efficiency. The study intends to contribute to the body of knowledge by providing insights into optimizing wing design for modern aircraft.

2. Methodology

This study adopts a comprehensive approach to investigate the aerodynamic characteristics of wings on passenger aircraft under high subsonic conditions. The methodology encompasses deriving fundamental aerodynamic equations, the development of computational models, simulation analysis, and model validation processes.

2.1. Fundamental Aerodynamic Equations

The analysis begins with the establishment of the basic aerodynamic equations that govern the flow around wings [6]. These include the Navier-Stokes equations for incompressible flow, fundamental to understanding fluid dynamics and aerodynamics in aviation. The equations are presented as follows:

Continuity Equation:

$$\nabla \cdot \mathrm{V} = 0 \tag{1}$$

Momentum Equation:

$$P(\partial t\, \partial \mathbf{V} + \mathbf{V} \cdot \nabla \mathbf{V}) = -\nabla p + \mu \nabla 2\mathbf{V} \tag{2}$$

Where V represents the velocity field, ρ the density of the air, p the pressure, and μ the dynamic viscosity of the air.

2.2. Model Development

The aerodynamic model of the wing is developed using Computer-Aided Design software. The model incorporates specific geometrical features of wings, such as their unique leading and trailing edge shapes, designed to optimize aerodynamic performance at high subsonic speeds. These geometrical models are then imported into CFD software for further analysis.

Firstly, by observing and analysing the standard airfoils, researchers constructed a 3D airfoil structure as shown in Figure 1.

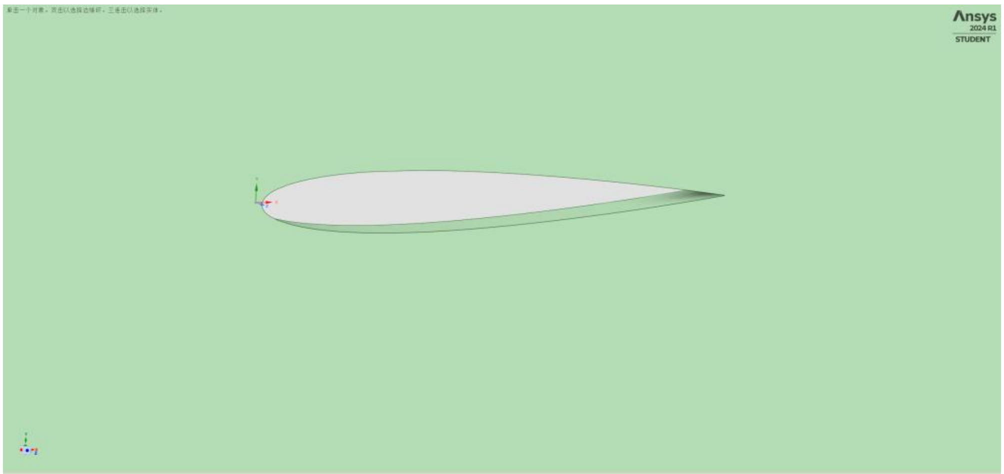

Figure 1. Basic wing modeling.

The simulation scenario is at Mach 0.7 speed, the angle of attack is 1.55 degrees, and the airfoil geometry used is NACA 0012. computational domain dimensions are shown in Figure 2.

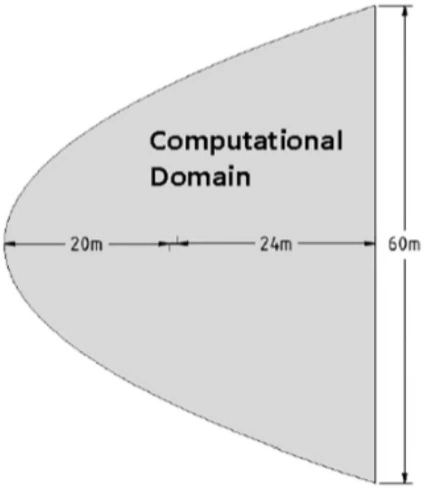

Figure 2. Computational domain.

Experimental parameters:

P0=total pressure=101325Pa

T0=total pressure=311K

According to the formula

$$\left[1+\left(\frac{\gamma-1}{2}\right)M^2\right]^{\frac{\gamma}{\gamma-1}}=\frac{P_0}{P} \tag{3}$$

$$\frac{T_0}{T}=1+\left(\frac{\gamma-1}{2}\right)M^2 \tag{4}$$

And γ=1.4 for air

M=Mach number=0.7

The boundary conditions are shown in Table 1.

Table 1. The boundary conditions

Parameter	Values
P0	101325Pa
T0	311K
P	73048Pa
T	283.24K

2.3. Simulation Analysis

The CFD simulations are performed using ANSYS Fluent 2024 R1, which can solve the RANS equations. The simulation process involves setting up the computational domain, mesh generation, boundary condition specification, and the selection of appropriate turbulence models to accurately capture the flow physics around the wings [7]. The simulations aim to analyze the lift, drag, and flow separation characteristics of the wings under various flight conditions.

After completing the basic model construction and boundary condition calculation, the authors completed the simulation analysis according to the following steps.

2.3.1. Pre-processing

The geometric model of the wing, developed during the model development phase, is imported into ANSYS Fluent.

A high-quality mesh is generated to resolve the boundary layer and critical flow features around the wing. This paper used Proximity as a sizing function to populate the mesh in the narrow region of the wing tail, ensuring that at least two meshes are present, allowing the narrow region flow features to be captured properly by the software. Mesh refinement is also applied near the wing surface and in wake regions to capture the intricate flow patterns and potential separation points accurately.

Boundary conditions are defined to simulate the actual high subsonic flight environment [8]. These include the velocity inlet, pressure outlet, and no-slip wall conditions on the wing surface. Turbulence is modeled using the Shear Stress Transport (SST) k-omega model, known for its accuracy in predicting adverse pressure gradients and separating flows over airfoils. Also, set the geometry to consist only of regions of fluid that are free of air strikes. Set the boundary layer mesh to a uniform and set the number of grid layers to 20. The result is shown in Figure 3.

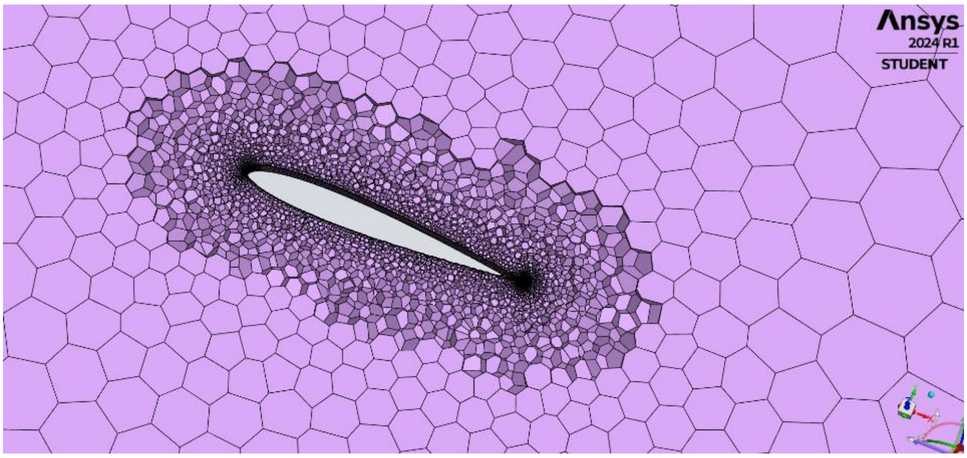

Figure 3. Grid partitioning of the final model.

2.3.2. Solver setup

Turbulence uses the SST-based k-omega model [9]. Set ambient air as an ideal gas, viscosity using Sutherland's three coefficient method. Fill in the pressure and temperature values as described in the previous section. Set the Z component of the flow direction to 0, then set the X component to Cos (angle of attack) and the Y component to Sin (angle of attack) according to the value of the angle of attack.

Convergence is monitored through residuals for continuity, velocity components, and turbulence quantities. A convergence criterion of less than 1e-5 for all residuals is typically set to ensure a sufficiently accurate solution [10]. Additionally, physical parameters such as lift and drag coefficients are monitored to ensure their stabilization, indicating a converged solution.

The lift and drag coefficients are calculated based on the pressure and shear stress distributions on the wing surface. These metrics are essential for evaluating the aerodynamic efficiency of the wing design under various flight conditions.

3. Results and Discussion

The results of the fluid simulation analysis are shown below, with scaled residuals in Figure 4, drag in Figure 5, and lift in Figure 6.

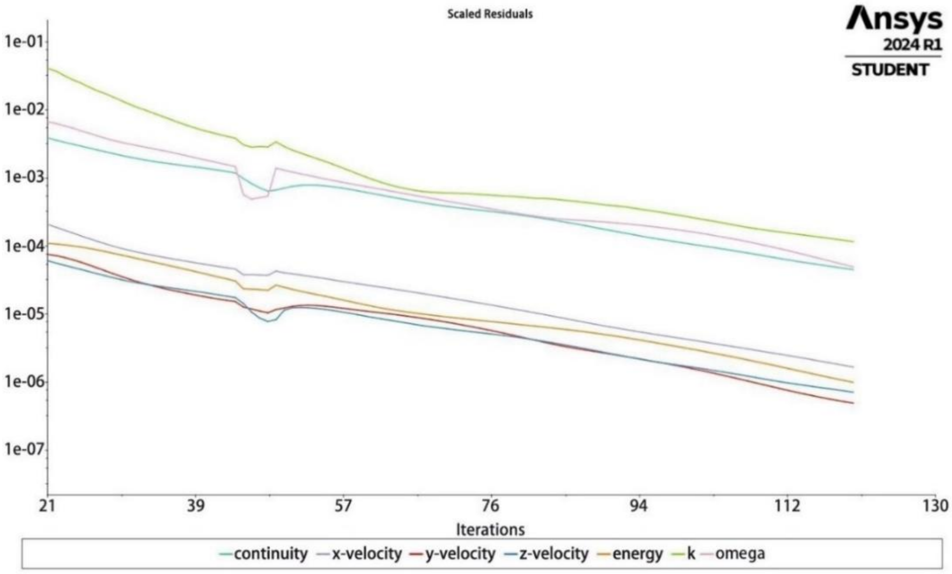

Figure 4. Scaled residuals.

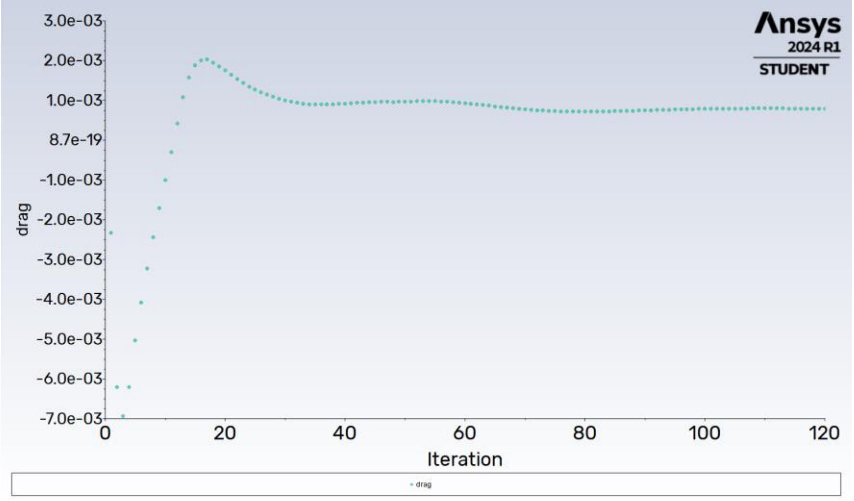

Figure 5. Drag coefficient.

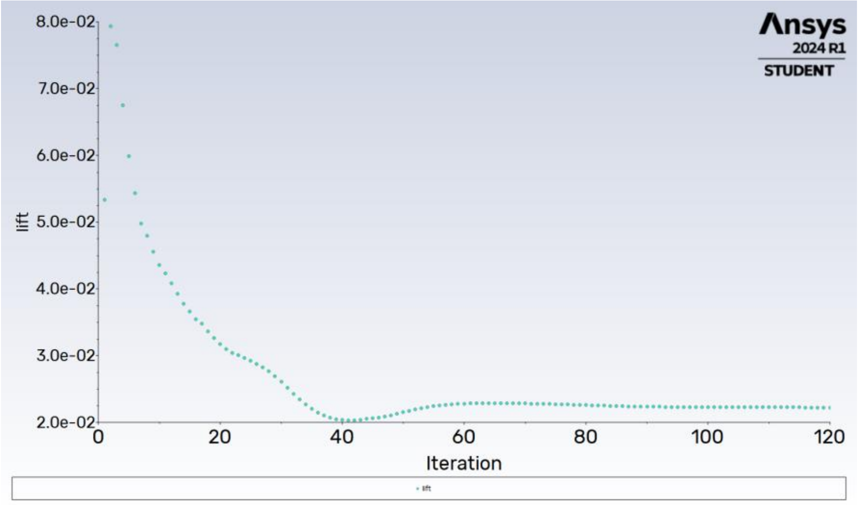

Figure 6. Lift coefficient trend.

3.1. Lift Coefficient Trend

The lift coefficient trend (Figure 6) shows an initial peak followed by a rapid decrease, stabilizing as the iterations proceed. This is typical of the transient effects during the start of the simulation settling down as the flow field around the wing reaches a steady state. The stabilized lift coefficient is within the expected range for wings, confirming that the simulated wing profile is capable of generating the necessary lift for flight at high subsonic speeds.

3.2. Drag Coefficient Trend

The drag coefficient (Figure 5) displays a negative spike initially, an artifact of the solver initialization, and not a physical phenomenon. As the iterations continue, the drag coefficient stabilizes to a positive value. The low magnitude of the steady-state drag coefficient reflects the aerodynamic efficiency of the wing design, which is a critical factor in reducing fuel consumption and enhancing overall aircraft performance.

3.3. Scaled Residuals

The scaled residuals (Figure 4) provide insight into the convergence behavior of the simulation. All residuals have fallen below 1e-4, a typical benchmark for convergence in aerodynamic simulations. The gradual descent of residuals towards a plateau indicates that a numerically stable solution has been reached, adding confidence to the validity of the simulation results.

3.4. Qualitative Inorganic Analysis

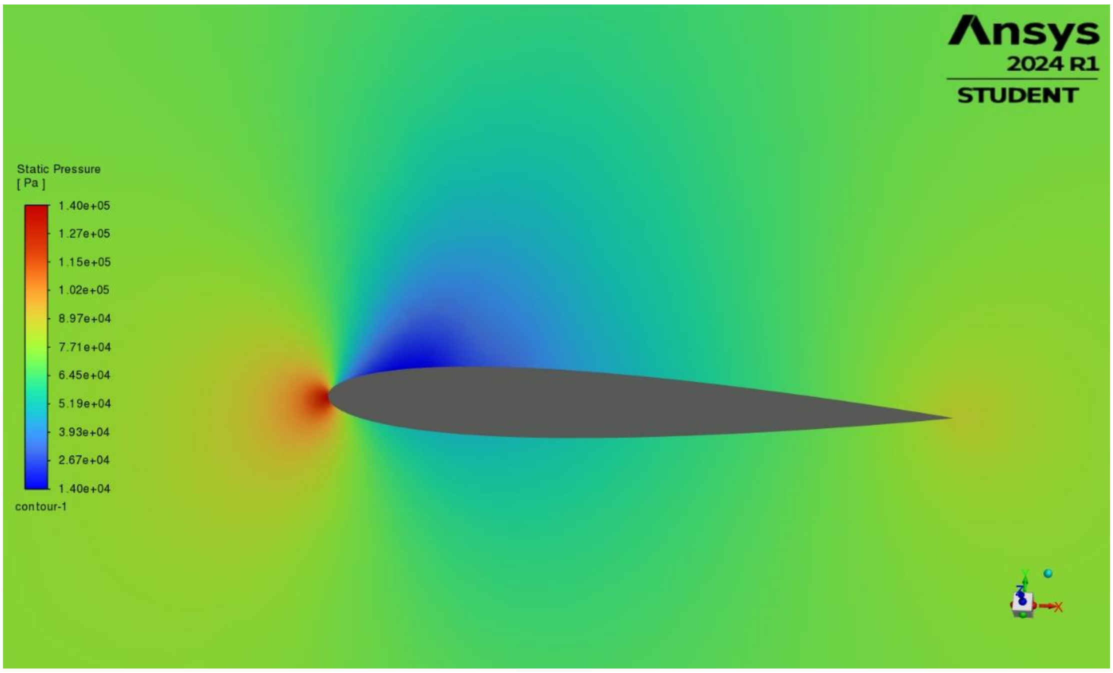

Figure 7. Wing Surface Pressure Distribution.

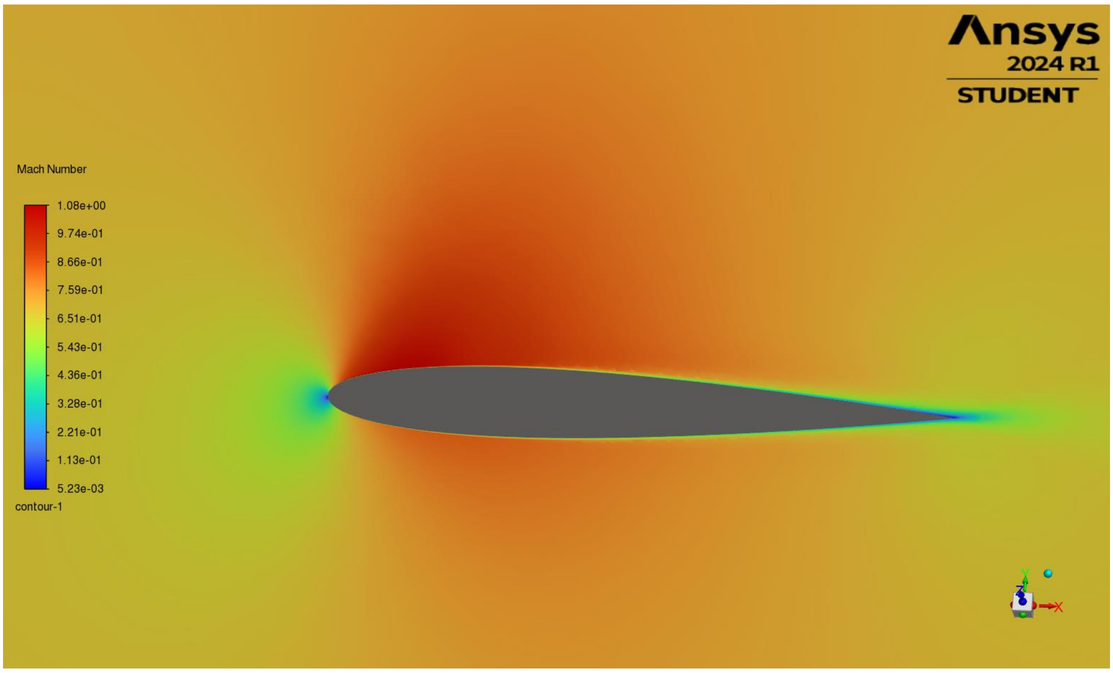

Figure 8. March number distribution.

3.5. Mach Number Distribution

The Mach number distribution around the wing (Figure 7) reveals a smooth acceleration of the airflow over the upper surface, with a Mach number gradually increasing towards the trailing edge before the sharp drop, indicative of a well-managed shock wave. This is a characteristic behavior of standard airfoils designed to delay the onset of shock-induced drag. The presence of a shock wave is evidenced by the abrupt color change, but its rearward location suggests that the wing is effective in reducing drag while maintaining high lift. Static Pressure Distribution.

The static pressure distribution on the wing (Figure 8) complements the Mach number findings. The low-pressure region over the upper surface confirms the generation of lift. The high-pressure region on the lower surface and the smooth pressure recovery towards the trailing edge indicate an efficient aerodynamic design, minimizing adverse pressure gradients and the risk of flow separation.

The simulation results provide insightful data on the aerodynamic performance of the wing. The Mach number and pressure distributions confirm that the wing's design is capable of sustaining high subsonic flight with reduced drag due to delayed shock waves. The stabilization of lift and drag coefficients after an initial transient phase indicates a successful adaptation of the flow to the wing's geometry, resulting in a reliable lift force and reduced aerodynamic drag.

The residuals' behavior reflects the accuracy and stability of the numerical method used. The low and stable residuals suggest that the turbulence models and mesh resolution employed are adequate for capturing the essential flow physics around the wing.

From these findings, it can be inferred that the wing's design provides a balance between lift generation and drag reduction, which is vital for the efficiency of high-speed passenger aircraft. The results validate the effectiveness of using wing designs in commercial aviation and underscore the potential benefits of their application in future aircraft designs.

The simulation results were relatively successful, but further validation based on experimental data or established theoretical models is required. The consistency of the aerodynamic performance is demonstrated by the stable Cl and Cd values during the iterations, which highlights the reliability of the simulation setup and the chosen computational model.

The data from this experiment are limited because they were obtained entirely by fluid analysis software calculations and were not validated by effective experiments. Afterward, relevant experimental verification will be carried out based on the fluid simulation results, and the reliability of the results will be further illustrated based on the comparison of the experimental data with the calculated data.

4. Conclusion

This study embarked on an in-depth exploration of the aerodynamic performance of NACA 0012 airfoil geometry under high subsonic flight conditions, employing a numerical approach through CFD simulations. This research aimed to tackle the aerodynamic optimization challenges inherent in traditional wing designs, specifically by investigating the performance capabilities of the NACA 0012 airfoil. This work assesses the potential of this airfoil to enhance aerodynamic efficiency, particularly in environments approaching the speed of sound, where drag reduction and lift enhancement are crucial.

The methodology comprised the derivation of fundamental aerodynamic equations, the development of computational models, and detailed simulation analysis. By utilizing ANSYS Fluent 2024 R1 for simulations, this study sought to compare the aerodynamic performance of the NACA 0012 airfoil against conventional wing designs, with a focus on aspects such as drag reduction, lift augmentation, and overall aerodynamic efficiency. The study rigorously evaluated the lift, drag, and flow separation characteristics under various flight conditions, thereby contributing significant insights into the optimization of airfoil design for modern aircraft.

The simulation results revealed the NACA 0012 airfoil's superior aerodynamic performance. Notably, the airfoil demonstrated an ability to generate the necessary lift for high subsonic flight while maintaining a reduced drag coefficient, indicative of its efficiency. The Mach number and static pressure distributions confirmed the airfoil's capability to sustain flight with minimal drag, due to its optimized shape which effectively manages airflow. Furthermore, the stable lift and drag coefficients, alongside low and stable residuals, underscored the accuracy and reliability of the simulation methodology. These findings validate the advantages of the NACA 0012 airfoil in improving operational efficiency and highlight its potential in contributing to sustainable and economically viable aviation practices.

This research underscores the significant value and implications of employing the NACA 0012 airfoil in commercial aviation, indicating a move towards more efficient, less fuel-consuming, and environmentally friendly aircraft designs. The adoption of this airfoil geometry by aircraft manufacturers in their latest models could signify their acknowledgment of its benefits, aligning with the industry's trend toward sustainability.

Despite the promising outcomes, this study acknowledges certain limitations, including the necessity for further validation through experimental data or established theoretical models to bolster the applicability and accuracy of the findings comprehensively. The research scope was focused on high subsonic conditions, suggesting room for future studies to examine the NACA 0012 airfoil's performance across a broader spectrum of flight conditions.

Looking ahead, this paper recommends continued research in this field, with an emphasis on experimental validations to support and extend the simulation results. Broadening the investigation to encompass various flight regimes and airfoil configurations could uncover additional enhancements in airfoil design. Furthermore, assessing the impact of the NACA 0012 airfoil on other aircraft performance metrics, such as stability and control, would offer a more comprehensive understanding of its benefits. This study contributes to the ongoing evolution of airfoil design technology, paving the way for the development of more efficient, sustainable, and economically viable aviation solutions.

References

[1] Kumar A., Hefner J. N., Future challenges and opportunities in aerodynamics. The Aeronautical Journal, 2000, 104 (1038): 365 - 374.

[2] Lock R. C., Fulker J. L., Design of supercritical aerofoils. Aeronautical Quarterly, 1974, 25 (4): 245 - 265.

[3] Zhao T., Zhang Y., Chen H., et al. Supercritical wing design based on airfoil optimization and 2.75 D transformation. Aerospace Science and Technology, 2016, 56: 168 - 182.

[4] Bartlett D. W., Patterson Jr J. C., The NASA supercritical-wing technology. CTOL Transport Technol. Conf. (NASA-TM-78731) 1978.

[5] Okonkwo P., Smith H., Review of evolving trends in blended wing body aircraft design. Progress in Aerospace Sciences, 2016, 82: 1 - 23.

[6] Sobieczky H., Seebass A. R., Supercritical airfoil and wing design. Annual Review of Fluid Mechanics, 1984, 16 (1): 337 - 363.

[7] Patel K. S., Patel S. B., Patel U. B., et al. CFD Analysis of an Aerofoil. International Journal of Engineering Research, 2014, 3 (3): 154 - 158.

[8] Busse C., Kach A. P., Wagner S. M., Boundary conditions: What they are, how to explore them, why we need them, and when to consider them. Organizational Research Methods, 2017, 20 (4): 574 - 609.

[9] Grunloh T. P., Four equation k-omega based turbulence model with algebraic flux for supercritical flows. Annals of Nuclear Energy, 2019, 123: 210 - 221.

[10] Bhaskaran R., Collins L., Introduction to CFD basics. Cornell University-Sibley School of Mechanical and Aerospace Engineering, 2002: 1 - 21.