Journal of Academic Research (Applied Sciences), Vol 28, Issu 2, 2024 "01-15"

Numerical Investigation of NACA-0015 Airfoil Performance Using ANSYS: A Detailed Study of Lift, Drag, and Stall Characteristics

Salem Fathi Elsheltat[1], Abdulbaset Ahmed Alshara[2], Ali Alhussain Altaweel[3], Eman Abdulsalam[4]

[1,3,4]Department of Mechanical Engineering, Faculty of Engineering, Misurata University, Libya. [2]Department of Biomedical Engineering, Faculty of Medical Technology-Misurata, Libya.

Article information	Abstract
Key words *Angle of attack; Computational Fluid Dynamics (CFD); Lift and drag; NACA-0015 airfoil.* *Received 08 09 2024, Accepted 19 09 2024, Available online 21 09 2024*	This article seeks to numerically analyze the aerodynamic performance of the NACA-0015 airfoil blade, with a focus on examining the effects of varying the angle of attack on lift, drag, and stall characteristics. The design of an airplane's wing is critical for maximizing lift while minimizing drag, both of which are regulated by adjusting the angle of attack during flight. To explore these dynamics, Computational Fluid Dynamics (CFD) analysis is employed using ANSYS software, particularly its FLUENT tool, to simulate fluid flow around the airfoil. The airfoil geometry, with a chord length of 0.06 meters and a span of 0.25 meters, is modeled in ANSYS Design Modeler. The CFD simulations are performed using the Realizable k-epsilon turbulence model, analyzing angles of attack ranging from 0° to 18° under low Reynolds numbers (6×10^4 to1.6×10^5). Through this comprehensive approach, the study provides a wider understanding of the flow characteristics over the NACA-0015 airfoil, contributing to the precise knowledge of airfoil performance in aviation applications.

1. Introduction

Aerodynamics is the science that describes the movement of bodies through air, focusing on the forces that act on objects in motion within an airflow or on stationary objects exposed to moving air. As a branch of dynamics, it examines how air and other gases interact with surfaces, influencing their movement and stability. In aviation, aerodynamics is particularly concerned with three key components: the aircraft, the relative wind, and the atmosphere (Raymer 1992).These elements work together to determine the performance and behavior of an aircraft during flight. The movement of an aircraft through the air is determined by the magnitude and direction of various forces acting upon it. According to Newton's Third Law of Motion, "for every action, there is an equal and opposite reaction." When an airplane is airborne, it is influenced by four primary aerodynamic forces: lift and thrust, which assist in flight, and drag and weight, which oppose it (FAA 2023). These forces work in balance to dictate the aircraft's performance and stability in the air. The

salem.sheltat@eng.misuratau.edu.ly

aerodynamic profile of an airfoil significantly influences lift and drag, crucial for efficient aircraft performance. A high lift-to-drag ratio is essential for minimizing thrust requirements, but flow separation can destabilize the wing, increasing drag and reducing lift. Consequently, extensive research focuses on flow control techniques to mitigate separation, enhance lift, and improve flight efficiency. Effective flow control methods can reduce skin friction and drag, leading to benefits like increased payload, reduced fuel consumption, and improved landing capabilities, with potential drag reduction of up to 15% by maintaining laminar flow (Cengel and Turner 2004). Uddin and colleagues (M. Uddin et al. 2015) conducted both numerical and experimental investigations on the NACA-0015 airfoil, aiming to validate and compare its aerodynamic characteristics across methods. Similarly, (Rasal and Katwate 2017) explored flow separation control on the NACA0012 airfoil by adding dimples of varying sizes (1%, 2%, and 3% of chord length). They found that a 3% dimple size significantly enhanced performance. (Rubel et al. 2016) experimentally and numerically analyzed the performance of NACA-0015 and NACA-4415 airfoils across a range of angles of attack, focusing on flow separation dynamics, lift, drag, and pressure contours. Costes and his team (Costes et al. 2005) performed a grid convergence study on the NACA-0015 airfoil at static angles of attack, revealing that very fine meshes were necessary to achieve convergence, though both CFD methods (elsA and CFL3D) underestimated experimental flow separation. CFD has been extensively used to investigate airfoil performance across different flow conditions. A study by (El Maani, Radi, and El Hami 2018) conducted CFD analysis of the transonic flow over a NACA 0012 airfoil, demonstrating the accuracy of numerical simulations compared to experimental data. The study emphasized the importance of turbulence modeling, particularly using the k-Omega SST model, to accurately predict lift, drag, and pressure coefficients under high-speed compressible flows. In a subsequent study, (Teruna et al. 2021) explored the optimization of airfoil shape using the Class Shape Transformation (CST) method combined with Genetic Algorithms (GA). The research focused on reducing drag and improving the lift-to-drag ratio for airfoils used in wind turbine applications. The optimization process showed that the aerodynamic performance of the modified airfoils significantly improved compared to the original configurations, highlighting the impact of shape optimization in enhancing energy efficiency. (Akram and Kim 2021) extended this approach by integrating CST with GA to optimize the aerodynamic shapes of subsonic and transonic airfoils. Their study reported that optimized airfoils achieved significant improvements in the lift-to-drag ratio, with reductions in drag by 10-12% and improvements in aerodynamic stability. The research underscored the importance of CFD as a tool for airfoil design and performance enhancement in aerospace and renewable energy sectors.

In the work of (Khalid 2022), CFD analysis was performed on a NACA 0012 airfoil using Ansys Fluent to examine how increasing the angle of attack affects the lift coefficient. The airfoil geometry was designed in SOLIDWORKS, and the K-omega turbulence model was used. Simulations were conducted at a speed of 32 m/s for angles of attack ranging from 0° to 20°. Their results showed that the lift coefficient increases with the angle of attack up to a certain point, after which flow separation occurs, causing the lift coefficient to decrease. The findings were then validated against existing literature. Also, very recent research in

(2024) has utilized various numerical analysis methods to enhance aerodynamic performance and study flow characteristics of airfoils. Specifically, CFD simulations have been applied to NACA 0012 and S809 airfoils to explore dynamic stall behavior and flow separation control through synthetic jets. Additionally, studies have assessed the effects of design modifications such as leading-edge slots and textured surfaces to improve lift-to-drag ratios, benefiting applications in wind turbines and aircraft (S. M. N. Uddin et al. 2024).

In our study, we will provide a comprehensive analysis of flow characteristics over the NACA-0015 airfoil blade as a function of the angle of attack. The research will be complemented by numerical analysis using ANSYS Software. The primary goal is to determine the lift and drag forces at different angles of attack and airspeeds, as well as to identify stall conditions at various speeds.

2. Identification of this research

This research aims to analyze the aerodynamic performance of the NACA-0015 airfoil blade by investigating the impact of varying the angle of attack on lift, drag, and stall characteristics. The study is driven by the critical role of airfoil design in optimizing aircraft wing performance, specifically in maximizing lift while minimizing drag. To explore these aerodynamic properties, Computational Fluid Dynamic (CFD) simulations are conducted using ANSYS FLUENT to model fluid flow around the airfoil. The geometry of the airfoil, with a chord length of 0.06 meters and a span of 0.25 meters, is developed in ANSYS Design Modeler, and the simulations employ the Realizable k-epsilon turbulence model across angles of attack from 0° to 18° at low Reynolds numbers (6×10^4 to 1.6×10^5). The study simulates airflow around the airfoil by solving conservation equations of mass and momentum under specific flow conditions, including an inlet velocity defined at 0° AOA. The comprehensive approach ingrating CFD analysis in ANSYS provides valuable insights into the aerodynamic performance of the NACA-0015 airfoil, contributing to the precise understanding of airfoil behavior in aviation applications.

3. Key factors influencing performance

3.1 Reynolds's number

The Reynolds number (Re) is a dimensionless parameter that indicates whether fluid flow is laminar or turbulent, defined as in equation (1).

$$Re = \frac{\rho V L}{\mu} \qquad (1)$$

where ρ is fluid density, V is flow velocity, L is characteristic length, and μ is fluid viscosity. The Reynolds number compares inertial forces to viscous forces, highlighting the role of viscosity in fluid flow. High Reynolds numbers result from increased flow velocity, larger characteristic lengths, or reduced viscosity. In flow over a flat plate or airfoil, the flow starts as laminar and transitions to turbulent around a critical Reynolds number of 1×10^5, becoming fully turbulent near 3×10^6 (John D. Anderson Jr. 1995).

3.2 Lift and Drag

Lift is the force acting perpendicular to the direction of flow, generated when an object alters the direction of fluid flow, such as an airfoil deflecting air. It depends on factors like flow velocity, air properties, wing geometry, and angle of attack (AOA)—the angle between the airfoil's chord line and the relative wind. As air flows over an airfoil, a low-pressure zone forms on the upper surface and a high-pressure zone below, creating a pressure difference that produces lift. Drag, on the other hand, opposes motion through the air and occurs whenever lift is generated. It is caused by air resistance, skin friction, and the byproducts of lift, acting parallel to the relative wind and opposite to thrust. Smoother surfaces reduce drag, while larger structures increase it, requiring engine power to overcome and maintain flight.Figure 1 illustrates a free body diagram of an airfoil subjected to lift and drag forces (John D 2017; Raymer 1992; FAA 2023).

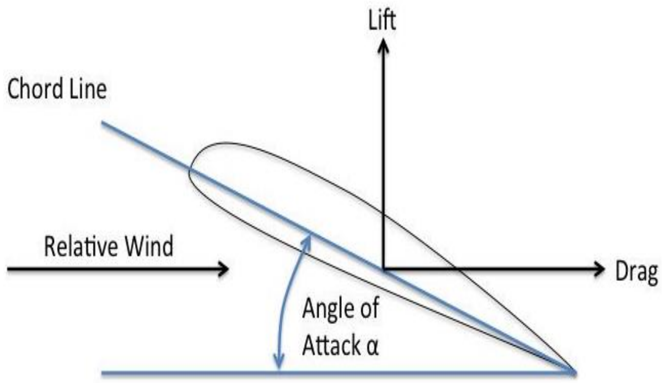

Figure 1 Free body diagram of airfoil.

3.3 Weight and Thrust

The weight of an airplane is determined by its size, materials, payload, and fuel, and it always acts downward due to gravity. For an aircraft to take off, lift must overcome this weight. According to Newton's First Law, an object remains in its state of motion unless acted upon by an external force—here, lift is that force, generated by increasing the angle of attack (AOA) or speed. Thrust, produced by engines, propels the aircraft forward; as the wings cut through the air, lift is generated, pushing the airplane upward (John D 2017; Raymer 1992; FAA 2023).

3.4 Lift and drag coefficients (C_L , C_D)

The lift coefficient (C_L)and drag coefficient (C_D) are dimensionless numbers used to model how shape, inclination, and flow conditions affect lift and drag forces. The lift coefficient is defined as in equation (2):

$$C_L = \frac{L}{\rho * 0.5 * V^2 * A} \qquad (2)$$

where L is the lift force, ρ is air density, V is velocity, and A is wing area. Similarly, the drag coefficient is defined as in equation (3):

$$C_D = \frac{\boldsymbol{D}}{\rho * 0.5 * V^2 * A} \qquad (3)$$

where D is the drag force. Both coefficients express the ratio of the force to dynamic pressure(q) times the reference area. Thus, they can be simplified as in equation (4 and 5):

$$C_L = \frac{\boldsymbol{L}}{q * A} \qquad (4)$$

$$C_D = \frac{\boldsymbol{D}}{q * A} \qquad (5)$$

Lift is perpendicular to the flight path, while drag is parallel. The lift-to-drag ratio (L/D) indicates aerodynamic efficiency: high L/D means more lift or less drag. In cruise, lift equals weight and thrust equals drag; high lift allows for larger payloads, while low drag reduces thrust requirements (John D 2017; Raymer 1992; FAA 2023).

3.5 Stall

Stall occurs when the wing's angle of attack exceeds the critical point, causing airflow separation and a sudden loss of lift. Stall behavior varies by airfoil; some experience gradual lift reduction, while others see a sharp drop with rapid changes in pitching moment. High angles of attack prevent flow reattachment, leading to an abrupt loss of lift. Airfoil thickness also affects drag, lift, stall behavior, and structural weight. Stall recovery generally involves lowering the nose, leveling the wings, and increasing power to restore airflow over the wing (Raymer 1992).

4. CFD Model

Computational Fluid Dynamics (CFD) numerically approximates the equations governing fluid motion by converting complex partial differential equations into solvable algebraic equations. With advances in computing power since the 1950s, CFD has become essential for simulating conditions that are difficult to test experimentally or solve analytically. CFD codes consist of three main components: (1) a pre-processor for inputting geometry, generating grids, and setting flow parameters and boundary conditions; (2) a flow solver for solving the governing equations; and (3) a post-processor for visualizing results (John D. Anderson Jr. 1995). ANSYS Fluent offers extensive modeling capabilities for flow, turbulence, heat transfer, and chemical reactions, applicable to diverse fields like aerospace, combustion, and bioengineering. It includes specialized models for in-cylinder combustion, aeroacoustics, turbomachinery, and multiphase systems (ANSYS 2018). This study uses ANSYS Fluent for numerical analysis.

4.1 Geometry

The NACA-0015 airfoil coordinates used in this study were sourced from the NACA website (Figure2 a),and imported into ANSYS DesignModeler (Figure2 b).

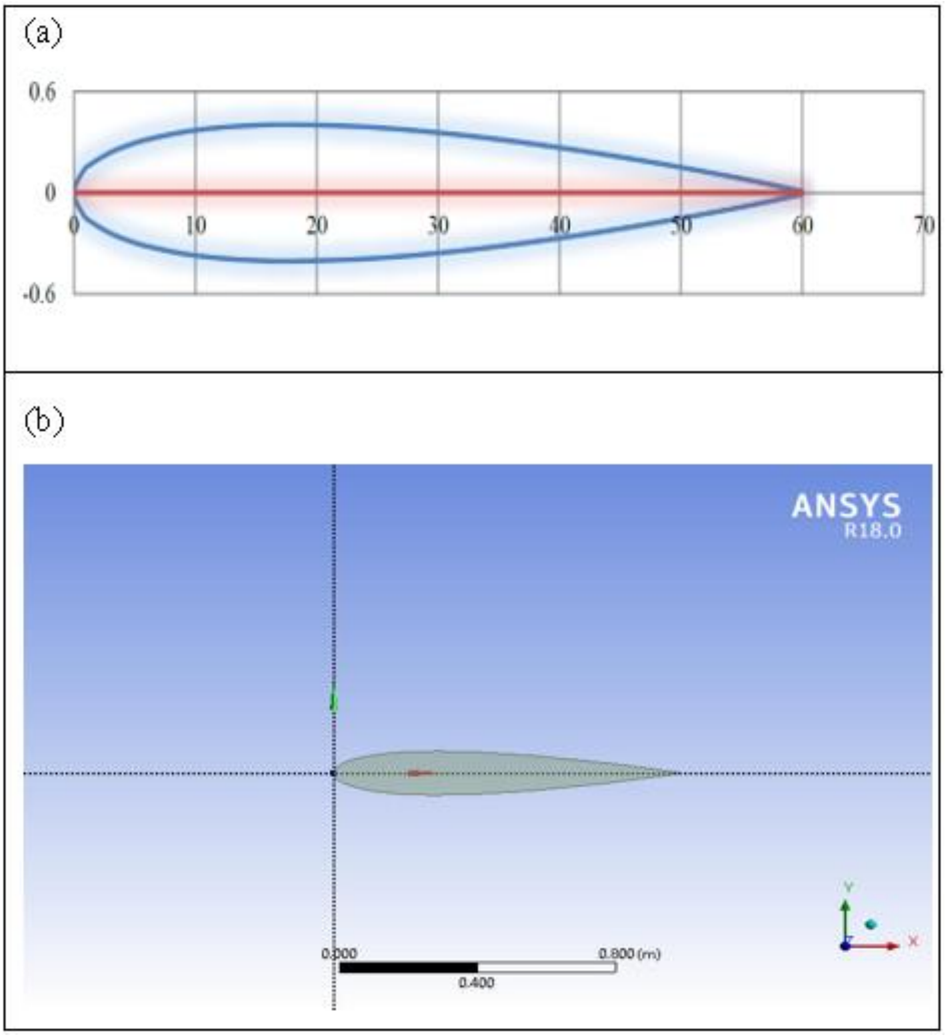

Figure2 Geometry of the airfoil . (a) Chord length and thickness (mm) with (b) Airfoil in DesigneModeler.

4.2 Mesh and model Setup

To analyze the fluid flow around the airfoil, a C-mesh technique was used, comprising an arc with a 75 mm radius and a 75 x 75 mm rectangular grid. The mesh was refined with fine relevance settings and high smoothing to enhance accuracy, particularly around complex areas like the trailing edge. The final mesh, with 14,099 nodes and 13,792 elements, ensures detailed coverage of the airfoil for accurate CFD analysis. The meshed geometry was then imported into FLUENT to solve the coupled momentum and pressure-based continuity equations in a 2D planar space. Viscous-Realizable k-epsilon modeling with Reynolds-averaged Navier–Stokes (RANS) equations was used. The airfoil material was set to aluminum, and the fluid was ideal air with a density of 1.186 kg/m³ and a dynamic viscosity of 1.84 x 10^{-5} kg/(m·s) at 288.15 K.

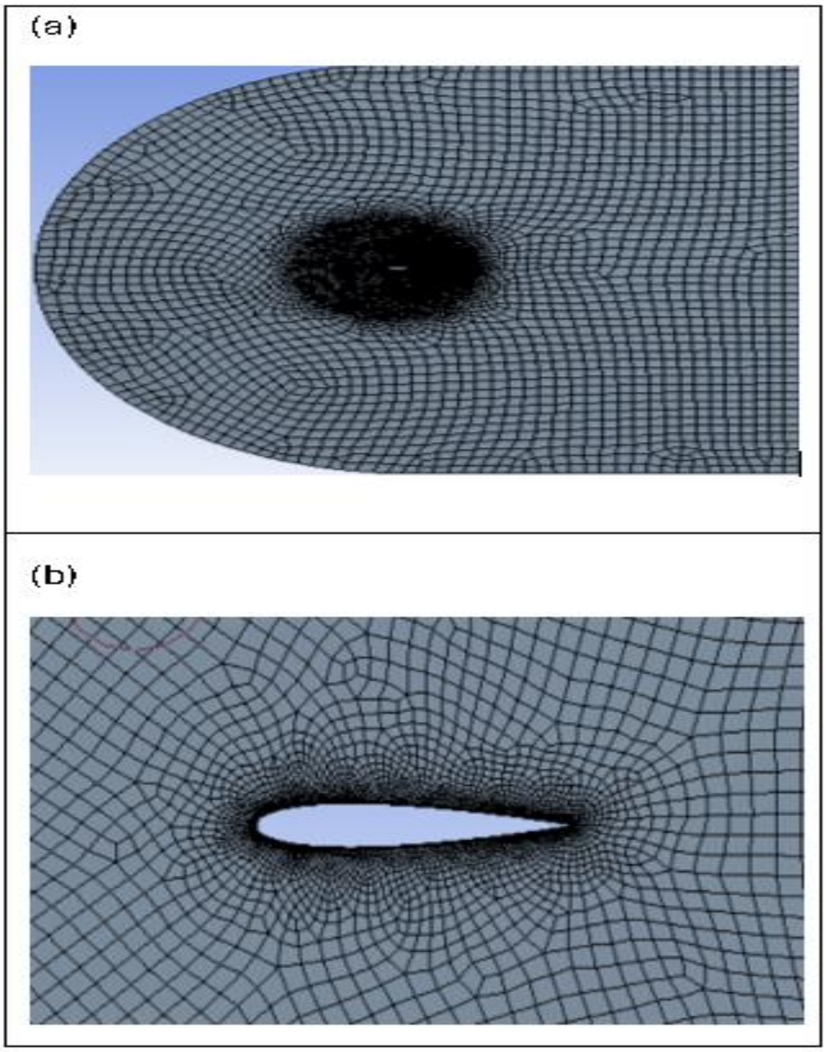

Figure 3 Mesh of the airfoil . (a) c-mesh around the airfoil with (b) Detailed view of the mesh.

4.3 Numrical Soultion

The pressure-velocity coupling in the simulation is achieved using the SIMPLE scheme, while second-order upwind discretization is applied for spatial accuracy, ensuring precise interpolation of cell center values from surrounding faces. During the solution process, convergence criteria or a set number of iterations are used to solve the flow equations. Results from FLUENT then can be displayed as mesh files and XY plots. The drag force (F_D) and lift force (F_L) are the horizontal and vertical components of the resultant force (F_R) acting on the airfoil, depending on the angle of attack (AOA). They are defined as in equations (6 and 7):

$$F_D = F_R * \cos(AOA) \qquad (6)$$
$$F_L = F_R * \sin(AOA) \qquad (7)$$

The simulation is conducted over a range of angles of attack from 0° to 18° at varying velocities.

5. Results and discussions

5.1 Angle of attack

The results show the variation of the lift coefficient (C_L) and drag coefficient (C_D) with changes in the angle of attack (AOA) at different velocities. To facilitate analysis, the data from ANSYS was imported into Microsoft Excel and plotted in X-Y charts for C_L and C_D against AOA, as depicted in Figure 4 , and Figure 5.

Figure 4 illustrates that at all tested velocities, the lift coefficient increases nearly linearly with the AOA up to a critical point, where stall occurs. Beyond this critical angle, the lift sharply decreases as flow separation increases, reversing the relationship between lift and AOA. Identifying these critical points is crucial, as exceeding the stall angle can cause a loss of lift, risking aircraft control. Notably, stall occurs at an AOA of 12° for a velocity of 25 m/s, and at 13° for a velocity of 35 m/s.

Figure 5 shows that the drag coefficient gradually increases with the AOA before reaching the stall. Post-stall, C_D rises sharply due to increased flow separation, significantly increasing aerodynamic resistance. To mitigate this undesirable increase in drag, it is recommended to reduce the AOA immediately and increase thrust to regain control and restore optimal flight conditions.

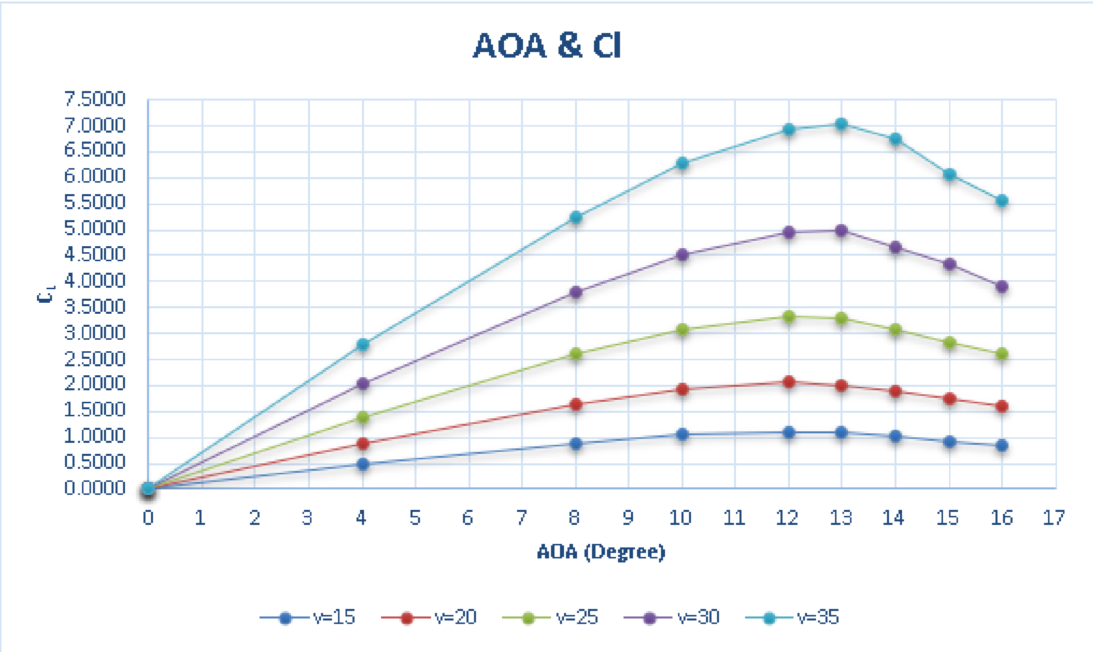

Figure 4 Comparison between lift coefficient, C_L , and the angle of attack, AOA (degree) , at different velocities, v.

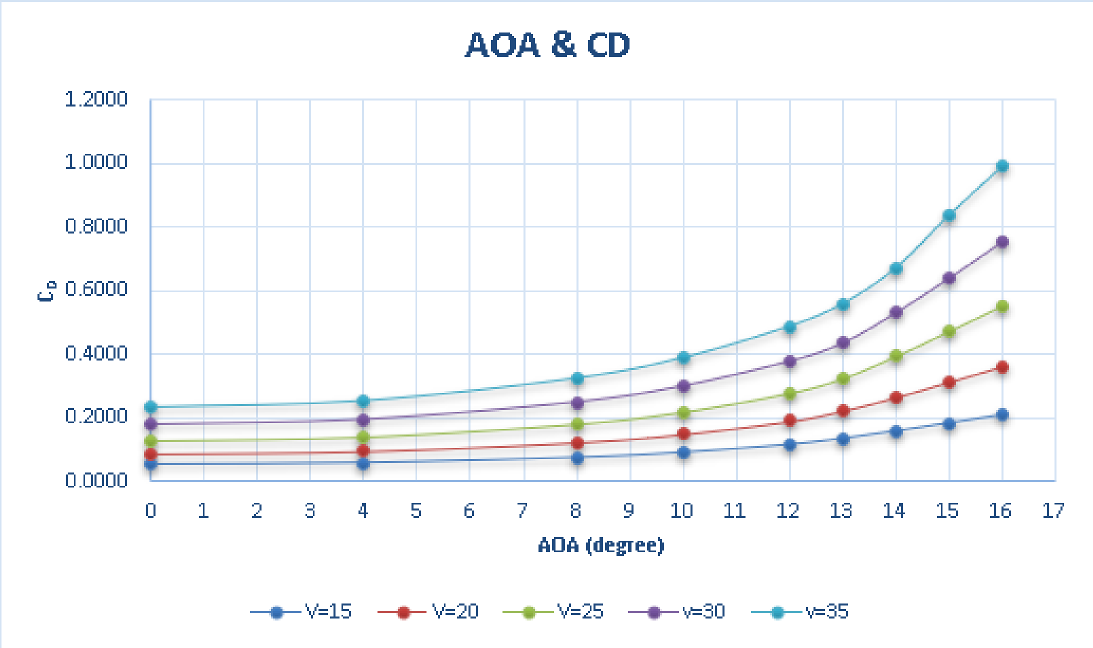

Figure 5 Comparison between drag coefficient, C_D , and the angle of attack, AOA (degree), at different air velocities, v.

5.2 Velocity contours

The simulation of flow around the NACA-0015 symmetric airfoil at a velocity of 25 m/s was conducted using ANSYS FLUENT. The resulting velocity magnitude contours at various angles of attack (AOA) are shown in Figure 6.

In Figure 6a, at zero angle of attack, the velocity streams are symmetrically distributed across the upper and lower surfaces of the airfoil, resulting in no lift force. However, as shown in Figure 6b, c, and d, increasing the angle of attack causes a decrease in velocity along the lower surface, leading to flow separation near the trailing edge, which could indicate the onset of stall. At the leading edge, the velocity difference creates a high-pressure area known as the stagnation point. The blue regions in the figures indicate areas where the velocity drops, particularly at the trailing edge as the angle of attack increases, further contributing to potential stall conditions.

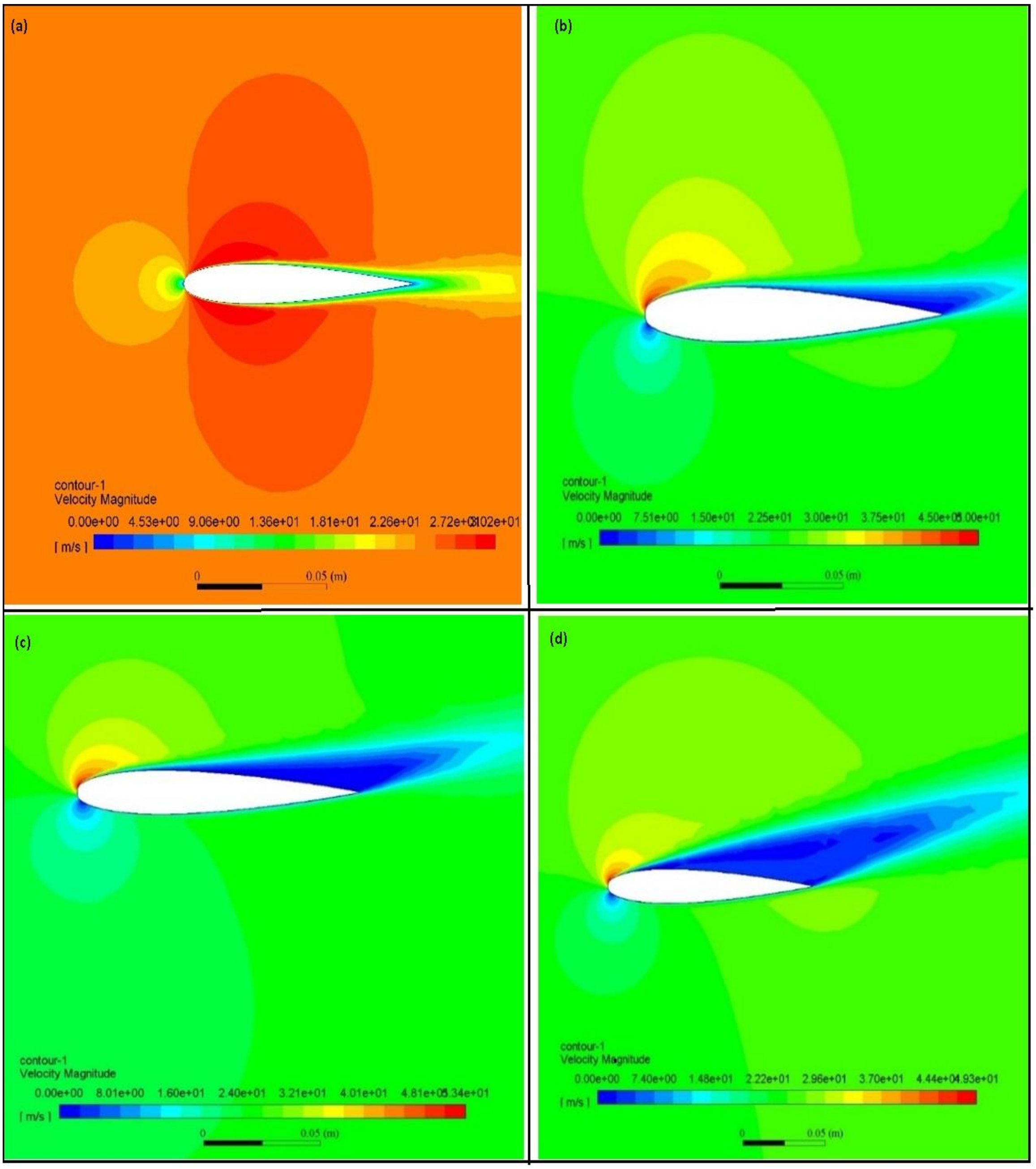

Figure 6 Contours of velocity magnitude at different angle of attacks. (a) AOA=0̊ ,with (b) AOA=10̊ , with (c) AOA=13̊, and (d) AOA=16̊.

5.3 Velocity vectors

The velocity vectors in Figure 7 depict how the airflow changes direction around the airfoil's upper and lower surfaces. As the AOA increases, the velocity accelerates over the upper surface, creating low pressure and contributing to lift, while the flow decelerates along the bottom surface, resulting in higher pressure. This visualization highlights the airflow behavior and its interaction with the airfoil, demonstrating how changes in AOA influence lift and drag characteristics.

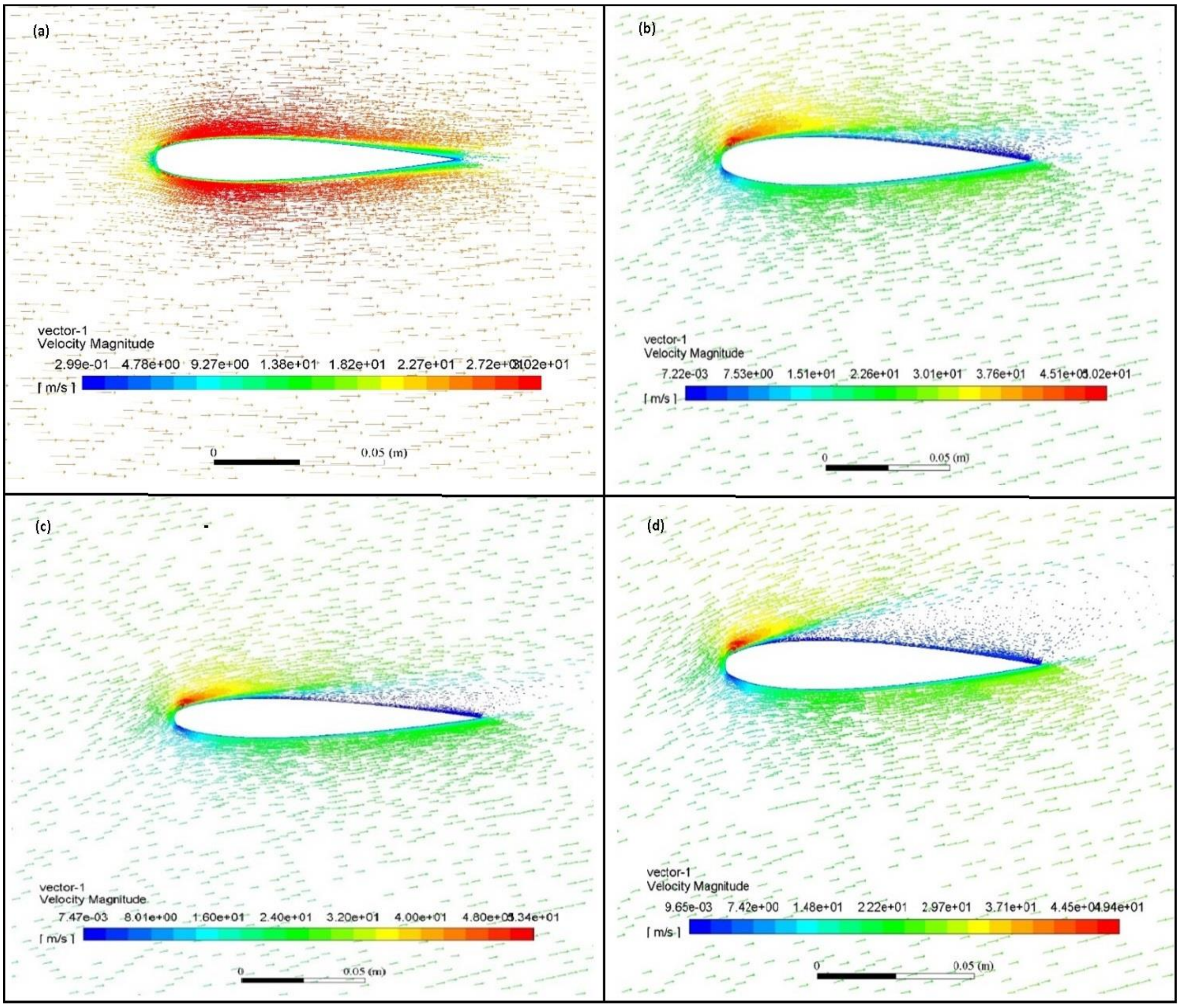

Figure 7 Vectors of velocity magnitude at different angle of attacks. (a) AOA=0° ,with (b) AOA=10° , with (c) AOA=13°, and (d) AOA=16°.

5.4 Pressure counters

Figure 8 shows the static pressure contours around the NACA-0015 airfoil at various angles of attack (AOA) at a velocity of 25 m/s. Lift is generated due to the pressure difference between the upper and lower surfaces of the airfoil. As the AOA and velocity increase, the pressure beneath the airfoil rises, enhancing lift up to the stall point. The highest pressure is observed at the leading edge, with low pressure on the upper surface and high pressure on the lower surface, demonstrating the aerodynamic forces acting on the airfoil.

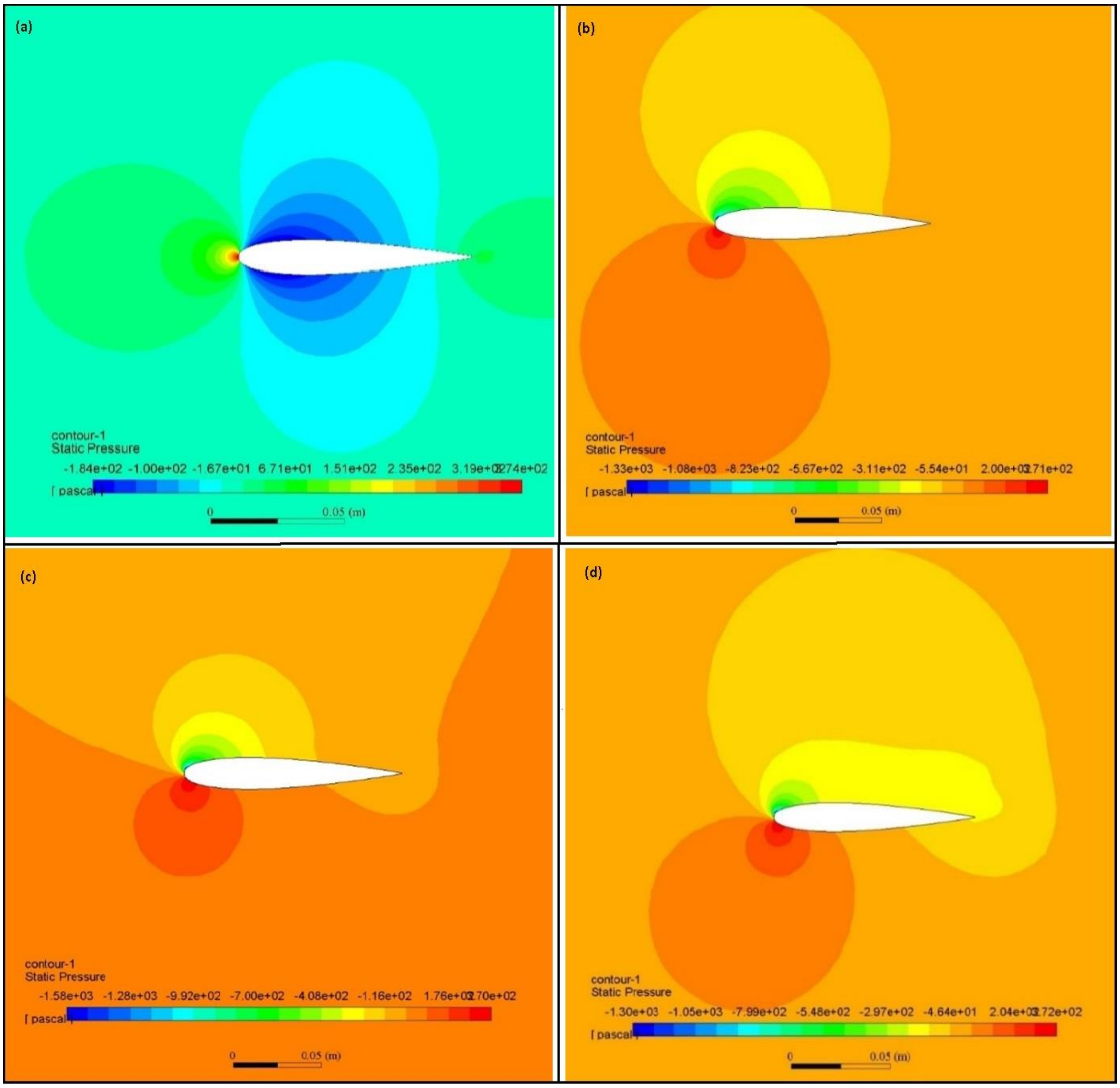

Figure 8 Contour of static pressure at different angle of attacks. (a) AOA=0°,with (b) AOA=10°, with (c) AOA=13°, and (d) AOA=16°.

6. Conclusions

Based on this study, the following conclusions were drawn:

- ✓ ANSYS Fluent is a reliable simulation tool for analyzing fluid mechanics and its applications in mechanical engineering.
- ✓ The numerical results indicate that the realizable k-epsilon model effectively simulates flow characteristics around an airfoil at varying angles of attack.
- ✓ Lift increases almost linearly with the angle of attack until reaching the critical stall point.
- ✓ To prevent stall and maintain aircraft stability, pilots should avoid exceeding the critical angle of attack.

- ✓ At higher speeds, stalls occur at larger angles of attack.
- ✓ In the event of a stall, pilots should increase speed and reduce the angle of attack to restore balance.
- ✓ Maximum pressure occurs at the leading edge (stagnation point), while minimum velocity is found at the trailing edge (separation point).
- ✓ Lift generation depends not only on wing design but also on the aircraft's speed, which plays a crucial role in maximizing lift.

7. References

Akram, Md Tausif, and Man-Hoe Kim. 2021. "CFD Analysis and Shape Optimization of Airfoils Using Class Shape Transformation and Genetic Algorithm—Part I." *Applied Sciences* 11 (9): 3791-. https://doi.org/10.3390/app11093791.

ANSYS. 2018. "Ansys Fluent | Fluid Simulation Software." 2018. https://www.ansys.com/products/fluids/ansys-fluent.

Cengel, Yunus A., and Robert H. Turner. 2004. *Fundamentals of Thermal-Fluid Sciences*. 2nd edition. Boston: McGraw-Hill Science/Engineering/Math.

Costes, M., V. Gleize, J. Szydlowski, L. Sankar, G. Guzel, and M. Rhee. 2005. "Grid Sensitivity Study for the Turbulent Viscous Flow around a NACA0015 Airfoil at Stall." http://hdl.handle.net/20.500.11881/1261.

El Maani, R., B. Radi, and A. El Hami. 2018. "CFD Analysis of the Transonic Flow over a NACA 0012 Airfoil." *Incertitudes et Fiabilité Des Systèmes Multiphysiques* 2 (2). https://doi.org/10.21494/ISTE.OP.2018.0307.

FAA. 2023. "Pilot's Handbook of Aeronautical Knowledge | Federal Aviation Administration." 2023. https://www.faa.gov/regulations_policies/handbooks_manuals/aviation/phak.

John D, Anderson. 2017. *Fundamentals of Aerodynamics*. 6th ed. McGraw-Hill Education. https://www.amazon.com/Fundamentals-Aerodynamics-John-Anderson-Jr/dp/1259129918.

John D. Anderson Jr. 1995. *Computational Fluid Dynamics: The Basics With Applications*. International Ed edition. New York: McGraw-Hill.

Khalid, Mirza Haseeb. 2022. "CFD Analysis of NACA 0012 Aerofoil to Investigate the Effect of Increasing Angle of Attack on Coefficient of Lift and Coefficient of Drag." *Journal of Studies in Science and Engineering* 2 (1): 74–86. https://doi.org/10.53898/josse2022216.

Rasal, Sandesh K., and Rohan R Katwate. 2017. "EXPERIMENTAL INVESTIGATION OF LIFT & DRAG PERFORMANCE OF NACA0012 WIND TURBINE AEROFOIL." *IJATES* 5 (4): 594–99.

Raymer, Daniel P. 1992. *Aircraft Design: A Conceptual Approach*. 2nd Printing edition. Washington, D.C: Amer Inst of Aeronautics &.

Rubel, Robiul Islam, Md K. Uddin, Md Zahidul Islam, and Md Rokunuzzaman. 2016. "Comparison of Aerodynamics Characteristics of NACA 0015 & NACA 4415." Preprints. https://doi.org/10.20944/preprints201610.0095.v1.

Teruna, Christopher, Francesco Avallone, Damiano Casalino, and Daniele Ragni. 2021. "Numerical Investigation of Leading Edge Noise Reduction on a Rod-Airfoil ConfigurationUsing Porous Materials and Serrations." *Journal of Sound and Vibration* 494:115880-. https://doi.org/10.1016/j.jsv.2020.115880.

Uddin, Md, Md Islam, Md Rokunuzzaman, and Robiul Islam Rubel. 2015. "Experimental and Numerical Measurement of Lift and Drag Force of NACA 0015 Aerofoil Blade." In . Bangladesh: RUET, Dept. of Mechanical Engineering. http://103.99.128.10:8080/xmlui/handle/123456789/236.

Uddin, S. M. Nasim, Mohammad Rejaul Haque, M. Merajul Haque, Md. Fazlay Alam, and Abu Hamja. 2024. "Numerical Investigation of the Enhancement of the Aerodynamic Performance for Newly Modified Blended Airfoils Utilizing S809, S829, and NACA 2412 Baseline Shapes." *Arabian Journal for Science and Engineering* 49 (2): 2233–48. https://doi.org/10.1007/s13369-023-08180-2.

8. Biographies

Dr. Salem Fathi Elsheltat is a lecturer in the Department of Mechanical Engineering at Misurata University. He holds a Ph.D. in Tribology from Cardiff University, UK, a MSc in Mechanical Engineering from the University of Bridgeport, USA, and a BSc in Mechanical Engineering from Misurata University, Libya. Dr. Salem's research interests focus on applied mechanics and biomedical engineering.

التحليل العددي لأداء الجنيح باستخدام الانسيس: دراسة تفصيلية لخصائص الرفع والسحب والانهيار في الطيران

استلمت الورقة بتاريخ 2024/08/15،
وقبلت بتاريخ 2024/09/15
ونشرت بتاريخ 2024/09/21

الكلمات المفتاحية: زاوية الهجوم؛ ديناميكيات الموائع الحسابية ؛ الرفع والسحب؛ الجنيح نوع (NACA-0015)

الملخص

تهدف هذه الورقة إلى تحليل الأداء الديناميكي الهوائي لشفرات الجنيح (**NACA-0015**)، مع التركيز على كيفية تأثير تغيير زاوية الهجوم على خصائص الرفع والسحب والانهيار. يعد تصميم جناح الطائرة أمرًا حاسمًا لتحقيق أقصى قدر من الرفع مع تقليل السحب، وهذان العاملان يتم تنظيمهما من خلال ضبط زاوية الهجوم أثناء الطيران. لاستكشاف هذه الديناميكيات، يتم استخدام تحليل ديناميكيات الموائع الحسابية باستخدام برنامج الأنسيس لمحاكاة تدفق الموائع حول الجنيح. يتم تصميم هندسة الجنيح، التي يبلغ طول وترها 0.06 مترًا ومدى 0.25 مترًا، باستخدام أداة النمذجة الحاسوبية و تُجرى محاكيات **(CFD)** باستخدام نموذج الاضطراب **(Realizable k-epsilon)**، حيث يتم تحليل زوايا الهجوم التي تتراوح من 0° إلى 18° تحت أرقام رينولدز منخفضة تتراوح بين 6×10^4 و 1.6×10^5. من خلال هذا النهج الشامل، توفر الدراسة فهمًا أعمق لخصائص التدفق حول الجنيح ، ويساهم في المعرفة الأوسع بأداء الأجنحة في تطبيقات الطيران.