Afro-Asian Journal of Scientific Research (AAJSR)
المجلة الأفروآسيوية للبحث العلمي (AAJSR)
E-ISSN: 2959-6505
Volume 2, Issue 4, October - December 2024
Page No: 23-33
Website: https://aajsr.com/index.php/aajsr/index
SJIFactor 2024: 5.028 ISI 2024: 0.580 معامل التأثير العربي (AIF) 2023: 0.51

Experimental Analysis of NACA-0015 Airfoil Performance: CFD and Wind Tunnel Validation of Lift, Drag, and Stall Behavior

Salem Fathi Elsheltat[1] *, Haithem Elderrat[1], Eman Abdulsalam[1], Ali Alhussain Altaweel[1]
[1] Department of Mechanical Engineering, Faculty of Engineering, Misurata University, Misurarta, Libya

التحليل التجريبي لأداء الجنيح: التحقق ومحاكاة الرفع والسحب والانهيار في الطيران باستخدام ديناميكا الموائع الحسابية ونفق الرياح

سالم فتحي الشلتات [1*]، هيثم اسماعيل الضراط[1]، ايمان عبدالسلام[1]، علي الحسين الطويل[1]
[1] قسم الهندسة الميكانيكية، كلية الهندسة، جامعة مصراتة، مصراتة، ليبيا

*Corresponding author: salem.sheltat@eng.misuratau.edu.ly

Received: July 07, 2024	Accepted: October 05, 2024	Published: October 16, 2024

Abstract:
This work presents an experimental analysis of the aerodynamic performance of the NACA-0015 airfoil, focusing on the effects of varying the angle of attack on lift, drag, and stall behavior. The design and optimization of an airplane's wing are essential for enhancing lift and minimizing drag, both of which are influenced by changes in the angle of attack during flight. Engineering these aerodynamic dynamics, experiments were conducted using a low-speed wind tunnel at the Misurata University Faculty of Engineering's labs. This setup facilitated the measurement of lift and drag forces across various flow velocities and angles of attack, with a particular focus on identifying early stall points that occur at lower flow velocities. The experimental data were then compared with existing Computational Fluid Dynamics (CFD) simulation results to validate the accuracy and reliability of the experimental findings. This thorough approach provides a powerful insight into the flow dynamics around the NACA-0015 airfoil. This comprehensive approach offers a general understanding of the flow behavior over the NACA-0015 airfoil, expanding our knowledge of its aerodynamic performance and driving advancements in airfoil research within the aviation sector.

Keywords: Angle of attack; Computational fluid dynamics; Lift and drag; NACA-0015 airfoil; Stall; Wind tunnel.

الملخص
يقدم هذا العمل تحليلًا تجريبيًا للأداء الديناميكي الهوائي للجنيح (*NACA-0015*)، مع التركيز على تأثيرات تغيير زاوية الهجوم على سلوكيات الرفع والسحب والانهيارفي الطيران. يُعد تصميم وتحسين جناح الطائرة أمرًا ضروريًا لتعزيز الرفع وتقليل السحب، وكلاهما يتأثر بتغيرات زاوية الهجوم أثناء الطيران. ولتحقيق هذه الديناميكيات الهوائية، تم إجراء التجارب باستخدام نفق رياح منخفض السرعة في مختبرات كلية الهندسة بجامعة مصراتة. أتاح هذا الإعداد قياس قوى الرفع والسحب عبر سرعات تدفق وزوايا هجوم مختلفة، مع التركيز بشكل خاص على تحديد نقاط الانهيار المبكرة التي تحدث عند سرعات تدفق منخفضة. تم بعد ذلك مقارنة البيانات التجريبية مع نتائج المحاكاة باستخدام ديناميكيات الموائع الحسابية (*CFD*) للتحقق من دقة وموثوقية النتائج التجريبية. يوفر هذا النهج الشامل فهمًا أعمق لسلوك التدفق حول الجنيح ، مما يوسع

معرفتنا بأدائه الديناميكي الهوائي ويدفع عجلة التقدم في أبحاث الأجنحة ضمن قطاع الطيران.يقدم هذا النهج الشامل فهمًا أعمق لسلوك التدفق حول الجنيح ، مما يوفر رؤى قيمة حول أدائه الديناميكي الهوائي ويسهم في مجال أبحاث الأجنحة في الطيران.

الكلمات المفتاحية: الإنهيار في الطيران؛ الجنيح؛ النفق الهوائي؛ ديناميكيات الموائع الحسابية؛ زاوية الهجوم؛ قوى الرفع والسحب.

1. Introduction

Airfoil design is critical for the aerodynamic performance of aircraft wings, as it significantly affects lift and drag forces, which are vital for stable and efficient flight. The NACA-0015, a symmetric airfoil commonly used in aerodynamic research, is the focus of this study. This research explores the flow behavior around the NACA-0015 airfoil at different angles of attack using Computational Fluid Dynamics (CFD) simulations in ANSYS Fluent and experimental wind tunnel testing. By analyzing how lift and drag coefficients change with varying flow conditions, the study offers important insights into stall behavior and the overall aerodynamic efficiency of the airfoil, aiding in the optimization of wing design in aviation.

1.1 Motivation

The motivation for this study is to precisely determine the lift and drag coefficients of the NACA-0015 airfoil at different angles of attack, as these factors are essential for optimizing wing design to achieve efficient lift and reduced drag during flight. This requires a full understanding of flow characteristics through both computational (CFD) and experimental (wind tunnel) approaches.

1.2 Significance of this research

This research is significant as it combines CFD analysis with wind tunnel testing to validate the aerodynamic performance of the NACA-0015 airfoil. The integration of numerical and experimental methods offers a dependable approach for identifying stall angles and examining the aerodynamic behavior of airfoils. This work provides valuable insights into aerodynamics and supports the improvement of aircraft wing design processes.

1.3 Identification of this work

This work investigates the flow characteristics of the NACA-0015 airfoil using wind tunnel experiments. It involves performing wind tunnel tests at low Reynolds numbers, and different angles of attack. A key finding is the identification of early stall points at low flow velocities, with results validated through a comparison of existing numerical.

1.4 Literature review of related studies

Airfoil performance is a critical component in the design and optimization of aircraft wings, directly impacting lift, drag, and stall characteristics. The study of airfoil performance has evolved with the integration of both computational fluid dynamics simulations and experimental techniques, such as wind tunnel testing. These methods help to evaluate aerodynamic behavior under a wide range of conditions, providing understandings into improving airfoil designs. Several studies [1-18] have utilized both CFD and experimental methods, including wind tunnel testing, to assess these aerodynamic properties under different conditions.

James et al., [5] introduced a method for predicting wind-induced cross ventilation using wind tunnel testing coupled with CFD. This study demonstrated the utility of CFD simulations in conjunction with wind tunnel data for predicting airflow through building openings. By conducting both steady-state and transient CFD simulations, the research showed the potential for predicting average cross ventilation rates and flow fluctuations, providing a robust method for assessing building designs' cross ventilation potential. Similarly, Şahin and Acir [16] conducted a combined numerical and experimental investigation of the NACA 0015 airfoil to analyze lift and drag performances at different angles of attack at low Reynolds numbers. This study utilized wind tunnel tests and CFD simulations using the FLUENT software.

However, several studies have also focused on purely experimental methods to assess airfoil performance. Hoseinzadeh et al., [1] conducted an extensive experimental investigation of the CFJ0025-065-196 airfoil using a wind tunnel equipped with advanced measurement tools. The study examined various parameters, such as the impact of the angle of attack and jet flow velocity on velocity profiles and wake effects. The experiments, conducted at different attack angles and jet flow velocities, provided detailed insights into the airfoil's aerodynamic behavior and contributed to refining performance models for this type of airfoil. Balaji and Lowast [14] studied the performance of a Modified Co-Flow Jet (MCFJ) airfoil equipped with a converging nozzle near the leading edge to reduce pumping power. The experimental results showed that the MCFJ airfoil demonstrated an improved lift coefficient, and a delayed stall angle compared to the baseline airfoil. The findings highlighted the potential of modified designs, such as the MCFJ, in enhancing aerodynamic performance while reducing energy consumption, showcasing the effectiveness of innovative design approaches. Investigations into the aerodynamic performance of airfoils also include studies on noise reduction and control.

Mohammad et al., [10] explored the effects of channeling on the aerodynamic noise and performance of a NACA 0012 airfoil used in wind turbine applications. Both experimental and numerical methods were employed to assess the impact of different channel sizes and directions on noise levels and aerodynamic efficiency. The study found that channeled airfoils exhibited increased drag and reduced lift compared to standard configurations, providing critical data for optimizing airfoil design for specific applications like wind turbines. Nagaraju et al., [8] conducted an experimental study on the effects of aspect ratio adjustments and the integration of triangular dimples on the NACA 4412 airfoil, aimed at enhancing the performance of an H-type Darrieus wind rotor. The results demonstrated that modifications, such as the addition of triangular dimples, improved the self-starting capability and expanded the operational range of the tip speed ratio (TSR). This modification led to a 13.6% increase in maximum power coefficient (Cp, max) compared to a dimple-free rotor. The findings underscore the potential of structural modifications in improving airfoil performance, particularly in low-wind-speed applications.

The reviewed literature reveals a diverse range of approaches to studying airfoil performance, from combining CFD with experimental methods to purely experimental investigations focusing on innovative modifications. Each study contributes uniquely to the understanding of how airfoils behave under different conditions, providing valuable data for refining design parameters. Continued research that integrates both numerical and experimental methods is essential for further advancements in airfoil technology, enhancing aerodynamic performance in various applications, from aircraft to renewable energy systems.

2. Mathematical Model

The Reynolds number (Re) is a dimensionless parameter used to determine whether fluid flow is laminar or turbulent. It compares inertial forces to viscous forces, with higher values indicating turbulent flow. For example, flow over an airfoil starts laminar and transitions to turbulent at a critical Reynolds number of around 1×10^5, becoming fully turbulent near 3×10^6. Lift and drag are key aerodynamic forces. Lift acts perpendicular to airflow and is generated by pressure differences caused by airflow over an airfoil. Drag opposes motion through the air, caused by air resistance, skin friction, and the byproducts of lift. Both forces depend on factors like flow velocity, wing geometry, and the angle of attack (AOA). Weight and thrust are also fundamental to flight. Weight, influenced by the aircraft's size, materials, and payload, always acts downward. Thrust, produced by engines, moves the aircraft forward, allowing the wings to generate lift. For takeoff, lift must overcome weight and thrust must overcome drag.

The lift coefficient (C_L) and drag coefficient (C_D) are dimensionless numbers that model how factors like shape, inclination, and flow conditions affect lift and drag forces. These coefficients express the ratio of force to dynamic pressure times the reference area. The lift-to-drag ratio (L/D) indicates aerodynamic efficiency; a higher L/D ratio means more lift or less drag, improving flight performance. A stall occurs when the wing's angle of attack exceeds a critical point, causing a sudden loss of lift due to airflow separation. Recovery from a stall generally involves reducing the angle of attack, leveling the wings, and increasing power to restore airflow over the wing. Stall behavior varies depending on airfoil shape and thickness, affecting drag, lift, and structural characteristics [19-24].

The primary objective of this study is to experimentally investigate the aerodynamic characteristics of a NACA-0015 airfoil. The experiment will be conducted using a low-speed wind tunnel at the Mechanical Engineering Lab of Misurata University. The focus is to determine the lift and drag forces experienced by the airfoil at different angles of attack and varying airspeeds. Additionally, the study will examine stall behavior as a key area of interest. To validate the findings, the experimental results will be compared with numerical data from Elsheltat and colleagues [24].

- Reynold’s number (Re) is 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 lift coefficient (C_L) is defined as in equation (2):

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

Where A is wing area, and (F_L) is the lift force.

Similarly, the drag coefficient (C_D) is defined as in equation (3):

$$C_D = \frac{F_D}{\rho * 0.5 * V^2 * A} \quad (3)$$

Where (F_D) is the drag force.

3. Design Specifications of the NACA 0015 Airfoil Specimen

The NACA 0015 airfoil, part of the 4-digit series from the National Advisory Committee for Aeronautics (NACA), is used in this study. This airfoil is symmetrical, with the first two digits "00" indicating no camber, and the last two digits "15" representing a thickness-to-chord length ratio of 15%. In other words, the airfoil's thickness is 15% of its chord length. The design specifications for the NACA 0015 airfoil are as follows:

- **Chord length (C):** 0.06 m
- **Span length (L):** 0.25 m
- **Surface area (A):** A=C×L=0.06×0.25=0.0156 m^2

Relevant air properties include:

- **Density (ρ):** 1.186 kg/m^3
- **Dynamic viscosity (μ):** 1.84×10^{-5} kg/(m·s) at 288.15 K

4. Methodology for Experimental Analysis

The Mechanical Engineering Laboratory at Misurata University is equipped with a low-speed wind tunnel for experimental testing, as illustrated in Figure 1. The wind tunnel used in this study features a test section with a cross-sectional area of 304 mm in width, 304 mm in height, and 457 mm in length, constructed from transparent material Figure 2 (f). To ensure a uniform, low-turbulence airflow, the tunnel includes a honeycomb section, as shown in Figure 2 (b). The test specimen is positioned at the center of the test section, Figure 2 (e). The wind tunnel is powered by a 1.5-kilowatt motor that drives the fan, Figure 2 (c). Lift and drag forces are measured using a balanced arm mechanism, Figure 2 (a), and airspeed is determined using an inclined tube manometer Figure 2 (d).

Figure 1: Wind Tunnel at the Faculty of Engineering Laboratory, Misurata University.

(a) Balanced Arm

(b) Honey comb

(c) Motor

(d) Speed Measure

(e) Airfoil

(f) Test Section

Figure 2: Components of the Experimental Setup.

This experimental study investigates how varying the airspeed and angle of attack in the test section affects the lift and drag forces on an airfoil, as summarized in Table 1. First, the airspeed is set to 15 m/s, and the angle of attack varies from 0 to 18 degrees. The corresponding lift force (F_l) and drag force (F_d) are measured using a balanced arm, with the results recorded in Table 1 (a). Then, the airspeed is increased to 20 m/s, and the angle of attack is again varied from 0 to 18 degrees. The lift and drag forces are measured and presented in Table 1 (b). Finally, the airspeed is set to 25 m/s, and the angle of attack varies from 0 to 18 degrees. The measured lift and drag forces are tabulated in Table 1 (c).

Table 1: Measured Lift and Drag Forces from Experimental Data.

Tabel 1 (a) V=15 m/sec			Tabel 1 (b) V=20 m/sec			Tabel 1 (c) V=25m/sec		
Angle of attack in degree	Lift force F_l	Drag force F_d	Angle of attack in degree	Lift force F_l	Drag force F_d	Angle of attack in degree	Lift force F_l	Drag force F_d
0	0	0.34	0	0	0.43	0	0	0.53
8	0.4	0.37	8	0.96	0.47	8	1.87	0.63
12	0.47	0.41	12	1.45	0.5	12	2.45	0.74
13	0.81	0.47	13	1.83	0.63	13	2.91	0.87
14	0.84	0.5	14	1.9	0.7	14	3.41	0.97
16	0.67	0.71	16	1.29	0.75	16	3.44	1.6
18	0.64	0.87	18	1.22	0.96	18	2.2	2.1

The measured data, along with the air properties, were then applied to equations (1), (2), and (3) to obtain the desired results. These results are analyzed and discussed in the following section.

5. Experimental Results and Discussions

The results derived from applying the measured data to the mathematical model are presented in Table 2.

Table 2: Experimental Results.

Tabel 2 (a) V=15 m/sec			Tabel 2 (b) V=20 m/sec			Tabel 2 (c) V=25m/sec		
Angle of attack in degree	lift coefficient C_l	Drag coefficient C_d	Angle of attack in degree	lift coefficient C_l	Drag coefficient C_d	Angle of attack in degree	lift coefficient C_l	Drag coefficient C_d
0	0.0000	0.1618	0	0.0000	0.1151	0	0.0000	0.0908
8	0.1903	0.1761	8	0.2570	0.1258	8	0.3204	0.1079
12	0.2237	0.1951	12	0.3881	0.1338	12	0.4197	0.1268
13	0.3854	0.2237	13	0.4898	0.1686	13	0.4985	0.1490
14	0.3997	0.2379	14	0.5086	0.1874	14	0.5842	0.1662
16	0.3188	0.3379	16	0.3453	0.2008	16	0.5893	0.2741
18	0.3046	0.4140	18	0.3266	0.2570	18	0.3769	0.3598
Re = 60911.45			Re = 81215.23			Re = 101519.15		

The results from Table 2 were exported to Microsoft Excel to visualize the relationships between key variables. Figure 3 illustrates the relationship between the angle of attack and the lift coefficient at airspeeds of 15, 20, and 25 m/s. The experimental data show that the lift coefficient increases linearly with the angle of attack, but only up to the stall point. The plot indicates that the stall occurs at a lower angle of attack for lower velocities and at a higher angle for higher velocities. Specifically, the stall point is observed at an angle of attack of 14 degrees for a velocity of 20 m/s and at 16 degrees for a velocity of 25 m/s.

Figure 3: Experimental Comparison of Angle of Attack (AOA) and Lift Coefficient (C_L).

Figure 4 displays the relationship between the angle of attack and the drag coefficient at the same velocities of 15, 20, and 25 m/s. The experimental results show that the drag coefficient gradually increases with the angle of attack up to the stall point. Beyond the stall point, the drag coefficient rises sharply with further increases in the angle of attack. To mitigate this sudden rise in drag, the angle of attack should be reduced immediately, and the velocity should be decreased. From Figure 4, it is evident that the drag coefficient increases with the angle of attack at a constant velocity, reaching a maximum value of 0.414 at an angle of attack (AOA) of 18 degrees. Additionally, the data demonstrate that at the same angle of attack, increasing the velocity results in a lower drag coefficient. This relationship between the angle of attack (AOA), drag coefficient (C_D), and velocity can be summarized as follows:

- The relationship between AOA and C_L at a constant velocity is directly proportional.
- The relationship between velocity and C_L at a constant AOA is inversely proportional.
- In summary, Figure 4 shows that the drag coefficient can reach a maximum value of 0.414 at an angle of 18 degrees.

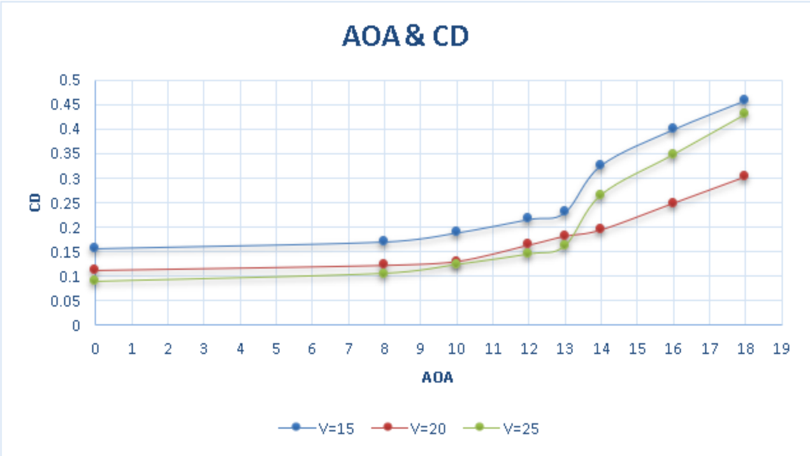


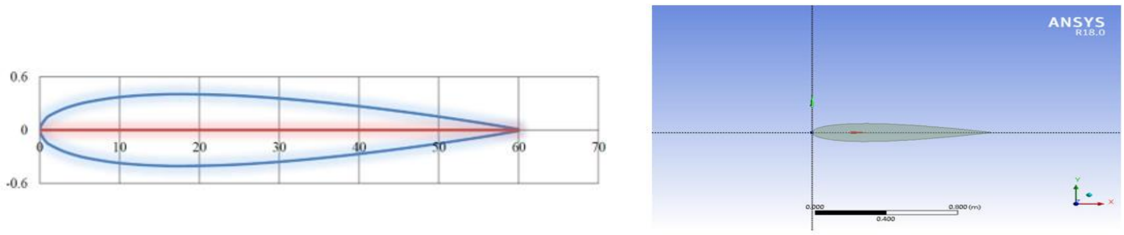
Figure 4: Experimental Comparison of Angle of Attack (AOA) and Drag Coefficient (C_D).

6. Comparison Between CFD Simulations and Wind Tunnel Experiments

To confirm the findings, the experimental results will be compared with the numerical data provided by Elsheltat and colleagues [24].

6.1 CFD Model

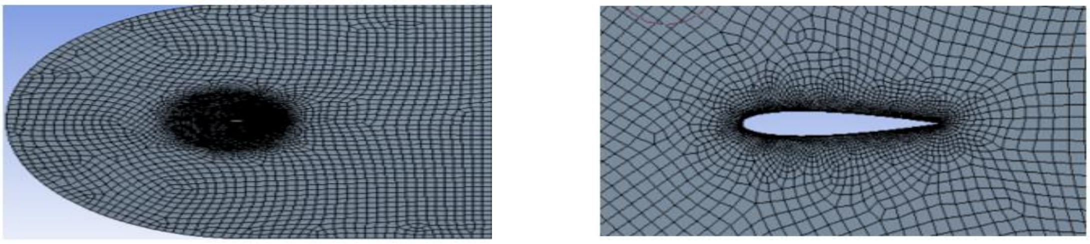
Computational Fluid Dynamics (CFD) applies numerical techniques to approximate fluid motion equations by converting complex partial differential equations into simpler, solvable algebraic forms. Since the 1950s, advancements in computing technology have made CFD a vital tool for simulating scenarios that are difficult to reproduce through experiments or solve analytically. The CFD process typically involves three steps: a pre-processor that sets up the geometry, grid, and flow conditions; a flow solver that computes the governing equations; and a post-processor that visualizes the results [19]. ANSYS Fluent, a popular CFD software, offers robust modeling capabilities for simulating flow, turbulence, heat transfer, and chemical reactions in diverse applications such as aerospace, combustion, and bioengineering. In Elsheltat and team's study [24] , ANSYS Fluent was utilized to numerically analyze the fluid flow around a NACA-0015 airfoil. The coordinates of the airfoil, shown in Figure 5 (a), were obtained from the NACA website and imported into ANSYS Design Modeler (Figure 5 (b)).



(a) Chord length and thickness (mm) (b) Airfoil in Design Modeler

Figure 5: Geometry of the airfoil.

A C-mesh technique was used to flow analysis around the airfoil, featuring an arc with a 75 mm radius and a 75 x 75 mm rectangular grid. The mesh was refined with high smoothing and fine relevance settings to enhance accuracy, especially around complex regions like the trailing edge. The final mesh, illustrated in Figure 6, consisted of 14,099 nodes and 13,792 elements, ensuring a detailed representation of the airfoil for accurate CFD analysis. This meshed geometry was then imported into FLUENT to solve the coupled momentum and pressure-based continuity equations in a two-dimensional plane using Viscous-Realizable k-epsilon modeling with Reynolds-averaged Navier–Stokes (RANS) equations. The airfoil was modeled with aluminum properties, while 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. The simulation utilized the SIMPLE scheme for pressure-velocity coupling and applied second-order upwind discretization for spatial accuracy. The flow equations were solved based on convergence criteria or a specified number of iterations. The drag force (F_D) and lift force (F_L) were computed as the horizontal and vertical components of the resultant force (F_R) acting on the airfoil, using the angle of attack (AOA). Their simulation examined a range of angles of attack from 0° to 18° at different velocities [24].



(a) C-mesh around the airfoil (b) Detailed view of the mesh.

Figure 6: Mesh of the airfoil.

6.2 Experimental Vs Numerical Results

To validate the current work, Table 3 presents a comparison between the lift coefficient values obtained from an existing numerical study [24] and those from the current experimental study. Based on the data presented in Table 3, Figure 7 and Figure 8, the results show a general correlation, but there are discrepancies due to several critical factors:

- **Manual Adjustments and Measurements:** The angle of attack adjustments and the arm balance readings were performed manually, introducing a significant source of error into the collected data. To eliminate these inaccuracies, the wind tunnel should be equipped with digital measurement devices, ensuring precise and consistent data acquisition.
- **Air Leakage in the Wind Tunnel:** The wind tunnel used in this experiment suffers from air leakage, leading to pressure loss and compromising the accuracy of the results. This flaw highlights a fundamental issue in the experimental setup that must be addressed to obtain reliable data.
- **Measurement Method for Lift and Drag:** The lift and drag forces were determined using an arm balance, a method prone to errors due to mechanical limitations and human factors. To enhance the accuracy of these measurements, the wind tunnel should be integrated with digital actuation sensors that can provide real-time, precise data, significantly reducing human error.
- **Higher Accuracy of ANSYS Software:** The ANSYS software used for numerical simulations offers a high level of accuracy, primarily because it minimizes human error through automated, algorithm-driven calculations. This highlights the importance of using advanced digital tools in experimental setups to reduce errors and improve the reliability of the data.

Table 3: Experimental (Exp) and numerical (Num) results for lift coefficient (C_L) at different angle of attacks (AOA). (a) at air velocity (v=15m/s), and (b) at air velocity (v=15m/s).

	Tabel 3 (a) V=15 m/sec		Tabel 3 (b) V=20 m/sec	
AOA	**Exp.**	**Num.**	**Exp.**	**Num.**
0	0.0000	0.0013	0.0000	0.0033
4	0.1065	0.4832	0.1295	0.8748
8	0.1903	0.8970	0.2570	1.6304
10	0.2237	1.0490	0.3881	1.9162
12	0.3854	1.1084	0.4898	2.0541
13	0.3997	1.0814	0.5086	2.0047
14	0.3188	1.0302	0.3453	1.8990
15	0.3117	0.9317	0.3299	1.7532
16	0.3046	0.8333	0.3266	1.6074

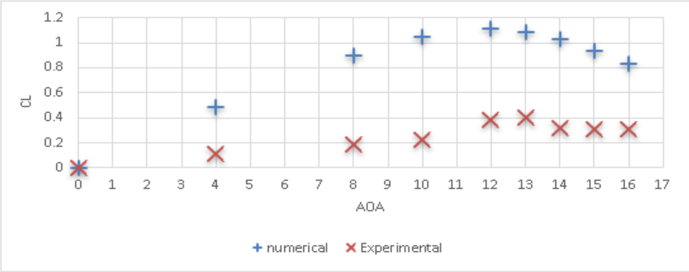


Figure 7: Comparison between Experimental and numerical lift coefficient (CL) at V = 15 m/sec.

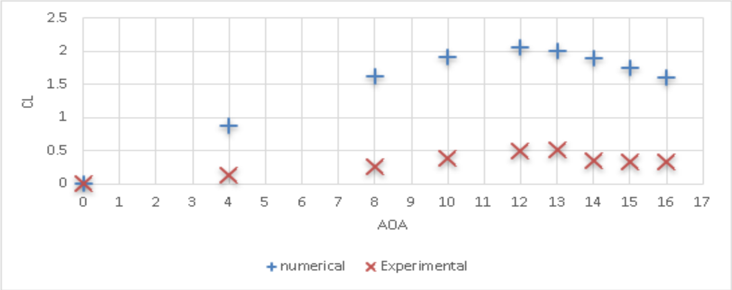


Figure 8: Comparison between Experimental and numerical lift coefficient (C_L) at V = 20 m/sec.

To strengthen the validity and accuracy of future studies, it is essential to address these shortcomings by incorporating digital measurement devices, improving the integrity of the wind tunnel, and adopting more precise data collection methods. Doing so will enhance the consistency and comparability of experimental and numerical results, providing a stronger foundation for validating CFD models.

7. Challenges

Here are the refined challenges faced during the experimental work:

- The wind tunnel used in this study had a maximum velocity limit of 25 m/s, which restricted the range of aerodynamic conditions that could be tested and may have affected the accuracy of the experimental data.
- At the maximum wind tunnel velocity of 25 m/s, the airfoil experienced significant vibrations, making it difficult to obtain accurate measurements from the arm balances and compromising the reliability of the data.
- The NACA-0015 airfoil used in the experiment originally had a flap and a slot, which were removed to match the surface area modeled in the ANSYS simulations. Although this modification did not alter the overall shape of the airfoil, it resulted in slight differences at the trailing edge, which may have impacted the results.
- An air leak was present in the wind tunnel at the test section under the airfoil, causing pressure loss and potentially affecting the accuracy and consistency of the experimental results.
- The weight scales on the arm balances used for measuring forces were not precise, introducing additional uncertainty into the data collection process.

8. Conclusions

Based on the findings of this study, several key conclusions can be drawn:

- The lift generated by an airfoil increases almost linearly with the angle of attack up to a critical point known as stall. Beyond this point, lift rapidly decreases, highlighting the importance of monitoring the angle of attack in flight.
- Pilots must ensure the critical angle of attack is not exceeded to prevent stall and maintain aircraft balance. Understanding this limitation is essential for safe flight operations.
- At high speeds, stall can occur at larger angles of attack, emphasizing the need for careful control of both speed and angle of attack to prevent loss of lift.
- In the event of a stall, the pilot should immediately increase speed and decrease the angle of attack to restore lift and regain control of the aircraft.
- Lift is not solely generated by the wings; the speed of the aircraft is also a critical factor in maximizing lift. This relationship underlines the importance of managing both aerodynamic design and flight speed.

- The comparison between numerical simulations and experimental data shows a strong correlation, with discrepancies of less than 10%, validating the accuracy and reliability of both approaches in studying aerodynamic behavior.
- ANSYS Fluent is a dependable simulation tool for mechanical engineers to analyze fluid mechanics and its applications, providing accurate insights into complex flow phenomena.

Overall, this study reinforces the importance of advanced simulation tools, such as ANSYS Fluent, for accurately modeling aerodynamic phenomena and highlights key principles that are essential for optimizing aircraft performance and safety.

REFERENCE

[1] S. Hoseinzadeh, A. Bahrami, S. M. Mirhosseini, A. Sohani, and S. Heyns, “A detailed experimental airfoil performance investigation using an equipped wind tunnel,” *Flow Meas. Instrum.*, vol. 72, pp. 101717-, 2020, doi: 10.1016/j.flowmeasinst.2020.101717.

[2] M. T. Akram and M.-H. Kim, “CFD Analysis and Shape Optimization of Airfoils Using Class Shape Transformation and Genetic Algorithm—Part I,” *Appl. Sci.*, vol. 11, no. 9, pp. 3791-, 2021, doi: 10.3390/app11093791.

[3] M. H. Khalid, “CFD Analysis of NACA 0012 Aerofoil to Investigate the Effect of Increasing Angle of Attack on Coefficient of Lift and Coefficient of Drag,” *J. Stud. Sci. Eng.*, vol. 2, no. 1, pp. 74–86, 2022, doi: 10.53898/josse2022216.

[4] R. El Maani, B. Radi, and A. El Hami, “CFD Analysis of the Transonic Flow over a NACA 0012 Airfoil,” *Incert. Fiabilité Systèmes Multiphysiques*, vol. 2, no. 2, 2018, doi: 10.21494/ISTE.OP.2018.0307.

[5] L. James Lo, David Banks, and A. Novoselac, “Combined wind tunnel and CFD analysis for indoor airflow prediction of wind-driven cross ventilation,” *Build. Environ.*, vol. 60, pp. 12–23, 2013, doi: 10.1016/j.buildenv.2012.10.022.

[6] A. K. Saha, R. Srivastava, K. Muralidhar, P. K. Panigrahi, and D. Das, “Design and Simulation of a Low Speed Wind Tunnel with Analysis of Wind Effects on an Airfoil (CFD),” in *Fluid Mechanics and Fluid Power - Contemporary Research*, India: Springer India Private Limited, 2016.

[7] G. C. Arjune and S. V. Bhivsane, “Design and Simulation of Wind Tunnel Using CFD Analysis,” in *Emerging Trends in Mechanical Engineering*, in Lecture Notes in Mechanical Engineering. , Singapore: Springer Singapore, 2020, pp. 31–45. doi: 10.1007/978-981-15-8304-9_3.

[8] A. S. Nagaraju, R. Gupta, and S. Bhowmik, “Effects of the influence of triangular dimple and aspect ratio on NACA 4412 airfoil on the overall performance of H-Darrieus wind rotor: An experimental investigation,” *Wind Eng.*, 2024, doi: 10.1177/0309524X241256955.

[9] O. Eisele, G. Pechlivanoglou, C. N. Nayeri, and C. O. Paschereit, “Experimental & numerical investigation of inflow turbulence on the performance of wind turbine airfoils,” in *Proceedings of the ASME Turbo Expo,* 2013. doi: 10.1115/GT2013-94689.

[10] H. K. Mohammad, S. M. Jali, A. Y. Q. A. Qwam Alden, V. Kilchyk, and B. Shrestha, “Experimental and Numerical Investigation of Channeling Effects on Noise and Aerodynamic Performance of NACA 0012 Aerofoil in Wind Turbine Applications,” *Renew. Energy Power Qual. J.*, vol. 21, no. 1, pp. 373–380, 2023, doi: 10.24084/repqj21.328.

[11] M. Uddin, M. Islam, M. Rokunuzzaman, and R. I. Rubel, “Experimental and Numerical Measurement of Lift and Drag Force of NACA 0015 Aerofoil Blade,” presented at the International Conference on Mechanical, Industrial and Materials Engineering, Bangladesh: RUET, Dept. of Mechanical Engineering, Dec. 2015. [Online]. Available: http://103.99.128.10:8080/xmlui/handle/123456789/236

[12] S. K. Rasal and R. R. Katwate, “EXPERIMENTAL INVESTIGATION OF LIFT & DRAG PERFORMANCE OF NACA0012 WIND TURBINE AEROFOIL,” *IJATES*, vol. 5, no. 4, pp. 594–599, 2017.

[13] P. M. Kumar, M. M. R. Surya, R. Kethala, and N. Srikanth, “Experimental investigation of the performance of darrieus wind turbine with trapped vortex airfoil,” IEEE, 2017, pp. 130–135. doi: 10.1109/PGSRET.2017.8251815.

[14] K. Balaji and G. J. J. W. Lowast, “Experimental investigations on the performance of a modified co-flow jet aerofoil,” *Indian J. Sci. Technol.*, vol. 13, no. 19, pp. 1873–1881, Jun. 2020, doi: 10.17485/IJST/v13i19.85.

[15] R. I. Rubel, M. K. Uddin, M. Z. Islam, and M. Rokunuzzaman, “Numerical and Experimental Investigation of Aerodynamics Characteristics of NACA 0015 Aerofoil,” *Int. J. Eng. Technol. IJET*, vol. 2, no. 4, Art. no. 4, Apr. 2017, doi: 10.19072/ijet.280499.

[16] İ. Şahin and A. Acir, “Numerical and Experimental Investigations of Lift and Drag Performances of NACA 0015 Wind Turbine Airfoil,” *Int. J. Mater. Mech. Manuf.*, vol. 3, no. 1, pp. 22–25, 2015, doi: 10.7763/IJMMM.2015.V3.159.

[17] C. Teruna, F. Avallone, D. Casalino, and D. Ragni, “Numerical investigation of leading edge noise reduction on a rod-airfoil configuration using porous materials and serrations,” *J. Sound Vib.*, vol. 494, pp. 115880-, 2021, doi: 10.1016/j.jsv.2020.115880.

[18] S. M. N. Uddin, M. R. Haque, M. M. Haque, Md. F. Alam, and A. Hamja, “Numerical Investigation of the Enhancement of the Aerodynamic Performance for Newly Modified Blended Airfoils Utilizing S809, S829, and NACA 2412 Baseline Shapes,” *Arab. J. Sci. Eng.*, vol. 49, no. 2, pp. 2233–2248, Feb. 2024, doi: 10.1007/s13369-023-08180-2.

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

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

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

[22] Federal Aviation Administration(FAA), “Pilot’s Handbook of Aeronautical Knowledge | Federal Aviation Administration.” [Online]. Available: https://www.faa.gov/regulations_policies/handbooks_manuals/aviation/phak

[23] S. F. Elsheltat, A. A. Alshara, and W. Elshara, “Aerodynamic Evaluation of a Symmetrical Airfoil Using Wind Tunnel Testing,” *Afr. J. Adv. Pure Appl. Sci. AJAPAS*, pp. 49–56, 2024, [Online]. Available: https://aaasjournals.com/index.php/ajapas/article/view/946

[24] S. F. Elsheltat, A. A. Alshara, A. A. Altaweel, and E. Abdulsalam, “Numerical Investigation of NACA-0015 Airfoil Performance Using ANSYS: A Detailed Study of Lift, Drag, and Stall Characteristics,” *J. Acad. Res.*, vol. 28, no. 2, 2024.