International Journal of Inventive Engineering and Sciences (IJIES)
ISSN: 2319-9598 (Online), Volume-12 Issue-5, May 2025

Numerical Analysis of Lift Coefficients for NACA 4412 Airfoil Across Different Angles of Attack

Neloy Kumar Das, Arup Das, Syeda Tanjila Sarwar

Abstract: The accurate prediction of aerodynamic performance is critical for the design and optimization of airfoils used in aerospace, automotive, and renewable energy applications. This study focuses on evaluating and comparing the lift coefficients of the NACA 4412 airfoil using three distinct methodologies: CFD, wind tunnel experimentation, and the Panel method. The primary objective is to assess the accuracy and limitations of each technique in capturing the aerodynamic characteristics of the airfoil. CFD simulations were conducted using ANSYS FLUENT, applying a steady-state, incompressible flow model with appropriate turbulence modeling to capture flow behavior across a range of angles of attack. Experimental validation was performed in a controlled wind tunnel environment to generate benchmark data. Additionally, the Panel method analysis was executed using XFOIL, a commonly used inviscid flow solver known for its computational efficiency. The results demonstrate a strong agreement between CFD simulations and experimental data, particularly in predicting lift coefficients at moderate angles of attack. In contrast, XFOIL consistently overestimated lift values, especially at higher angles, due to its inability to accurately model flow separation and viscous effects. This discrepancy highlights the inherent limitations of potential flow methods when applied to complex flow regimes. By systematically comparing these approaches, the study emphasizes the critical need for high-fidelity numerical or experimental validation when assessing airfoil performance. The findings advocate for a cautious application of simplified methods like the Panel method in preliminary design stages and reinforce the role of CFD as a reliable tool in aerodynamic analysis. This work contributes to the ongoing refinement of predictive tools for airfoil design, ensuring more accurate performance assessments in real-world applications.

Keywords: Airfoil optimization, Aerodynamic performance, XFOIL, CFD, NACA 4412.

Abbreviations:
CFD: Computational Fluid Dynamics
NACA: National Advisory Committee for Aeronautics
AoA: Angle of Attack
SST: Shear Stress Transport
VII: Viscous/Inviscid Interaction
CL: Coefficients Such as Lift

Manuscript received on 12 April 2025 | First Revised Manuscript received on 19 April 2025 | Second Revised Manuscript received on 01 May 2025 | Manuscript Accepted on 15 May 2025 | Manuscript published on 30 May 2025.

***Correspondence Author(s)
Neloy Kumar Das, Department of Mechanical Engineering, Bangladesh University of Engineering and Technology (BUET), Dhaka, Bangladesh. Email: joy244430@gmail.com, ORCID ID: 0009-0000-6215-9330.
Arup Das*, Lecturer, Department of Mechanical Engineering, Military Institute of Science and Technology (MIST), Dhaka, Bangladesh. Email: dasarup184@gmail.com, ORCID ID: 0009-0005-9140-5947.
Syeda Tanjila Sarwar, Department of Mechanical Engineering, Bangladesh University of Engineering and Technology (BUET), Dhaka, Bangladesh. Email: syedaatanjila@gmail.com, ORCID ID: 0009-0007-0064-0349.

© The Authors. Published by Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP). This is an open access article under the CC-BY-NC-ND license http://creativecommons.org/licenses/by-nc-nd/4.0/

I. INTRODUCTION

Airfoil design is a necessary component of the process of designing proper aerodynamic surfaces, such as wings and rotor blades, that allow them to generate lift to fly. The further need for generating more efficient aerodynamic components, due to the continuous expansion of the aviation sector has made airfoil optimization become one of these points of interest in large body research about enhancing the aerodynamic effectiveness of aircraft and UAVs [1]. This method is advantageous for applications in renewable energy systems, especially in the assessment of wind turbines [2]. The lift-to-drag ratio is critical for flight efficiency and improvements to airfoil performance can enhance this significantly. Lift production is a function of the pressure difference across the airfoil surfaces, and these pressure differences are dependent upon the geometry of the airfoil (e.g., its curvature, camber, thickness) [3]. Significant improvement in the performance of an airfoil especially at low speeds, where subsonic flight is done, can be achieved by changing these parameters. Among these, National Advisory Committee for Aeronautics (NACA) developed the NACA 4412, which wing cross-section is well known for its performance at providing lift over a range of angle of attack (AoA) values, which make it an ideal candidate for low-speed, subsonic aircraft [4]. Computational and experimental methods are commonly used for the analysis of lift, drag and stall behavior of an airfoil. Common airfoil analysis tools such as ANSYS Fluent, XFOIL and XFLR5 are powerful software able to simulate airflow and pressure distribution over them [5]. Computational Fluid Dynamics (CFD), offers a more detailed and accurate representation of the complex flow phenomena around the airfoil, including the prediction of turbulent flow, pressure distribution, and wake formation [6]. Hence, the most important details of flow characteristic through CFD simulations will be investigated as boundary layer behavior, vortex shedding and separation phenomenon by tools like ANSYS Fluent using a model representing airfoil in different flight conditions [7]. CFD simulations have shown that the NACA 4412 airfoil can generate high lift numbers at low angles of attack, and that critical separation points with associated flow stall – particularly at high AoA - would be sited unless carefully synthesized [8]. During the last decades, its ability to predict experimental

© Copyright: All rights reserved.

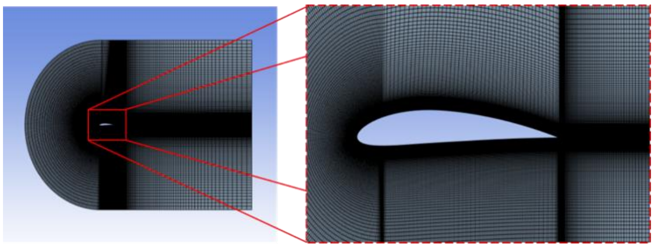
data has also been validated by many different applications so that CFD offers a good compromise between high computational demands and required precision (high-fidelity) in aerodynamics [9]. For instance, XFLR5 is very efficient to simulate two-dimensional flows around airfoils like the NACA 4412 using Panel methods [10]. The NACA 4412 airfoil has been studied in the past and shows a good performance for lift generation at low angles of attack (AoAs) with relatively high lift-to-drag ratio at subsonic speeds. Nevertheless, when the AoA extends further, flow separation occurs in the airfoil which causes lift to abruptly fall [11]. This phenomenon, commonly named as “stall” takes place near 17° AoA for NACA 4412 where the flow starts to separate from the airfoil surface [12]. It was demonstrated through numerical simulations using Ansys Fluent that the airfoil when optimized works best at around 8° AoA after which characterized drag forces effects start to be pronounced [13]. To improve airfoil performance under different conditions, a number of optimization strategies have been tackled. These devices have included the addition of vortex generators that delay flow separation, as well as surface modifications such slotted or grooved flaps to increase lift. Inspired by nature, biomimetic surface designs such as shark skin denticles have shown promise in reducing drag by manipulating turbulent boundary layer behavior [14]. There is also the effect of "ground effect" that changes lift-to-drag ratio for an airfoil that works near to the ground [15].

Therefore, in this paper; numerical analysis of the NACA 4412 airfoil was carried out with varying angles of attack by using XFLR5. In this paper, we investigate the lift coefficient characteristic to design an airfoil properly for a variety of engineering systems with special focus on low-speed flight.

II. METHODOLOGY

Aircraft wing airfoil NACA 4412 was examined for aerodynamic characteristics by Ansys and XFLR, at chord length of 1000 mm and at Reynolds number of 1.9×10^5. The purpose of this study is to simulate the aerodynamic performance of an airfoil using CFD and XFLR5, and evaluate the deviations between these simulations and experimental data. In order to ensure an accurate shape of the airfoil section, the NACA 4412 airfoil geometry was developed by using coordinates extracted from the NACA Airfoil Plotter [16]. ANSYS Design Modeler was also employed to build a mesh with “C-type” flow domain around the airfoil to ease the mesh generation process. This domain design enabled enhancement of mesh resolution in areas of interest including but not limited to development and separation of boundary layers and other flow features. The domain was enlarged sufficiently in the upstream and downstream as well as normal directions to have enclosure effects on the results kept to a minimum.

A computational fluid dynamics (CFD) compliant structured 2D grid mesh was used to discretize the airfoil and the flow field surrounding it mainly to allow neat cell distribution around the airfoil and particularly in the bottom layer. This type of grid structure was created by ANSYS Meshing, and sections of fine-spacing mesh were applied in particular regions that need more details. The spacing of the grid close to the airfoil was optimized through a biasing factor from eighty to three hundred divisions in order to accurately capture the boundary layer. As for the mesh design, about 250,000 linear elements were used which is quite a fine mesh ideal for calculating flow characteristics. The mesh quality was confirmed by examining orthogonal quality, aspect ratio, and skewness of the mesh. The meshed airfoil with computational domain is visible in Fig. 1.



[Fig. 1: Meshed Domain with Enlarged View.]

The fluid flow around the airfoil was simulated using the fundamental governing equations of fluid dynamics – the Continuity and Navier-Stokes equations. For incompressible, steady, and two-dimensional flow, the Continuity equation is expressed as:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \quad \dots \quad (1)$$

where u and v represent the velocity components in the x and y directions, respectively.

The Navier-Stokes equations, representing the conservation of momentum, are given by:

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \quad \dots \quad (2)$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \quad \dots \quad (3)$$

where ρ is the fluid density, μ is the dynamic viscosity, and p represents the pressure field. These equations were solved using Ansys to simulate the airflow over the airfoil. Here Equation (2) for x-direction and Equation (3) for y-direction. For turbulence modeling, the Shear Stress Transport (SST) k-ω turbulence model was employed due to its accuracy in predicting flow separation and boundary layer behavior at moderate Reynolds numbers. The SST k-ω model effectively combines the advantages of the k-ω model near the wall and the k-ϵ model in the free-stream, providing accurate turbulence representation. The turbulence model equations are expressed as:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho u_i k)=P_k-\beta^{*}\rho k\omega+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_k\frac{\mu_t}{\rho}\right)\frac{\partial k}{\partial x_j}\right] \quad \dots \quad (4)$$

$$\frac{\partial}{\partial t}(\rho\omega)+\frac{\partial}{\partial x_i}(\rho u_i\omega)=\alpha\frac{\omega}{k}P_k-\beta\rho\omega^2+\frac{\partial}{\partial x_j}\left[\left(\mu+\sigma_\omega\frac{\mu_t}{\rho}\right)\frac{\partial\omega}{\partial x_j}\right]+2(1-F_1)\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j} \quad \dots \quad (5)$$

As with before, we can summarize the analysis here in the following way: The ANSYS Fluent simulation was performed with the

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)
© Copyright: All rights reserved.

assumption of steadiness, incompressibility, and two-dimensionality sustaining correct geometries throughout the simulation. An inlet with velocity and a pressure outlet were specified, with the airfoil surface considered as a no-slip wall. The pressure-velocity coupling was carried out adopting the SIMPLE algorithm and second-order upwind schemes were used to discretize the momentum and turbulence equations for better accuracy. The given convergence criteria were based on the assumption that the residuals would not be greater than 10-6. In addition to CFD studies, analysis was carried out using XFLR5, an orthogonal flows analysis code among the first of potential flow theory and panel methods that is very often applied. XFLR5 on the other hand applies the Vortex Panel Method and Viscous/Inviscid Interaction (VII) method to determine aerodynamic coefficients such as lift (CL), drag (CD), and moment coefficients. The NACA 4412 also was reconstructed in XFLR5 with the same Reynolds number of 1.9×105 and the results have been gathered by performing the simulations at other angles of attack and discrimination with the CFD data. XFLR5 utilizes a panel approach to modeling the airfoil surfaces. The structure of the vortex panel is based upon estimation of flow level due to multiple potential flow about an airfoil wing. The assumption of potential flow in XFLR puts it in the sphere of providing rather inviscid solutions as such it cannot accurately account for flow separation and viscous effects as computational fluid dynamics (CFD) does. Nevertheless, estimating the aerodynamic forces is a strong element of the tool for some applications such as comparison and validation. Employing the comparison of the XFLR5 simulation results with those provided by ANSYS Fluent allowed to estimate how accurate and trustworthy the employed CFD model is in regards to its predictive capabilities of flow behavior, especially in the presence of viscous effects and flow separation.

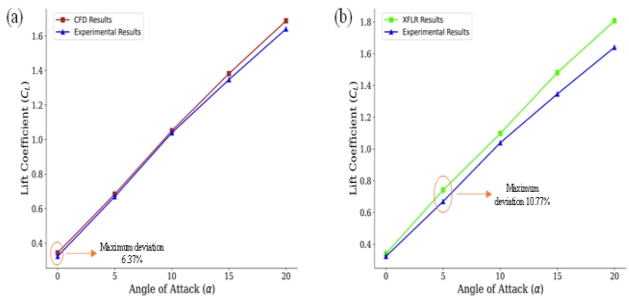
III. RESULT AND DISCUSSION

Lift coefficients for the NACA 4412 airfoil were measured across five simulations, each at a different angle of attack (AoA) [17]. For comparison, we used XFOIL5, an open-source software built in FORTRAN, based on the panel method, to calculate subsonic aerodynamic properties [18]. Experiments were also carried out to compare the lift coefficients at these five angles [19]. Verify the above results from CFD simulations using ANSYS FLUENT, predictions generated by XFLR5, and experimental studies for NACA 4412 airfoil [20].

Table- I: Lift Coefficients from CFD, XFLR5, and Experiments with Their Comparison

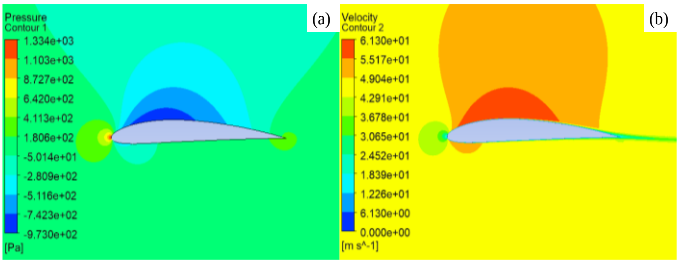
AoA (α)	C_L (CFD)	C_L (XFLR5)	C_L (Exp.)	CFD vs Exp. (%)	XFLR5 vs Exp. (%)
0°	0.3439	0.3399	0.3233	6.37	5.14
5°	0.6834	0.7409	0.6689	2.16	10.77
10°	1.0504	1.0968	1.0386	1.13	5.6
15°	1.381	1.4802	1.3462	2.58	9.96
20°	1.687	1.8069	1.6401	2.85	10.17

The lift coefficients predicted by XFLR5 are generally higher than those from experiments and CFD FLUENT, likely due to the panel method. Fig. 2 (a) shows CFD vs. experimental deviation, and Fig. 2 (b) shows XFLR5 vs. experimental deviation in lift coefficient (CL).

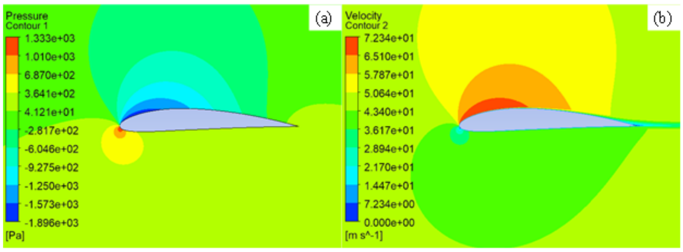


[Fig.2: Lift Coefficient Comparison Between (a) CFD vs. Experimental; (b) XFLR5 vs. Experimental]

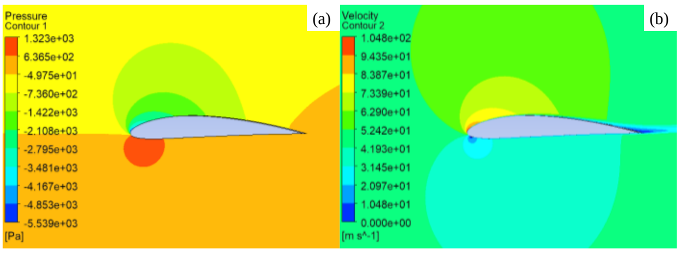
Lift coefficients, velocity contours and pressure contours for five different angles of attack with Reynolds number 1.9×10^5 were obtained upon performing simulations.



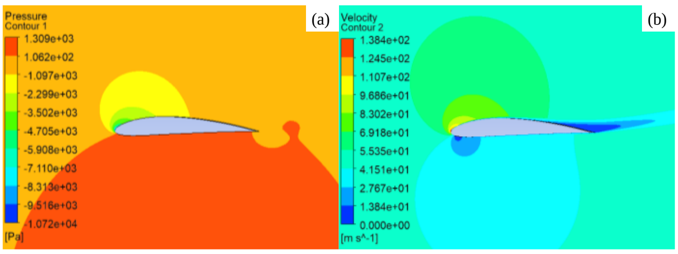
[Fig.3:(a) Pressure and (b) Velocity Contour for AoA 0°]



[Fig.4:(a) Pressure and (b) Velocity Contour for AoA 5°]

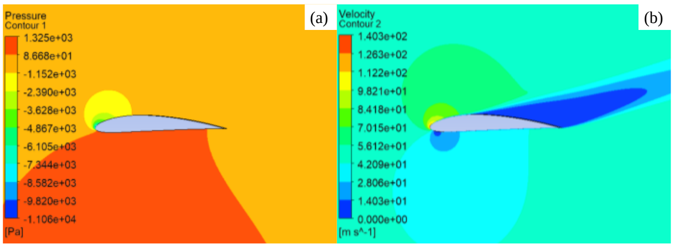


[Fig.5:(a) Pressure and (b) Velocity Contour for AoA 10°]



[Fig.6:(a) Pressure and (b) Velocity Contour for AoA 5°]

© Copyright: All rights reserved.



[Fig.7:(a) Pressure and (b) Velocity Contour for AoA 5°]

IV. CONCLUSION

In conclusion, this study's findings show a robust relationship between experimental data and Computational Fluid Dynamics (CFD) simulations, highlighting the accuracy of CFD in prediction aerodynamic performance. On the other hand, the XFLR5 Panel approach showed an ability to overestimate lift coefficients at different angles of attack, indicating that it is not a perfect fit for capturing intricate flow phenomena like viscosity and flow separation. This disparity highlights the need to use a variety of analysis techniques when assessing airfoil performance, especially in low-speed applications. Researchers can ensure a thorough understanding of airflow characteristics and improve the accuracy of performance forecasts by combining CFD with other analytical approaches to produce more accurate and precise evaluations.

DECLARATION STATEMENT

Authors are required to include a declaration of accountability in the article, counting review-type articles, that stipulates the involvement of each author. The level of detail differs; Some subjects yield articles that consist of isolated efforts that are easily voiced in detail, while other areas function as group efforts at all stages. It should be after the conclusion and before the references.

After aggregating input from all authors, I must verify the accuracy of the following information as the article's author.

- **Conflicts of Interest/ Competing Interests:** Based on my understanding, this article has no conflicts of interest.
- **Funding Support:** This article has not been funded by any organizations or agencies. This independence ensures that the research is conducted with objectivity and without any external influence.
- **Ethical Approval and Consent to Participate:** The content of this article does not necessitate ethical approval or consent to participate with supporting documentation.
- **Data Access Statement and Material Availability:** The adequate resources of this article are publicly accessible.
- **Authors Contributions:** Each author has individually contributed to the article. The concept and preliminary layout of the modelling were collaborative efforts among all con-tributors. All the simulations were mainly carried out by Neloy Kumar Das, Arup Das,Syeda Tanjila Sarwar .All Authors contributed on data gathering, processing results, and writing the final text and contributed insightful feedback and suggestions. All the authors gave useful feedback, reviewed, and approved the manuscript.

REFERENCES

1. S. Gössling and A. Humpe, "The global scale, distribution and growth of aviation: Implications for climate change," *Global Environmental Change*, vol. 65, Nov. 2020. DOI: https://doi.org/10.1016/j.gloenvcha.2020.102194
2. F. Thomas and J. Milgram, *Fundamentals of Sailplane Design* (Vol. 3), College Park Press, Maryland, 1999. URL: https://www.abebooks.com/Fundamentals-Sailplane-Design-Thomas-Fred-Milgram/10015417460/bd
3. J. D. Anderson and M. Bowden, *Introduction to Flight*, 9th ed., McGraw Hill, 2022. URL: https://dokumen.pub/introduction-to-flight-9nbsped-1260597997-9781260597998.html
4. Deperrois, "XFLR5: Analysis of foils and wings operating at low Reynolds numbers," *Guidelines for XFLR5*, 2009. URL: https://docslib.org/doc/6856685/xflr5-analysis-of-foils-and-wings-operating-at-low-reynolds-numbers
5. M. Rayhan, M. S. Hossain, R. H. Mim, and M. Ali, "Computational and experimental study on the aerodynamic performance of NACA 4412 airfoil with slot and groove," *Heliyon*, vol. 10, e31595, May 2024. DOI: http://doi.org/10.1016/j.heliyon.2024.e31595
6. H. Tebbiche and M. S. Boutoudj, "Passive control on the NACA 4412 airfoil and effects on the lift," in *Design and Modeling of Mechanical Systems-II*, M. Chouchane et al., Eds., Springer, 2015, pp. 775-780. DOI: http://doi.org/10.1007/978-3-319-17527-0_77
7. Q. Qu, X. Jia, W. Wang, P. Liu, and R. K. Agarwal, "Numerical study of the aerodynamics of a NACA 4412 airfoil in dynamic ground effect," *Aerospace Science and Technology*, vol. 38, pp. 56-63, 2014. DOI: https://doi.org/10.1016/j.ast.2014.07.016
8. M. Balasubramaniyan, S. Sakthivel, K. Balaji, S. Prakash, and M. N. Rajan, "Aerodynamic exploration of NACA 4412 airfoil," *International Research Journal of Modernization in Engineering, Technology and Science*, vol. 3, no. 1, pp. 193-200, 2021. URL: https://www.irjmets.com/uploadedfiles/paper/volume3/issue_1_january_2021/5581/1628083226.pdf
9. K. L. Hansen, R. M. Kelso, and B. B. Dally, "Performance variations of leading-edge tubercles for distinct airfoil profiles," *AIAA Journal*, vol. 49, no. 1, pp. 185-194, 2011. DOI: https://doi.org/10.2514/1.J050631
10. S. Dhakal, K. R. Bhattarai, and M. Sanjay, "Computational analysis of the aerodynamic performance of NACA 4412 and NACA 23012 airfoils," *International Research Journal of Engineering and Technology (IRJET)*, vol. 10, no. 12, pp. 265-274, Dec. 2023. DOI: http://doi.org/10.13140/RG.2.2.31495.37284
11. H. Johari, C. Henoch, D. Custodio, and A. Levshin, "Effects of leading-edge protuberances on airfoil performance," *AIAA Journal*, vol. 45, no. 11, pp. 2634-2642, Nov. 2007. DOI: https://doi.org/10.2514/1.28497
12. H. S. Yoon, P. A. Hung, J. H. Jung, and M. C. Kim, "Numerical investigation of how wavy leading edges affect the hydrodynamic characteristics of low aspect ratio wings," *Computers & Fluids*, vol. 49, no. 1, pp. 276-289, 2011. DOI: https://doi.org/10.1016/j.compfluid.2011.06.010
13. Drela, M., and M. Giles. "VISCOUS-INVISCID ANALYSIS OF TRANSONIC AND LOW REYNOLDS-NUMBER AIRFOILS." AIAA JOURNAL, vol. 25, no. 10, 1987, pp. 1347–55. DOI: https://doi.org/10.2514/3.9789
14. S. Ahmed Jurfani, D. Arup, I. M. Maynul and S. S. M. Arif Mahmud, A Bio-inspired Approach to Heavy Vehicle Drag Reduction in Turbulent Flow using Shark Dermal Denticles (June 9, 2024). Proceedings of the 14th International Conference on Mechanical Engineering (ICME 2023). DOI: http://dx.doi.org/10.2139/ssrn.4858852
15. Q. Qu, W. Wang, P. Liu, and R. K. Agarwal, "Airfoil Aerodynamics in Ground Effect for Wide Range of Angles of Attack," *AIAA Journal*, vol. 53, no. 10, pp. 3062–3073, 2015. DOI: https://doi.org/10.2514/1.J053366
16. NACA Airfoil Plotter, Aerospace Toolbox, MathWorks, 2024. URL: https://www.mathworks.com
17. E. N. Jacobs, K. E. Ward, and R. M. Pinkerton, "The characteristics of 78 related airfoil sections from tests in the variable-density wind tunnel," *Langley Memorial Aeronautical Laboratory*, National Advisory Committee for Aeronautics, Rep. 460, Nov. 1933. URL: https://scispace.com/pdf/the-characteristics-of-78-related-airfoil-sections-from-1o8sk4v0hi.pdf.
18. Zulkefli, N. F., Ahamat, M. A., Safri, N. F. M., Nur, N. M., & Rafie, A. S. M. (2019). Lift Enhancement of NACA 4415 Airfoil using Biomimetic Shark Skin Vortex Generator. In International Journal of Recent Technology and Engineering (IJRTE) (Vol. 8,

© Copyright: All rights reserved.

Issue 4, pp. 9231–9234). DOI: https://doi.org/10.35940/ijrte.d9222.118419

19. Rafie, A. S. M., Magaidi, A. M., & Marzuki, O. F. (2019). Aerodynamic Performance of Biomimicry Snake-Shaped Airfoil. In International Journal of Engineering and Advanced Technology (Vol. 9, Issue 2, pp. 3319–3323). DOI: https://doi.org/10.35940/ijeat.b3560.129219
20. Joshi, Y., kumar, A., Roy, A., & Harichandan, A. B. (2019). Moving Surface Boundary Layer Technique For NACA 0012 Airfoil At Ultra-Low Reynolds Number. In International Journal of Innovative Technology and Exploring Engineering (Vol. 9, Issue 2, pp. 3788–3795). DOI: https://doi.org/10.35940/ijitee.b6299.129219

AUTHOR'S PROFILE

Neloy Kumar Das, recently pursuing an MSc in Mechanical Engineering at Bangladesh University of Engineering and Technology (BUET), Dhaka. His academic focus includes computational modeling, thermofluidic systems, and sustainable technologies.

Arup Das, Lecturer in the Department of Mechanical Engineering at the Military Institute of Science and Technology (MIST), Dhaka, Bangladesh. His research focuses on bioinspired fluid dynamics, heat transfer enhancement, and aerodynamic optimization. He has authored several peer-reviewed journal articles and conference papers.

Syeda Tanjila Sarwar, pursuing her MSc in Mechanical Engineering at Bangladesh University of Engineering and Technology (BUET), Dhaka. Her research interests include heat transfer, computational fluid dynamics (CFD), and renewable energy systems. She has authored several peer-reviewed journal articles and conference papers.

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of the Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)/ journal and/or the editor(s). The Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP) and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

© Copyright: All rights reserved.