STM JOURNALS

Journal of
Offshore Structure and Technology

ISSN: 2349-8986
Volume 13, Issue 1, 2026
January–April
DOI (Journal): 10.37591/JoOST

https://journals.stmjournals.com/joost

Case Study JoOST

Hydrodynamic Analysis of Symmetrical NACA Airfoils: A CFD Simulation Approach

Sahul Muhique[1], Kalyan Kalita[2], Kalyan Das[3],*

Abstract

This study presents a comprehensive numerical investigation of the hydrodynamic characteristics of selected National Advisory Committee for Aeronautics (NACA) airfoil profiles using advanced computational fluid dynamics (CFD) simulations performed in ANSYS Fluent. The primary objective is to evaluate the fluid–structure interaction behavior of these airfoils when subjected to marine flow conditions relevant to offshore and submerged engineering applications. By modeling the interaction between incompressible fluid flow and various NACA geometries, the study systematically analyzes key hydrodynamic parameters, including pressure distribution, velocity fields, drag forces, and corresponding drag coefficients, which are critical for assessing the operational stability and performance of offshore structures. The numerical methodology employs a pressure-based solver with appropriate turbulence modeling to accurately capture flow separation and wake development around the airfoil surfaces. A detailed mesh sensitivity analysis is conducted to ensure grid independence and numerical stability, enabling precise prediction of drag forces and hydrodynamic coefficients. The influence of geometric parameters, such as airfoil thickness and camber, on hydrodynamic performance is examined through comparative graphical and tabular analyses. These analyses provide insights into the interdependencies between airfoil geometry and flow behavior under varying flow conditions. To validate the reliability of the numerical results, a rigorous error analysis is performed by comparing simulated outcomes across multiple mesh configurations, with all deviations maintained within a 5% margin of error. The findings of this study contribute to the optimization of submerged energy platforms, offshore structural components, and deep-sea exploration vessels by supporting informed design decisions aimed at minimizing hydrodynamic resistance and enhancing structural stability in marine environments.

Keywords: ANSYS Fluent, NACA airfoils, hydrodynamic analysis, drag coefficient, offshore structures

INTRODUCTION

The hydrodynamic analysis of National Advisory Committee for Aeronautics (NACA) airfoils is a fundamental aspect of modern marine and offshore engineering, particularly in the design and optimization of submerged lifting surfaces. When operating underwater, these airfoil geometries function as hydrofoils, and their hydrodynamic performance directly influences the efficiency, stability, maneuverability, and operational safety of underwater vehicles and offshore structures. Parameters such as lift, drag, pressure distribution, and flow separation characteristics play a decisive role in determining the energy consumption and controllability of submarines, autonomous underwater vehicles (AUVs), remotely operated vehicles (ROVs), and submerged energy systems [1–3].

*Author for Correspondence
Kalyan Das
E-mail: kkdas1971@gmail.com

[1]Student, Department of Mechanical Engineering, Assam Engineering College, Jalukbari, Kamrup Metro, Assam, India.
[2]Professor, Department of Mechanical Engineering, Assam Engineering College, Jalukbari, Kamrup Metro, Assam, India

Received Date: January 15, 2026
Accepted Date: February 09, 2026
Published Date: February 16, 2026

Citation: Sahul Muhique, Kalyan Kalita, Kalyan Das. Hydrodynamic Analysis of Symmetrical Naca Airfoils: A CFD Simulation Approach. Journal of Offshore Structure and Technology. 2026; 13(1): 31–38p.

For naval architects and marine engineers, the accurate characterization of the flow field around NACA profiles is essential to ensure that underwater vehicles can maneuver efficiently at target velocities while maintaining structural integrity under varying hydrodynamic loads. Improperly optimized hydrofoil geometries may lead to excessive drag, unsteady flow behavior, or early flow separation, all of which negatively affect propulsion efficiency and operational endurance. Consequently, understanding the relationship between airfoil geometry and hydrodynamic response is critical for achieving reliable and cost-effective marine designs [4–6].

Computational fluid dynamics (CFD) has emerged as a powerful and widely adopted tool in offshore engineering for predicting complex flow phenomena that are difficult and expensive to measure experimentally in marine environments. CFD enables detailed visualization of velocity fields, pressure contours, and wake structures, thereby providing valuable insight into the underlying physics governing fluid–structure interaction. In this study, CFD simulations are conducted using ANSYS Fluent to analyze multiple NACA airfoil profiles under submerged flow conditions. The objective is to establish quantitative relationships between geometric parameters, such as camber and thickness, and key performance indicators, including drag force and drag coefficient. The outcomes of this investigation contribute to the optimization of hydrofoil-based components used in underwater vehicles and offshore structures, ultimately enhancing hydrodynamic efficiency and operational reliability [6–10].

Modeling

A 4-digit NACA airfoil was examined in this study. The NACA airfoil series is controlled by four digits that designate the camber, position of the maximum camber, and thickness. If an airfoil number is NACA MPXX, then:

M is the maximum camber as a percentage of the chord.

P is the distance of the maximum camber from the airfoil leading edge in tenths of the chord.

XX is the maximum thickness of the airfoil as a percentage of the chord (Figures 1 and 2) and (Tables 1 and 2).

The NACA profiles simulated in this project are:

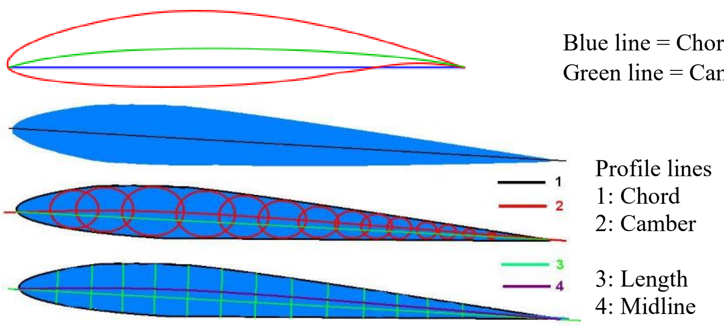

Figure 1. Geometry of NACA profile.

Table 1. NACA profiles.

NACA profiles	Length (m)	Thickness (m)
NACA 0008	5.14287	0.41143
NACA 0010	5.14287	0.51429
NACA 0015	5.14287	0.77143
NACA 0018	5.14287	0.92572
NACA 0020	5.14287	1.02857
NACA 0024	5.14287	1.23429
NACA 0030	5.14287	1.54286

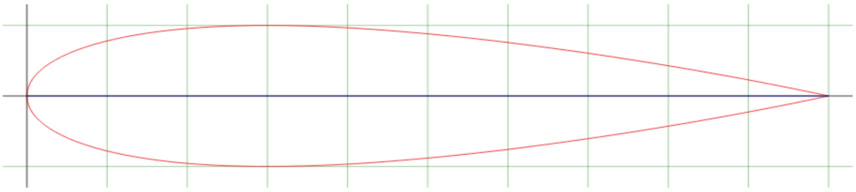

Figure 2. NACA0020.

Table 2. Meshing configuration.

Type	Sphere of influence		Relevance	Vertex sizing	Element shape
	Radius	***Element size***		***Element size***	
C	1 m	0.01 m	+100	0.001 m	Tetrahedron

All the appendages have NACA four-digit airfoil profiles given by:

$$\frac{y_t}{c} = \pm\left[1.4845\sqrt{\frac{x}{c}} - 0.6300\frac{x}{c} - 1.7580\left(\frac{x}{c}\right)^2 + 1.4215\left(\frac{x}{c}\right)^3 - 0.5075\left(\frac{x}{c}\right)^4\right]$$

Where,
c: Chord Length,
t/c *:* Maximum thickness-to-chord ratio;
$x = 0$ *:* Leading edge and
$x = c$: Trailing edge with non-zero thickness

By design, symmetrical four-digit NACA airfoils reach their peak thickness at a position 30% of the chord length from the leading edge.

Meshing Configuration

A grid convergence study was conducted to determine the optimal mesh density. The objective was to produce a mesh fine enough to eliminate spatial discretization errors, that is, achieving grid independence, without incurring excessive computational overhead.

A grid independence study was conducted to ensure that the results were independent of the spatial resolution. This involved systematically increasing the mesh resolution until the aerodynamic coefficients stabilized, indicating that the numerical data obtained reached a state of grid invariance.

The evaluation of the pressure drags across the NACA airfoil series provides a framework for enhancing the performance of submerged assets. The implementation of these optimized profiles allows for a reduction in the total drag and an increase in operational stability within dynamic current environments. Consequently, the findings suggest that adapting these specific geometries can significantly improve the propulsion efficiency and range of AUVs and other deep-sea platforms.

The computational domain was discretized using the patch-conforming tetrahedral method. To capture the high-gradient flow features, a sphere of influence was implemented at the vertex with a 1 m radius and a local element size of 0.01 m. Furthermore, the airfoil boundaries were refined using edge sizing with a fixed element length of 0.001 m to ensure an accurate boundary layer resolution.

Boundary Condition

- The simulation utilized a velocity inlet boundary condition evaluated across three discrete flow regimes: 10, 30, and 50 m/s.
- Density of water: 998.2 kg/m^3.

- Temperature of water (operating temperature): 298.15 K.
- Dynamic viscosity of water: 0.001003 N-s/m^2.
- Density of material of construction of submarine: 2719 kg/m^3.
- The relationship that provides the analytical basis for calculating the drag force is defined by the following equation:
  $$F_D = \frac{1}{2} \times C_D \times \rho \times A \times v^2$$
- Fluent uses the following equation to compute the drag coefficient formulated in the simulated flow field:
  $$c_d = \frac{1}{\rho v^2 A} \int_s dA(p - p_0)(\hat{n} \cdot \hat{\imath}) + \frac{1}{\rho v^2 A} \int_s dA(\hat{t} \cdot i) T_w$$
- The computational domain utilized a uniform velocity at the inlet of 50 m/s to characterize the flow field around the NACA airfoils.

ANALYSIS AND DISCUSSION

The hydrodynamic characteristics of the NACA airfoils were evaluated by analyzing the pressure contours and velocity profiles at a flow velocity of 50 m/s. The analysis initially focused on the Cd versus iteration curve to ensure numerical stability, and the simulations were continued until the drag coefficient, and the resulting drag force reached a converged state. To verify the results, the modal value of the simulated Cd was identified and used to calculate the drag force. A comparative error analysis was performed between the software-generated and analytical values to validate the model accuracy. These flow field visualizations provide a comprehensive understanding of hydrodynamics across airfoil surfaces.

Further examination of the pressure contours revealed distinct pressure gradients between the leading and trailing edges of the airfoils, highlighting the influence of geometric parameters, such as camber and thickness, on the flow behavior. High-pressure regions were consistently observed near the stagnation point at the leading edge, whereas low-pressure zones developed along the suction side, contributing significantly to the overall drag characteristics. The variations in pressure recovery along the chord length directly affected the wake formation and flow separation tendencies.

Velocity magnitude contours and streamline plots demonstrated smooth flow attachment along the airfoil surface for the optimized profiles, whereas thicker or higher-camber configurations exhibited localized flow deceleration and minor separation near the trailing edge. These phenomena resulted in increased drag coefficients owing to enhanced viscous effects and wake expansion. The velocity profiles downstream of the airfoils further confirmed the relationship between wake width and drag magnitude.

The comparative error analysis indicated that the discrepancy between the analytical and CFD-predicted drag forces remained within a 5% tolerance, confirming the reliability of the numerical model and mesh resolution. Overall, the results emphasize the importance of airfoil geometry selection for minimizing hydrodynamic drag and improving performance. The findings offer valuable insights into the design and optimization of hydrofoil-based components used in offshore structures, submerged energy devices, and underwater vehicles operating under high-velocity flow conditions.

RESULT

A series of NACA airfoils was simulated under a uniform velocity profile of 50 m/s. When compared with the analytical values, the maximum observed relative error was 1.95%. This high degree of conformity falls well within the acceptable tolerance levels for validation and thus confirms the accuracy and reliability of the current CFD model (Tables 3 and 4) and (Figures 3–7).

Table 3. Table for the NACA profiles along with their respective thickness, C_d, and drag forces.

NACA series	Thickness(m)	C_d	Pressure drag (N)	Theoretical value (N)	% error
NACA 0008	0.41143	0.02099	19013.33	18640.82	1.95
NACA 0010	0.51429	0.0298	32464.35	32036.49	1.32
NACA 0015	0.77143	0.0611	97267.79	98117.69	0.87
NACA 0018	0.92572	0.0735	143264.26	144318.22	0.74
NACA 0020	1.02857	0.0919	186648.67	185498.79	0.62
NACA 0024	1.23429	0.1162	301262.82	298604.44	0.88
NACA 0030	1.54286	0.1472	479604.85	478255.19	0.28

Table 4. Table for the NACA Profiles along with their respective drag forces and errors with respect to Equation (1).

NACA profile	Thickness (m)	Simulated drag force values (N)	Calculated value (N) (using Equation (1))	Error (%)
NACA 0008	0.41143	19513.33	19399.9660	0.59
NACA 0010	0.51429	33457.60	33626.0448	0.50
NACA 0015	0.77143	91137.61	91350.1259	0.23
NACA 0018	0.92572	143020.635	143180.6936	0.12
NACA 0020	1.02857	185558.23	185625.9888	0.04
NACA 0024	1.23429	291193.64	290963.2	0.08
NACA 0030	1.54286	505467.51	504319.7145	0.23

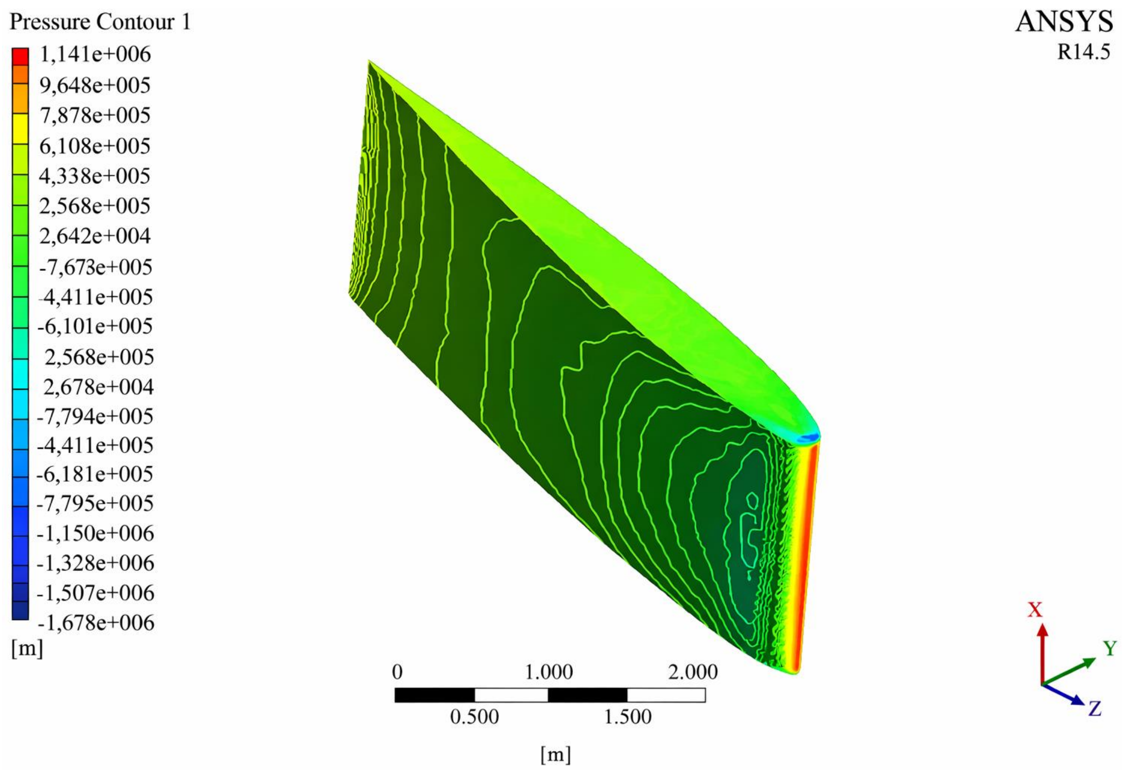

Figure 3. Pressure contour over NACA 0010 profile at a velocity of 50 m/s.

In the above Figure 3, the curve formed by the thickness versus drag force seems to almost coincide with the power curve ($y = ax^n + c$). Hence, by using MATLAB, an equation or relation between the thickness of the NACA profile and pressure drag force at 50 m/s velocity was obtained through the curve fitting process.

The equation thus obtained can be written as:

$$F = 173183.87t^{2.46486} \tag{1}$$

Where, 'F' is the pressure drag force (N) and 't' is the thickness of the NACA profile (m).

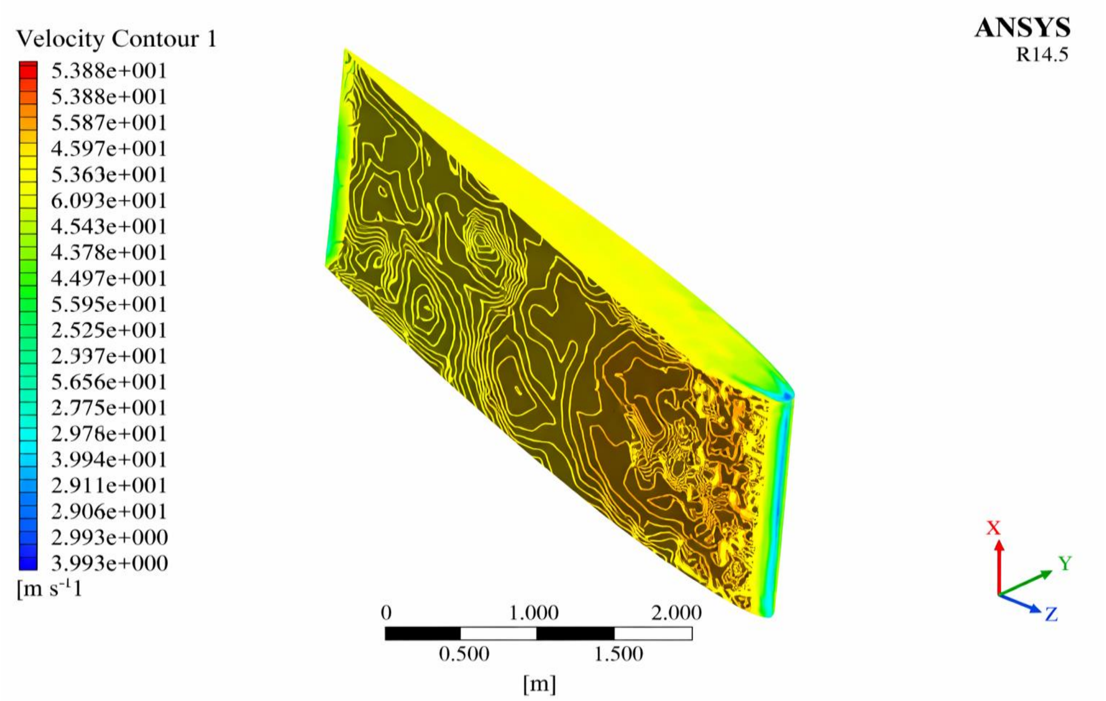

Figure 4. Velocity contour over the NACA 0010 profile at a velocity of 50 m/s.

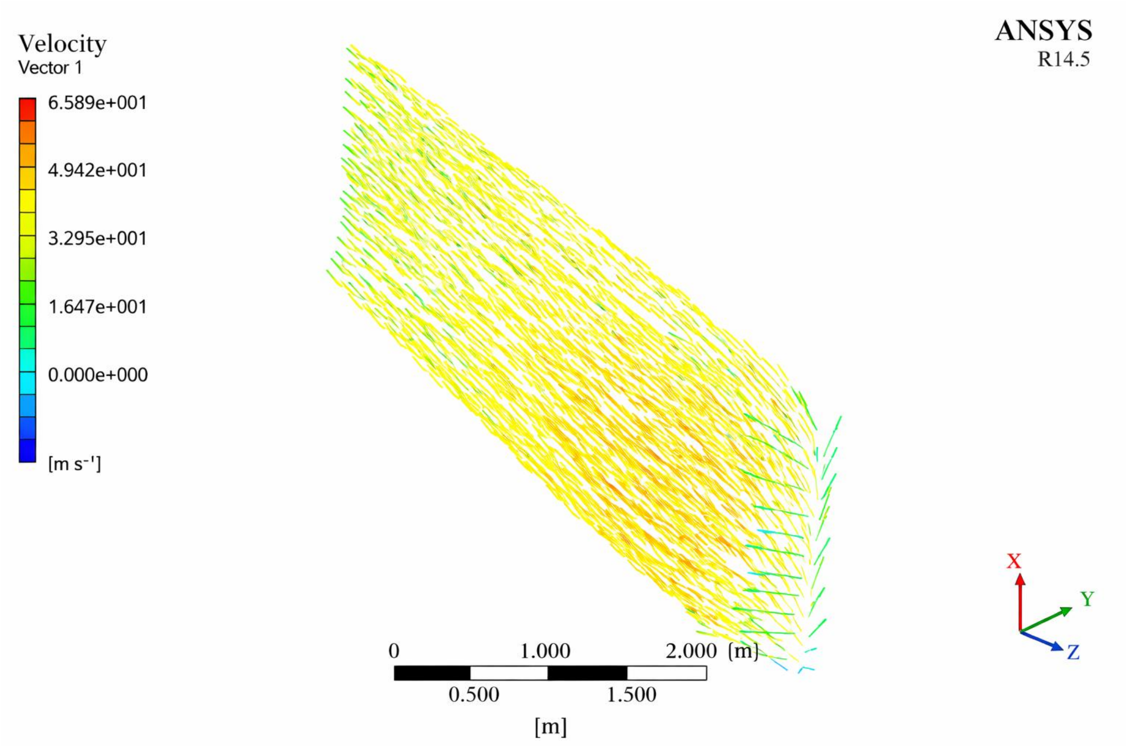

Figure 5. Velocity streamline over the NACA 0010 profile at a velocity of 50 m/s.

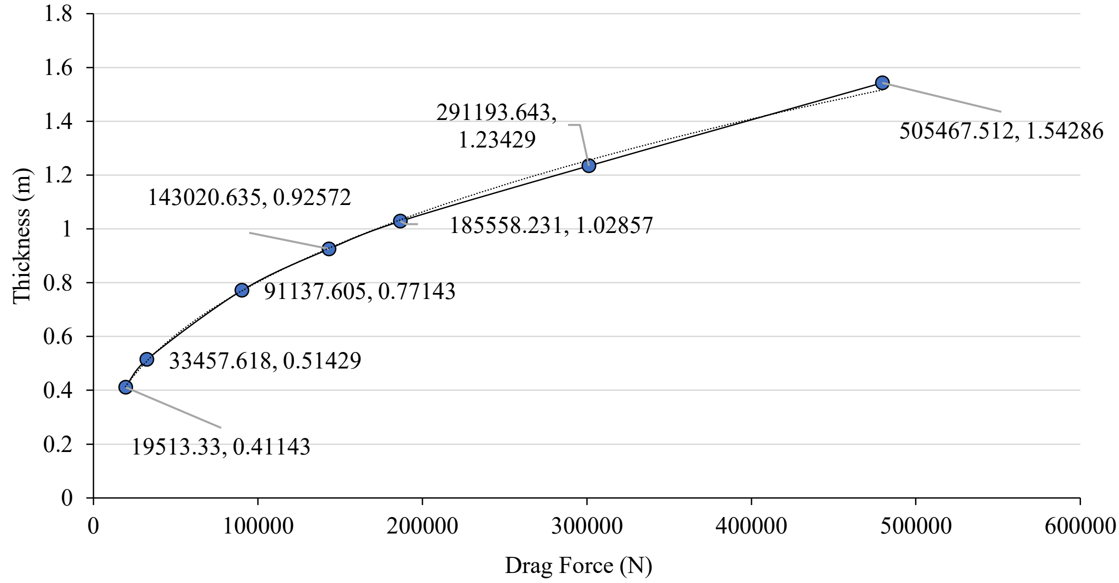

Figure 6. Thickness versus drag force for various NACA profiles.

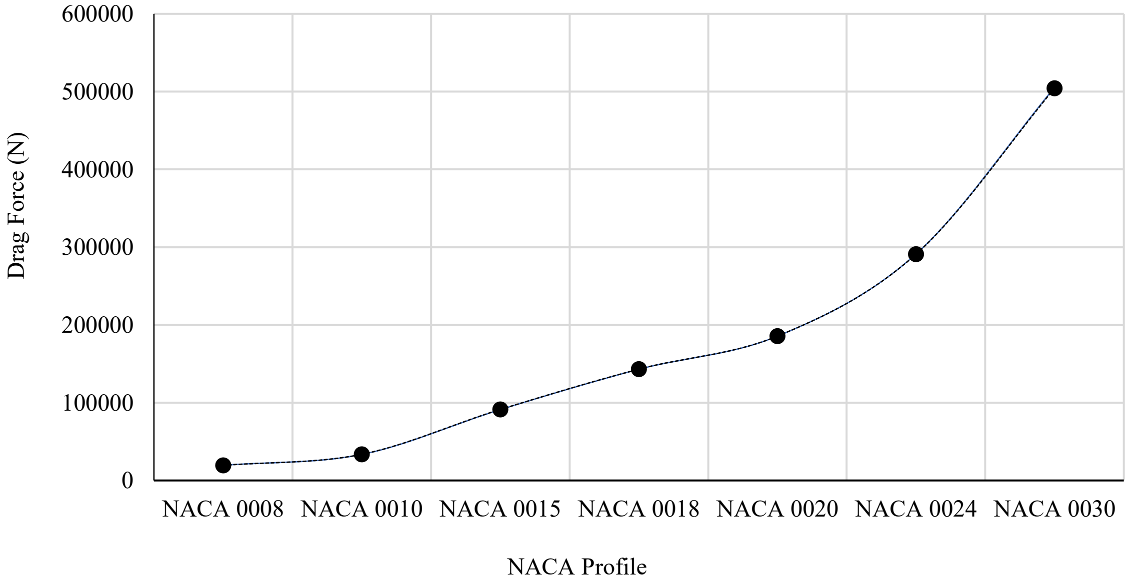

Figure 7. Comparison between simulated and calculated drag force values for various NACA profiles.

CONCLUSION

This study utilized ANSYS Fluent to perform a comprehensive flow analysis of NACA airfoils in motion, facilitating a comparative investigation of their geometric parameters to establish fundamental hydrodynamic relationships.

Seven standards symmetrical NACA airfoil profiles—NACA 0008, NACA 0010, NACA 0015, NACA 0018, NACA 0020, NACA 0024, and NACA 0030—were analyzed to determine their respective drag coefficients (Cd) and total drag forces. These simulations facilitated the characterization of the functional relationships between the airfoil thickness and both Cd and drag force. Furthermore, a comparative analysis between the simulated results and theoretical drag values revealed nearly identical trends, with the plotted data points almost entirely coinciding.

The relationship between the profile thickness and drag force exhibited a clear Power Curve distribution, that is, $y = ax^n + c$. Consequently, a regression analysis was performed using MATLAB to derive an equation for the NACA profiles with a length of 5.14 m traveling through water at a velocity of 50 m/s. The resulting mathematical model showed exceptional agreement with the CFD data, yielding a maximum error of only 0.59%. This high degree of correlation falls within the standard engineering tolerances, confirming the validity of the derived equation.

The findings of this study offer significant contributions to the optimization of submerged vehicles by facilitating the reduction of hydrodynamic drag and enhancement of stability in oceanic environments. This CFD-driven methodology establishes a robust framework for improving the operational performance of offshore platforms and deep-sea exploration systems. Future research may extend these results by integrating complex variables, such as wave-current interactions and multibody hydrodynamics, further refining their applicability to sophisticated offshore engineering challenges.

REFERENCES

1. Baker C. Estimating drag forces on submarine hulls. Defence R&D Canada-Atlantic; 2004. Report No.: CR 2004-125.
2. Mackay M. The standard submarine model: a survey of static hydrodynamic experiments and semi-empirical predictions. Defence R&D Canada-Atlantic; 2003. Report No.: TR 2003-079.
3. Kukner A. Investigation of flow distribution around a submarine. J Nav Sci Eng. 2006;12(2): 1–26.
4. Molland AF, Turnock SR. Wind tunnel investigation of the influence of propeller loading on ship rudder performance [technical report]. Ship Science Report No. 46. 1991.
5. Larsson L, Stern F, Visonneau M. Numerical ship hydrodynamics: an assessment of the Gothenburg 2010 workshop. Dordrecht: Springer; 2014.
6. Han KJ. Numerical optimization of hull/propeller/rudder configurations [thesis]. Gothenburg: Chalmers University of Technology, Department of Shipping and Marine Technology; 2008.
7. Kawabuchi M, Kubota M, Ishikawa S. Development of technology to estimate the flow field around ship hull considering wave making and propeller rotating effects. Mitsubishi Heavy Industries Technical Review. 2016;53(2):53–57.
8. Da-Qing L. Investigation on propeller-rudder interaction by numerical methods [thesis]. Gothenburg: Chalmers University of Technology, Department of Shipping and Marine Technology; 1994.
9. Mackay M. The standard submarine model: a survey of static hydrodynamic experiments and semi-empirical predictions. Defence R&D Canada–Atlantic TR. 2003;2003-2079.
10. Moonesun M. Naval Submarine Body form Design and Hydrodynamics. Chisinau: Lambert Academic Publishing; 2023.

