

AKADÉMIAI KIADÓ

Pollack Periodica •
An International Journal
for Engineering and
Information Sciences

18 (2023) 2, 29–34

DOI:

[10.1556/606.2023.00770](https://doi.org/10.1556/606.2023.00770)

© 2023 Akadémiai Kiadó, Budapest

ORIGINAL RESEARCH
PAPER



*Corresponding author.

E-mail: dr.ahmed.jameel@sadiq.edu.iq



AKJournals

The aerodynamic analysis of iced and clean NACA 4410 and 0012 airfoils

Ahmed Jameel Hamoud*

Department of Computer Engineering Technology, Faculty of Information Technology, Imam Ja'afar Al-sadiq University, Iraq

Received: December 11, 2022 • Revised manuscript received: January 30, 2023 • Accepted: February 12, 2023

Published online: May 16, 2023

ABSTRACT

The aims of the work is to develop a numerical iced airfoil; quarter round forward ice was tested on NACA 4410 and NACA 0012 airfoils at zero and non-zero angles of attack, and Reynolds numbers equal to $(2 \cdot 10^5, 2.4 \cdot 10^5)$ based on airfoil chord and Mach number 0.04. The two-dimensional steady-state momentum equations with the continuity equation have been solved applying a finite volume method to examine the turbulent flow over a clean and iced airfoil. The cambered airfoil NACA 4410 spends less power than the unsymmetrical airfoil at the same angle of attack. The reported numerical results demonstrated that for airfoil NACA 4410, the drag was increased by 40%, and the lift was reduced by 22%. However, for airfoil NACA 0012, the drag was increased by 43%, and the lift was decreased by 21%.

KEYWORDS

airfoil, iced airfoil, NACA 4410, NACA 0012

1. INTRODUCTION

There is a significant danger in-flight icing. It kills the smooth airflow, raises the drag, degrades the control authority, and reduces an airfoil's ability to lift. The real weight of the aircraft's ice is inferior to the damage it creates to airflow. On each exposed frontal surface of the plane, ice can be produced not only on the wings, and windshield or propeller, but also on the antennas, vents, intakes, control surfaces, and cowlings. A light aircraft may become so iced up under moderate to extreme conditions that it is impossible to continue the flight. The aircraft could stall at much slower speeds and lower attack angles than usual. This action can uncontrollably roll, or pitch and recovery may be unlikely. Extreme icing is defined by the Aeronautical Information Manual (AIM) of the United States as ice levels that are so high that anti-icing and de-icing equipment are unable to control the level of danger and require immediate diversion of flights [1].

Many researchers investigated the airflow over iced airfoils applying different numerical methods.

The researchers studied the influence of propeller slipstream on the aerodynamic characteristics of a static-wing Micro Air Vehicle (MAV) by simplifying it into an actuator volume and an actuator disk [2, 3], while in other analytically studied the distribution of pressure and velocity on airfoils of arbitrary sizes, taking into account the strict limit conditions. A precise second-order [4] and third-fifth order method is developed in inviscid incompressible flows [5, 6].

Other transport category aircraft was used in this investigation for a number of natural icing flying experiments and icing wind tunnel testing. Similar test settings were used to examine the outcomes on shielded and unprotected surfaces of the swept wings and empennage [7]. Laser surface structuring was either used in this study or as an alternative

method to coatings for producing icephobic characteristics and assisting deicing and anti-icing procedures on aerodynamic surfaces. Through direct laser interference patterning, a titanium airfoil with a non-planar surface is directly fabricated with hierarchical micro- and nano-structures [8]. There are researchers focused on improving a multi-element airfoil that can tolerate ice. With the help of the crucial horn-shaped ice that has collected during the holding phase, dual-objective optimization is conducted [9, 10], while others investigated the aerodynamic characteristics of a NACA 0012 wing geometry at reduced Reynolds numbers and angles of attack ranging from 0° to 90° [11].

2. MATHEMATICAL MODEL

The mathematical model of the physical domain and fluid flow is introduced in this chapter. This model includes the two-dimensional differential governing equations of flow analysis (continuity and momentum) in Cartesian coordinates with two equations for (k - ϵ) turbulent model.

2.1. The governing differential equations

Two different airfoils are used in this work, namely; NACA 0012 with a chord of (350 mm) and NACA 4410 with a chord of (250 mm), as it is shown in Fig. 1. Mixed cases are correctly investigated clean airfoil and iced airfoil with quarter round forward iced airfoil at zero and non-zero angles of attack.

The assumptions used for external airflow over the airfoils are:

1. Steady-state, two-dimensional, incompressible flow;
2. The fluid is Newtonian;
3. The working fluid obeys the law of ideal gases;
4. The physical properties of the air are constant;
5. It is assumed that the ice formation is stretched from the wing root to the tip with no spanwise variations.

The governing equations that describe the two-dimensional flow over an airfoil are (Verestage and Malalasekera [12]).

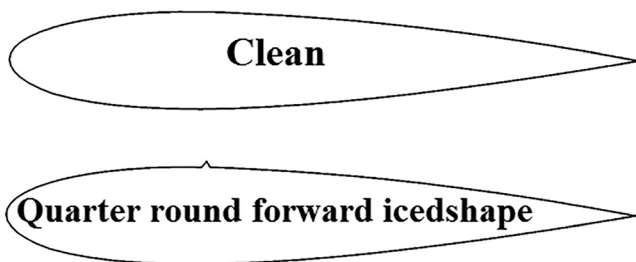


Fig. 1. Airfoil shapes clean and without ice

2.1.1. Conservation of mass. Conservation of mass denotes that mass cannot be created nor destroyed. Applying mass conservation law on a 2D control volume, the conservation equation is:

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

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

2.1.2. Momentum equations: force balance. Momentum equations are obtained from Newton's second law of motion that confirms that a fluid particle's rate of momentum change is equivalent to the totality of the forces applied to it. The derived equations of momentum can be drawn up as:

- u -momentum (x -direction)

$$\frac{\partial}{\partial x} (\rho uu) + \frac{\partial}{\partial y} (\rho uv) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left(\mu_{eff} \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu_{eff} \frac{\partial u}{\partial y} \right) + S_u, \quad (2)$$

- v -momentum (y -direction)

$$\frac{\partial}{\partial x} (\rho vu) + \frac{\partial}{\partial y} (\rho vv) = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left(\mu_{eff} \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mu_{eff} \frac{\partial v}{\partial y} \right) + S_v, \quad (3)$$

where μ_{eff} is the effective viscosity coefficient which is commonly evaluated as:

$$\mu_{eff} = \mu_l + \mu_t, \quad (4)$$

where μ_l is the laminar viscosity; μ_t is the turbulent eddy viscosity; S_u is a source term of u in the x -direction; S_v is a source term of v in the y -direction.

2.2. Turbulence

Turbulence describes the random and chaotic motion of the viscous fluid flow. Turbulent flow can be visualized by rotational motion commonly referred to as turbulent eddies or rotational movement of the stream, are defined over a wide range of length scales. Length scales represent the size of turbulent intensity, and it is dependent on the size of the model and Reynolds number (Re) Launder and Spalding [13],

$$T_i = 0.16 \cdot Re^{-1/8}. \quad (5)$$

Launder and Spalding's two-equation k -turbulence prototype of kinetic energy (k) and its dissipation rate (ϵ) [13]. This model shows the turbulent viscosity of the local values ϵ , k , and ρ by the expression:

$$\mu_t = \rho c_\mu k^2 / \epsilon, \quad (6)$$

where c_μ is an experimental constant.



The distribution of k and ε over the stream field is estimated from the following semi-pragmatic transport equations for k and ε , Ideriah, [14],

- Turbulence energy, k

$$\frac{\partial}{\partial x}(\rho u k) + \frac{\partial}{\partial y}(\rho v k) = \frac{\partial}{\partial x} \left(\frac{\mu_{eff}}{\sigma_k} \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial y} \left(\frac{\mu_{eff}}{\sigma_k} \frac{\partial k}{\partial y} \right) + G - \rho \varepsilon, \quad (7)$$

- Energy dissipation rate, ε

$$\frac{\partial}{\partial x}(\rho u \varepsilon) + \frac{\partial}{\partial y}(\rho v \varepsilon) = \frac{\partial}{\partial x} \left(\frac{\mu_{eff}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x} \right) + \frac{\partial}{\partial y} \left(\frac{\mu_{eff}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial y} \right) + c_1 \frac{\varepsilon}{k} G - c_2 \rho \frac{\varepsilon^2}{k}, \quad (8)$$

In the turbulence transport equations G represents the production of the kinetic energy from mean velocity gradients, and σ_ε , σ_k are the effective Prandtl number,

$$G = \mu_t \left\{ 2 \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 \right] + \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right)^2 \right\} + S_G, \quad (9)$$

where S_G is given by Ideriah, [12],

$$S_G = -\frac{2}{3} \mu_t \left[\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right]^2 - \frac{2}{3} \rho k \left[\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right], \quad (10)$$

where S_G is the source term of production of G .

The values of the empirical constant used here are given in Table 1.

These values have been specified from tests for water and air for essential turbulent shear flows, together with compatible shear flows Launder and Spalding, [11].

2.3. Boundary conditions

The border conditions at the airfoil are shown in Fig. 1:

- No-slip boundary condition is applied on a solid surface. All solid surfaces have $u = 0$, $v = 0$.
- Uniform and constant free stream velocity at $x = -c$ and at any y , $u = u_\infty$, $v = 0$ where c is the airfoil chord.

3. COMPUTATIONAL METHODOLOGY

The Computational Fluid Dynamics (CFD) is implemented in this work by applying a finite volume method. Fluent software, v6.26, 2009 is commonly solved the governing differential equations for airflow over a clean and iced airfoil. The methodology to use this software is presented in the following sections.

Table 1. Values of constants in the k - ε model

C_μ	C_1	C_2	σ_k	σ_ε
0.09	1.44	1.92	1.0	1.3

3.1. GAMBIT software

GAMBIT v2.4.6 [15]: mesh generation software is a discretization software used to mesh the region over the airfoil. The control volume in this analysis represents the flow domain over an airfoil. Discretization is the process of subdividing the volumes that represent the fluid and solid regions into smaller quantities called cells. In CFD, flow variables are solved from one of the cells to the next until the solution for the entire control volume is completed.

3.2. FLUENT software

FLUENT v6.26 [16] is the commercial CFD software used to conduct fluid analysis. FLUENT utilizes the discretized domain constructed in GAMBIT to solve the governing equations of the control volume in the present study. GAMBIT or other mesh generation software usually adds to FLUENT software for implementing an integrated meshing capability.

3.3. Simulation of iced airfoil in fluent

The computational method contains three parts; these are solver stage (processing), a pre-processing, and finally, a post-processing phase. The pre-processing phase includes building the computational environment, including the model's geometric assembly, meshing, and finally applying boundary conditions. Table 2 shows the ice dimensions on the airfoils. The solver stage is to submit the model to a CFD system and to solve the governing equations to achieve results. Then, in a post-processing phase, the effects of the solver are analyzed.

3.4. Meshing and building

Model construction, meshing, and assembly require an experimental and error method that several times involves several repetitions before the creation of proper geometry and mesh. Figure 2 represents the unstructured grid generated for an airfoil with and without ice in GAMBIT.

3.5. Governing equations and solution methods

The governing equations are resolved dynamically using an implicit linearization method in which the value of interest in each cell is figured out for each variable by using equations with both known and unknown values from neighboring cells. Increasing the undefined occurs in a sequence of equations eventually, which must be resolved

Table 2. Ice dimensions on the airfoils NACA 0012 and NACA 4410

Quarter ice dimension for NACA 4410			Quarter ice geometry for NACA 0012	
No. of point	x/c	y/c	x/c	y/c
1st point	0.34410	0.09542	0.30941	0.07911
2nd point	0.34911	0.08890	0.32412	0.06000
Center point	0.34412	0.08891	0.30945	0.06000



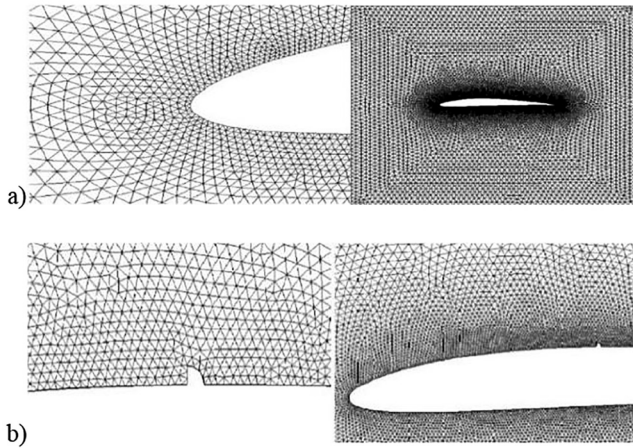


Fig. 2. Mesh for airflows over airfoil NACA 4410, a) clean airfoil; b) quarter round forward iced airfoil

instantaneously. The Semi-Implicit Pressure ReLated Equations (SIPRL) algorithm, coupled with the velocity and the pressure the relationship between pressure and velocity adjustment in order to achieve mass preservation, as well as a pressure scope. Computations require initialization of the flow based on some condition within the model before fluent is started. This initialization method serves as the initial guess of the solution's flow area. The inlet conditions initialize the flow field to the model for all computational simulations. A converged solution requires several repetitions, and the subsequent general processes are included:

1. Updated fluid properties to the current solution;
2. The u , v , momentum equations are each solved with current pressure and mass flux values to update the velocity field;
3. The velocities from phase one cannot recoup the permanency equation locally so that changes are made by amending Eqs (11), (12) to meet continuity [17],

$$u_w = \tilde{u}_w + \dot{u}_w, \quad (11)$$

$$v_s = \tilde{v}_s + \dot{v}_s, \quad (12)$$

where u_w and v_s are the components of velocity that content the continuity equation; $\tilde{u}_w$ and $\tilde{v}_s$ are the components of velocity obtained by sustaining the momentum equations; $\dot{u}_w$ and $\dot{v}_s$ are the corrections to the velocity components needed to satisfy the continuity equation.

Eqs (11) and (12) define $\dot{u}_w$ and $\dot{v}_s$ values for each cell in the domain. The $\tilde{u}_w$ and $\tilde{v}_s$ fields are obtained by solving the momentum equation;

4. Scalar equations are solved for turbulence and with the previously updated values from other variables;
5. A conjunction check is completed to the equations. The defaulting convergence criterion in ANSYS FLUENT is sufficient. This criterion requires that the scaled residuals (R^ϕ) defined by Eq. (13) decrease to 10^{-3} for all equations except the energy equation, for which the criterion is 10^{-6} [18],

$$R^\phi = \frac{\sum_{cells} p |\sum_{nb} a_{nb} \phi_{nb} + b - ap\phi|}{\sum_{cells} p |ap\phi|}, \quad (13)$$

where ap is the center coefficient; a_{nb} is the influence coefficients for the neighboring cells; ϕ is the global variable for the conservation equation at a cell [15].

In general, looked to empower residual plotting and adjust the convergence criteria by means of this dialog box.

3.6. Convergence criteria and tests

The basic approach used in this research to judge a solution's convergence involves monitoring the normalized continuity residuals, x , y , and momentum, k and ϵ after each iteration. The standardized residuals are determined when the unscaled residuals are first found. Un-scaled residuals are the product of numerical error computations, and as these solutions run over several iterations, mathematical errors become smaller and smaller, usually falling many magnitude orders. It is most useful to track the scaled (normalized) residuals to determine convergence. A scaled residual will be given by taking the most significant remaining values from the initial iterations and separating each subsequent remaining value by this amount. Figure 3 displays the scaled residuals of continuity, momentum, k , and ϵ as the solution progresses to the convergence of about 700 iterations. All cases will usually require 500–2000 iterations to attain the defined grade of convergence.

3.7. Calculation of drag and lift forces

The aerodynamic forces on the body are due to only two basic sources [19]:

1. Pressure distribution over the body surface P which acts normal to the surface;
2. Distribute shear stress on the surface of the body Γ , which acts tangential to the surface;
3. It is necessary to find the surface pressure coefficient to determine previously mentioned forces, which defined as:

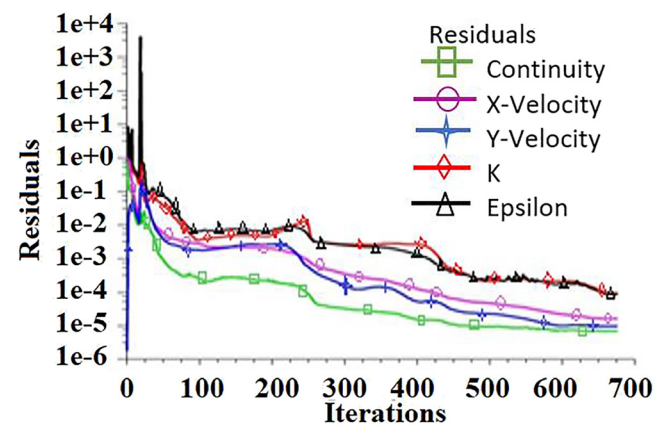


Fig. 3. A big tip gap of residual convergence

$$C_p = \frac{P - P_\infty}{q_\infty}, \quad (14)$$

where P and P_∞ are the local static pressure and free-stream static pressure, respectively; q_∞ is the free-stream dynamic pressure which found from $\rho_\infty V_\infty^2/2$.

The total drag (D_{total}) due to the pressure acting over the entire surface area is calculated [19]:

$$D_{upper} = \int_{L.E}^{T.E} P_u \sin \alpha \, ds_u, \quad (15)$$

$$D_{lower} = \int_{L.E}^{T.E} P_l \sin \alpha \, ds_l, \quad (16)$$

$$D_{total} = D_{upper} + D_{lower}. \quad (17)$$

The total lift (L_{total}) computes from the integration of the following equations:

$$L_{upper} = \int_{L.E}^{T.E} P_u \cos \alpha \, ds_u, \quad (18)$$

$$L_{lower} = \int_{L.E}^{T.E} P_l \cos \alpha \, ds_l, \quad (19)$$

$$L_{total} = L_{upper} + L_{lower}, \quad (20)$$

where α is the angle of attack, ds is the element surface area.

4. RESULTS AND DISCUSSION

It is eminent that any nongeometrical change in the airfoil shape would cause a decrease in the airfoil performance, and this can figure out in lift and drag coefficients. Figure 4a shows the decline in lift coefficient when the ice exists on the airfoil NACA 4410 at Mach number $M = 0.2$ when compared with the clear case. The figure shows the lift coefficient has increased to reach its maximum value at 1.4 when $\alpha = 12^\circ$. However, this value decreased from 1.4 to 1.2 for $M = 0.2$ at the same angle of attack due to the existence of ice. Figure 4b shows the increase in drag coefficient due to the ice compared with that of the clean case, where the ice has caused more disturbances for the air over the airfoil and thereby increases the drag force. The distinction of drag and lift coefficients for numerous Mach numbers over an iced airfoil is shown in the figure below.

The predicted drag performance of clean and iced airfoil NACA 4410 is presented in Fig. 4a. The iced airfoil shows higher values of the drag coefficient than that for the clean airfoil. At 12° angle of attack, the drag coefficient reaches 0.2 for horn iced airfoil. In general, the drag coefficient increased by more than 40% for a positive angle of attack, while the drag coefficient at AOA is increased considerably by about 60%.

Figure 4b displays the lift coefficient was taken as a function of angle of attack for the clean and iced airfoil. Figure 5 shows a comparison of lift to drag coefficient ratio for clean and iced airfoil NACA 4410 at different AOA and

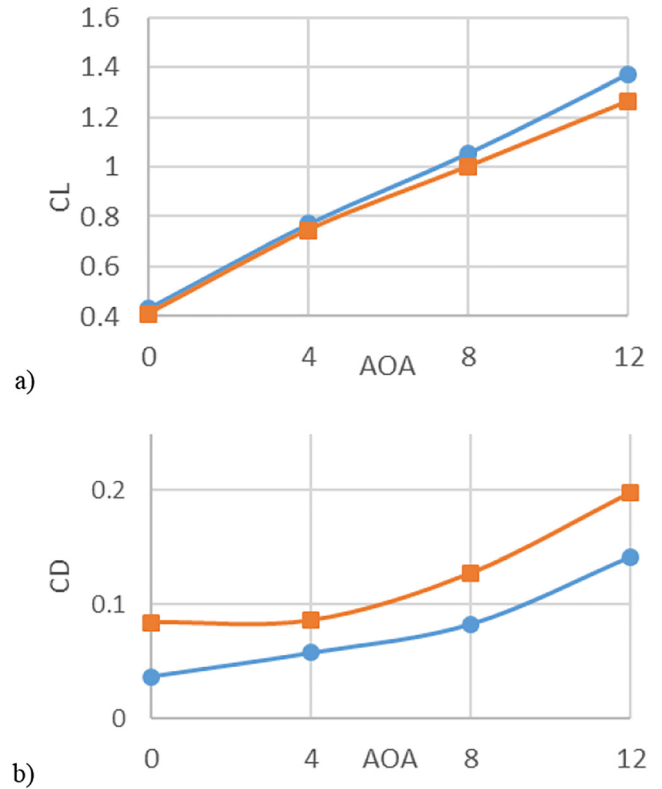


Fig. 4. A numerical comparison between clean (circled curve) and iced (squared curve) NACA 4410 airfoil at different negative Angle Of Attack (AOA) and Mach number (0.2), a) lift coefficient (CL); b) drag coefficient (CD)

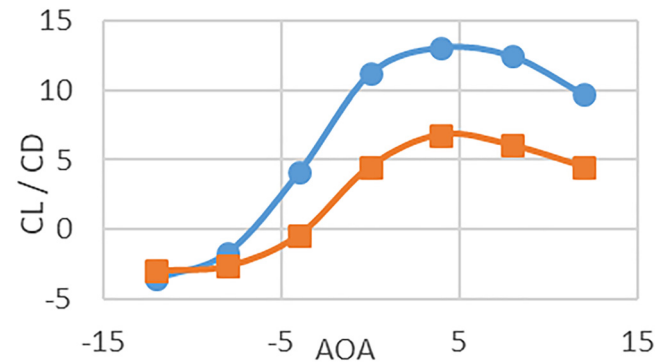


Fig. 5. Numerical CL/CD results for clean (circled curve) and iced (squared curve) NACA 4410 airfoil at different AOA

Re number equal to $2 \cdot 10^5$. A degradation in the lift coefficient is shown with the presence of ice on the airfoil.

5. CONCLUSIONS

An iced airfoil NACA 0012 and NACA 4410 at zero and non-zero angles of attack have been studied at this paper through a precise computational investigation of their aerodynamic characteristics. As stated by the preceding discussion, the following conclusions can be assumed:



1. The drag is increased by 25%, and the lift is decreased by 33.8% when the quarter forward ice shape presence on the airfoil NACA 4410 at an angle of attack equal to 4° ;
2. The results are compatible with previous literature, which demonstrates that cambered airfoils produce lift more than symmetrical straight airfoils and less drag at the same AOA;
3. A steep pressure decrease is reported for an iced airfoil with a quarter round forward iced cases at the location of ice (which was at the maximum thickness of the upper surface of the airfoil);
4. The drag and lift coefficients results for the ice cases show a deviation from that of the clear example, which indicates that the presence of ice on the airfoil is dangerous and could cause significant problems for the pilots;
5. For a negative angle of attack (0° , -4°), the pressure coefficient C_p at iced airfoil lower surface was less than that of the upper surface.

REFERENCES

- [1] S. Li and R. Paoli, "Aircraft icing severity evaluation," *Encyclopedia*, vol. 2, no. 1, pp. 56–69, 2022.
- [2] K. Chinwicharnam, E. D. G. Ariza, M. J. Moschetta, and C. Thiopyopas, "A computation study on the aerodynamic influence of interaction wing-propeller for a tilt-body MAV," *Aircraft Eng. Aero. Technol.*, vol. 87, no. 6, pp. 521–529, 2015.
- [3] K. A. Kasim, S. Mat, I. S. Ishak, and S. Mansor, "Propeller locations study of a generic delta wing UAV model," *Aircraft Eng. Aero. Technol.*, vol. 93, no. 1, pp. 127–138, 2020.
- [4] D. Mateescu and M. Abdo, "Efficient second-order analytical solutions for airfoils in subsonic flows," *Aero. Sci. Technol.*, vol. 9, no. 2, pp. 101–115, 2005.
- [5] N. R. Gande and A. A. Bhise, "Modified third and fifth order WENO schemes for inviscid compressible flows," *Numer. Algorithms*, vol. 88, pp. 249–279, 2020.
- [6] O. Voznyak, O. Savchenko, N. Spodyniuk, I. Sukholova, M. Kasynets, and O. Dovbush, "Air distribution efficiency improving in the premises by rectangular air streams," *Pollack Period.*, vol. 17, no. 3, pp. 111–116, 2022.
- [7] H. Tanaka, T. Ahmad, Y. Utsumi, S. Yonemoto, and G. Ichikawa, "Icing on full-scale swept wings in flight and icing wind tunnel," *J. Aircraft*, vol. 59, no. 6, pp. 1–13, 2022.
- [8] S. Alamri, V. Vercillo, A. I. Aguilar-Morales, F. Schell, M. Wetterwald, A. F. Lasagni, E. Bonaccorso, and T. Kunze, "Self-limited ice formation and efficient de-icing on super-hydrophobic micro-structured airfoils through direct laser interference patterning," *Adv. Mater. Inter.*, vol. 7, no. 22, 2020, Paper no. 2001231.
- [9] J. Dai, H. Li, Y. Zhang, and H. Chen, "Optimization of multi-element airfoil settings considering ice accretion effect," *Chin. J. Aeronaut.*, vol. 36, no. 2, pp. 41–57, 2023.
- [10] O. Savchenko, V. Zhelykh, Y. Yurkevych, S. Shapoval, and K. Kozak, "Using vortex tube for decreasing losses of natural gas in engineering system of gas supply," *Pollack Period.*, vol. 13, no. 3, pp. 241–250, 2018.
- [11] S. Eftekhari and A. S. M. Al-Obaidi, "Investigation of a NACA0012 finite wing aerodynamics at low Reynold's numbers and 0° to 90° angle of attack," *J. Aerospace. Technol. Manage.*, vol. 11, 2019, Paper no. e1519.
- [12] O. Zikanov, *Essential Computational Fluid Dynamics*. Wiley & Sons, Inc, 2019.
- [13] Z. Malikov, "Mathematical model of turbulence based on the dynamics of two fluids," *Appl. Math. Model.*, vol. 82, pp. 409–436, 2020.
- [14] F. J. K. Ideriah, *Review of Equation Solved in TEACH*, Private communication, 1975.
- [15] ANSYS, Inc, ANSYS Gambit [Computer Software], 2020. Canonsburg, PA, USA: ANSYS, Inc.
- [16] ANSYS Fluent, User's Guide, Version 6.3.26, 2009.
- [17] G. Recktenwald, *The SIMPLE Algorithm for Pressure-Velocity Coupling*, Department of Mechanical Engineering, Portland State University, 2012. Notes: ME 448/548.
- [18] N. Shelil, "Flashback studies with premixed swirl combustion," PhD Thesis, Cardiff University, 2009.
- [19] J. D. Anderson, *Fundamentals of Aerodynamics*. 6th ed., New York: McGraw-Hill Inc., 2017.

