Министерство цифровых технологий Республики Узбекистан

НАУЧНО-ИССЛЕДОВАТЕЛЬСКИЙ ИНСТИТУТ РАЗВИТИЯ ЦИФРОВЫХ ТЕХНОЛОГИЙ И ИСКУССТВЕННОГО ИНТЕЛЛЕКТА

БУХАРСКИЙ ГОСУДАРСТВЕННЫЙ УНИВЕРСИТЕТ

СОВРЕМЕННОЕ СОСТОЯНИЕ И ПЕРСПЕКТИВЫ РАЗВИТИЯ ЦИФРОВЫХ ТЕХНОЛОГИЙ И ИСКУССТВЕННОГО ИНТЕЛЛЕКТА

Сборник докладов международной научно-технической конференции

Бухара, 27-28 сентября 2024 г.

ЧАСТЬ 2

Ташкент – 2024

УДК 519.6+519.7:004.8
ББК 22.18
С 56

Современное состояние и перспективы развития цифровых технологий и искусственного интеллекта : сборник докладов международной научно-технической конференции, г. Бухара, 27-28 сентября 2024 г. : в 2 ч. Ч. 2 / Научно-исследовательский институт развития цифровых технологий и искусственного интеллекта. – Ташкент: изд-во НИИ РЦТИИ, 2024. – 445 с.

В настоящем сборнике вниманию читателей представлены доклады, отражающие результаты научных изысканий по направлениям интеллектуального анализа данных, обработки текстов, изображений и речевых сигналов, машинного обучения и представления знаний, развития цифровых технологий, математического моделирования, алгоритмизации, управления и оптимизации в технических, экономических и социальных системах, а также вопросы педагогики и образования в условиях цифровой трансформации. Авторами докладов рассматривается широкий спектр проблем развития технологий искусственного интеллекта. Освещаются вопросы применения цифровых технологий, анализа данных и машинного обучения при решении задач моделирования и управления объектами различной природы. Значительное место занимают исследования, связанные с теоретическими и прикладными вопросами разработки математических моделей, вычислительных алгоритмов и программного обеспечения. Приводятся результаты разработок моделей и программно-технических средств систем информационной безопасности, прикладных систем обработки информации, принятия решений и управления в нефтегазовой, машиностроительной, агропромышленной, телекоммуникационной, финансово-экономической отраслях и в образовании. Содержание сборника ориентировано на научных работников, докторантов, инженерно-технических работников, преподавателей ВУЗов, осуществляющих исследования, прикладные разработки, внедрение и эксплуатацию информационно-коммуникационных технологий, а также подготовку специалистов по соответствующим направлениям.

UDC 519.6

NUMERICAL STUDY OF THE FLOW AWAY AFTER BELL 540 AERODYNAMIC PROFILE BASED ON *THE K- ε* TURBULENCE MODEL

[1,2*]Hamdamov M.M., [2]Jumaev J., [1]Ergashev D.Y., [1]Muzaffarov S.A.

*mmhamdamov@mail.ru

[1]Institute of Mechanics and Seismic Resistance of Structures of the Academy of Sciences of the Republic of Uzbekistan, Tashkent, Uzbekistan

[2]Bukhara State University, Bukhara, Uzbekistan

[3]Oriental university, Tashkent, Uzbekistan

Annotation. Currently, the study of turbulent flows formed around aerodynamic profiles is one of the most important issues in the field of aircraft engineering and wind power. It is impossible or too expensive to study these currents by experiment. Therefore, it is appropriate to model it using modern package programs. This article presents a study of a turbulence model in subsonic flow around a BELL 540 airfoil with angles of attack from 0 to 12 degrees. For the numerical implementation of the turbulence equations, the finite element method was used, implemented using the software package Comsol Multiphysics. The k-e model was used to determine the turbulence. The results obtained were compared with experimental data and showed good agreement between them, which confirms the adequacy of the proposed turbulence model. The main aspects of the research methodology are discussed, including modeling parameters and analysis of the data obtained. This study contributes to the understanding of turbulent flow around airfoils and may be useful for developing more accurate engineering models.

Keywords: Navier–Stokes equations, separated flow, model $k-\varepsilon$, Comsol Multiphysics, BELL 540 airfoil, NACA, aerodynamic profiles.

1 Introduction

Turbulent flows are complex phenomena in the fields of aerodynamics and fluid dynamics that result from nonlinear interactions between particles of a liquid or gas. These interactions produce chaotic and unpredictable movements, making turbulence one of the most challenging objects to study in flow physics. Understanding turbulent flows has important implications for various engineering fields. In aerodynamics, for example, turbulence affects the aerodynamic characteristics of airplanes and other aircraft, as well as the efficiency and safety of their flights. In hydrodynamics, turbulent flows determine the behavior of water in rivers, oceans and pipelines, which is important for the design of hydraulic structures and water supply systems. In addition, turbulence plays a key role in the design and optimization of various mechanisms and machines such as turbomachines, pumps and fans. The study of turbulent flows is a current research task, as it contributes to the development of more accurate and efficient engineering models. However, despite significant advances in this field, many aspects of turbulent flows remain poorly understood, which creates a need for further research and development. Thus, understanding turbulent flows is of fundamental importance for various engineering applications and is the subject of active research nowadays.

In this study, we focus on the numerical simulation of turbulent flow around the BELL 540 airfoil. This airfoil is widely used in aerodynamic research due to its simplicity and good lift performance. We will study the flow around the airfoil at various angles of attack, ranging from 0 to 20 degrees. For numerical modeling we use the Comsol software package Multiphysics , which provides extensive capabilities for solving a variety of problems in continuum mechanics, including modeling turbulent flows.

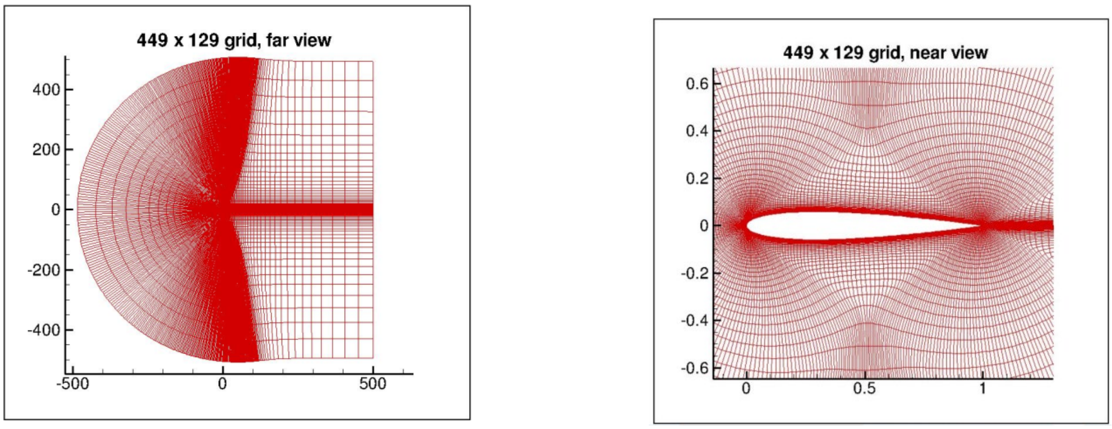
Aerodynamics research is an important component of wind turbine design and optimization . BELL 540 is one of the most common airfoils. Powerful techniques such as computational fluid dynamics (CFD) can be used to study flow around an airfoil and determine its aerodynamic characteristics.

CFD research allows you to conduct virtual experiments by simulating flow around an airfoil under different conditions. To estimate aerodynamic parameters, various approaches are used, such as models $k-\varepsilon$ and $k-\omega$, to solve the Navier-Stokes equations and turbulence equations.

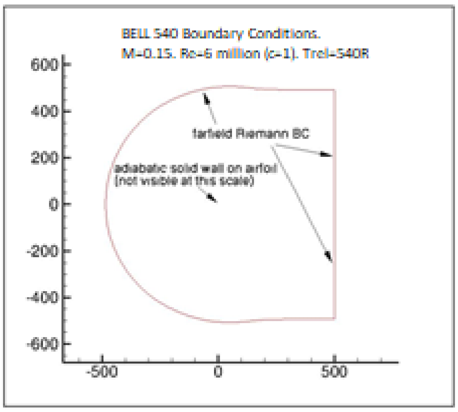
The main goal of this study is to verify the adequacy of the proposed turbulence model by comparing the obtained numerical data with the results of experimental measurements. Successful comparison of these results will confirm the applicability of our model to real engineering problems and increase the level of confidence in numerical methods in aerodynamics. Later in the article we will describe in detail the research methodology, present the results obtained and discuss their significance for practical applications.

2 Physical and mathematical formulation of the problem

The BELL 540 turbulent airfoil must be operated in virtually incompressible conditions. Reynolds number per chord Re = 5 million. In Fig. 1 shows the computational mesh and boundary conditions [13].



a)



b)

Figure 1. 2D Profile BELL 540. a) computational mesh b) boundary conditions.

2.1 Mathematical model

The BELL 540 airfoil, Reynolds -averaged Navier-Stokes (RANS) equations were used. These equations underlie the mathematical description of the dynamics of an incompressible fluid and represent a system of differential equations that model changes in speed and pressure in a liquid medium in time and space.

Navier-Stokes equations in averaged form take into account turbulent flows and represent the following system of equations:

Mass conservation equation (continuity equation), which describes the law of conservation of mass within the computational domain:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

The momentum conservation equation, which describes the change in fluid velocity under the influence of external and internal forces:

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} \tag{2}$$

Where

$\overline{u}_i$ - components of the average velocity field, $\overline{p}$ - average pressure, ν - kinematic viscosity, τ_{ij} - components of the stress tensor, ρ - density.

Reynolds -averaged Navier-Stokes equations makes it possible to take into account turbulent effects and their influence on the flow around the BELL 540 airfoil . These equations are solved by numerical methods, such as finite element analysis, using specialized software packages such as COMSOL Multiphysics. This approach provides detailed data on the flow characteristics and its impact on the profile.

The study of a turbulence model $k-\varepsilon$ for problems of turbulent flow in a flow around the BELL 540 airfoil is the purpose of this article. The obtained numerical data are compared with known experimental data available on the NACA Turbulence website Modeling Resource (TMR) [12-15].

2.2. Turbulence models

When the chain of equations for correlations of turbulent quantities breaks down into equations for first-order correlations, it is obvious that the turbulence model of the first level of closure $k-\varepsilon$ is the most effective. The k-epsilon (k-ε) turbulence model is one of the most popular and widely used in engineering calculations, especially for general turbulent flow modeling problems. It includes equations for the turbulent kinetic energy (k) and the dissipation rate (ε), which allows describing turbulence on large scales.

The main advantages of the k-ε model are:

Simplicity and reliability: The k-ε model is easy to use and fairly stable, making it suitable for most engineering applications.

Broad range of application: The model copes well with a variety of problems, from modeling turbulent flows in pipes to more complex aerodynamic problems.

Low computational cost: Compared to more complex models such as SST, the k-ε model requires fewer computational resources.

This model is currently very popular and is included in many CFD packages[16-20].

$$\begin{cases} \frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - 2\rho\varepsilon M_t^2 + S_k \\ \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}G_b + S\varepsilon \end{cases} \tag{3}$$

The notation used here is

$$C_1 = \max\left[0.43, \frac{\eta}{\eta + 5}\right], \quad \eta = S\frac{k}{\varepsilon}, \quad S = \sqrt{2S_{ij}S_{ij}}, \quad \mu_t = \rho C_\mu \frac{k^2}{\varepsilon}, \quad C_\mu = \frac{1}{A_0 + A_S \frac{kU^*}{\varepsilon}}, \quad U^* \equiv \sqrt{S_{ij}S_{ij} + \tilde{\Omega}_{ij}\tilde{\Omega}_{ij}},$$

$$\Omega_{ij} = \overline{\Omega}_{ij} - 2\varepsilon_{ijk}\omega_k, \quad A_S = \sqrt{6}\cos\varphi, \quad \varphi = \frac{1}{3}\cos^{-1}(\sqrt{6}W), \quad W = \frac{S_{ij}S_{jk}S_{ki}}{\tilde{S}^3}, \quad \tilde{S} = \sqrt{S_{ij}S_{ij}}, \quad S_{ij} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right),$$

$G_k = -\rho\overline{u_i'u_j'}\dfrac{\partial u_j}{\partial u_i}$, $S \equiv \sqrt{2S_{ij}S_{ij}}$, $G_b = \beta g_i \dfrac{\mu_t \partial T}{\Pr_t \partial x_i}$, $a_0 = 1/\Pr = k/\mu c_p$, $\beta = -\dfrac{1}{\rho}\left(\dfrac{\partial \rho}{\partial T}\right)$, $G_b = -g_i \dfrac{\mu_t}{\rho \Pr_t}\dfrac{\partial \rho}{\partial x_i}$,

$a = \sqrt{\gamma RT}$, $\Pr_t = 1/a_t$, $M_t = \sqrt{\dfrac{k}{a^2}}$.

The empirical constants $k-\varepsilon$ of the model take standard values: $C_{1\varepsilon} = 1.44$ $C_2 = 1.9$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.2$, $A_0 = 4.04$.

2.3 Solution method

For the standard turbulence model $k-\varepsilon$ standard COMSOL Multiphysics solvers were used.

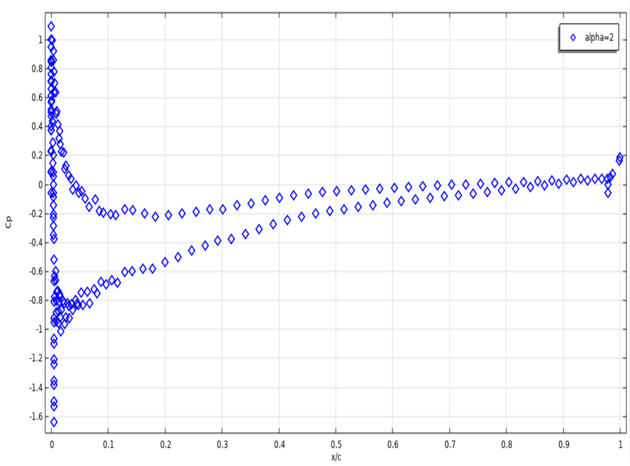
3 Results and discussion

The change in pressure on the channel wall depending on the distance is called the distribution of the surface pressure coefficient [20-24].

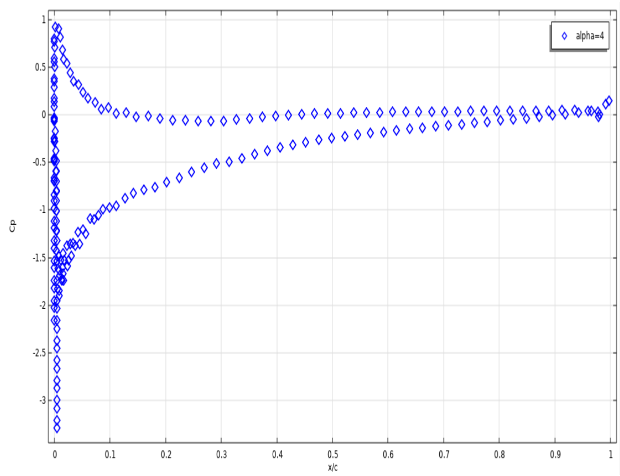
$$C_p = \frac{p - p_\infty}{0.5\rho U_0^2}.$$

where p is the pressure at a point on the surface of the profile, P_∞ is the pressure of the free flow, ρ is the density of the free flow, U_0 is the speed of the free flow.

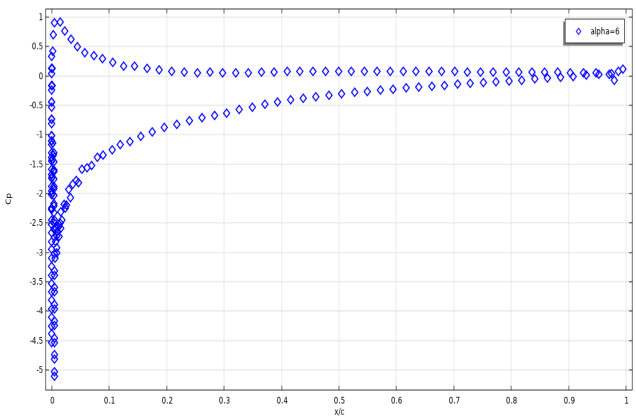
Comparisons of the obtained numerical results with known experimental data are shown below. In Fig. 2 shows the pressure coefficients for various angles of attack of the profile surface.



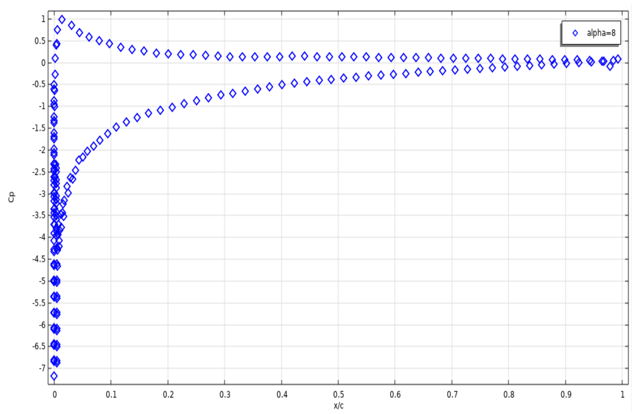
α= 2



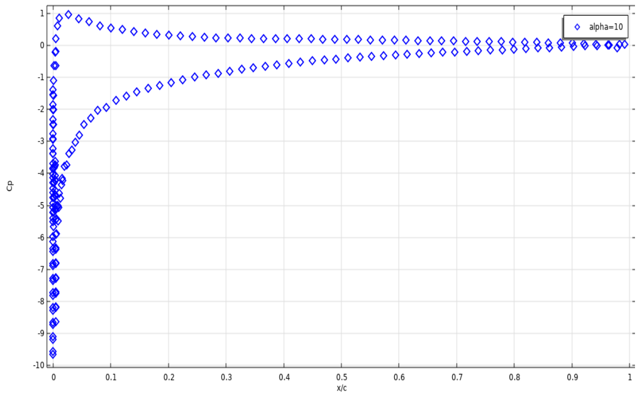
α= 4



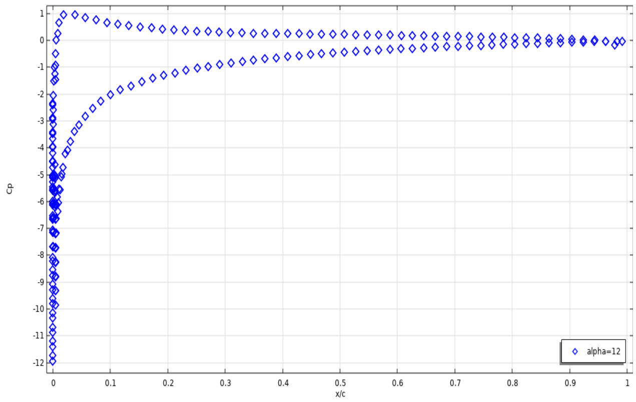
α= 6



α=8



α= 10



α= 12

Figure 2. Pressure coefficient varying depending on the angle of attack of the profile surface

The effect of angles of attack on lift coefficients is shown in Fig. 3.

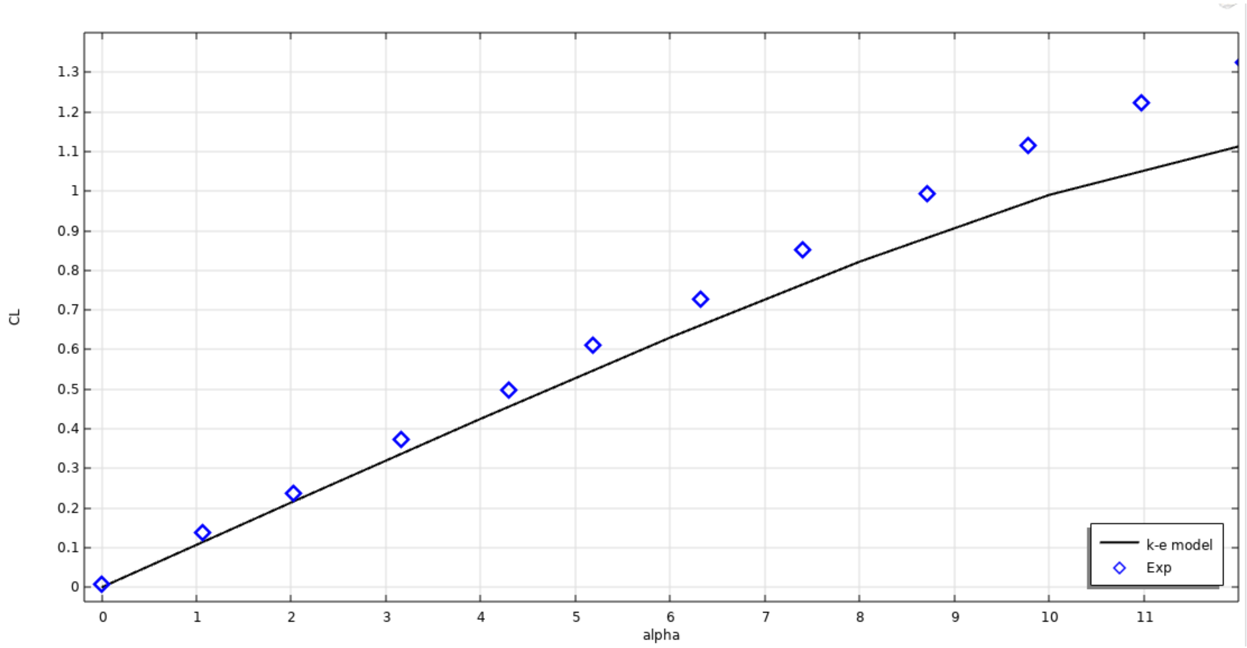
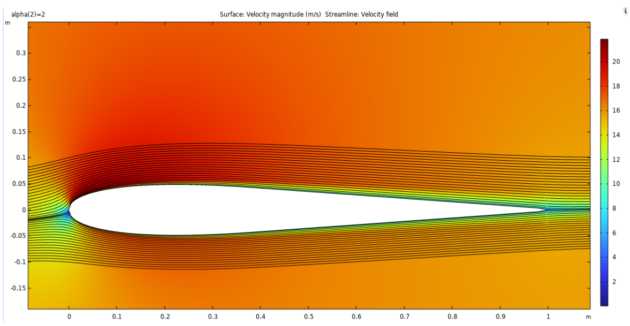
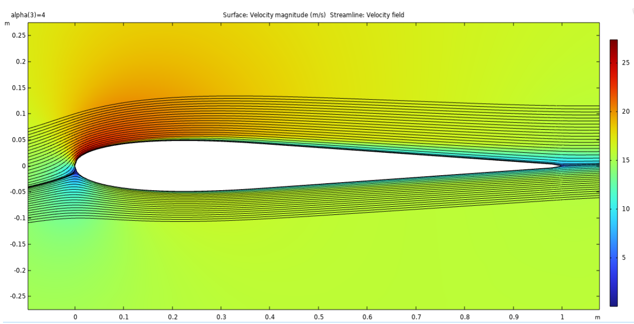


Figure 3. The influence of angles of attack on lift coefficients.

The results of the experiment are similar to the results $k-\varepsilon$ models as shown in Figures 2-3. In Fig. 4 shows isolines of the flow velocity at different angles of attack.



α= 2



α= 4

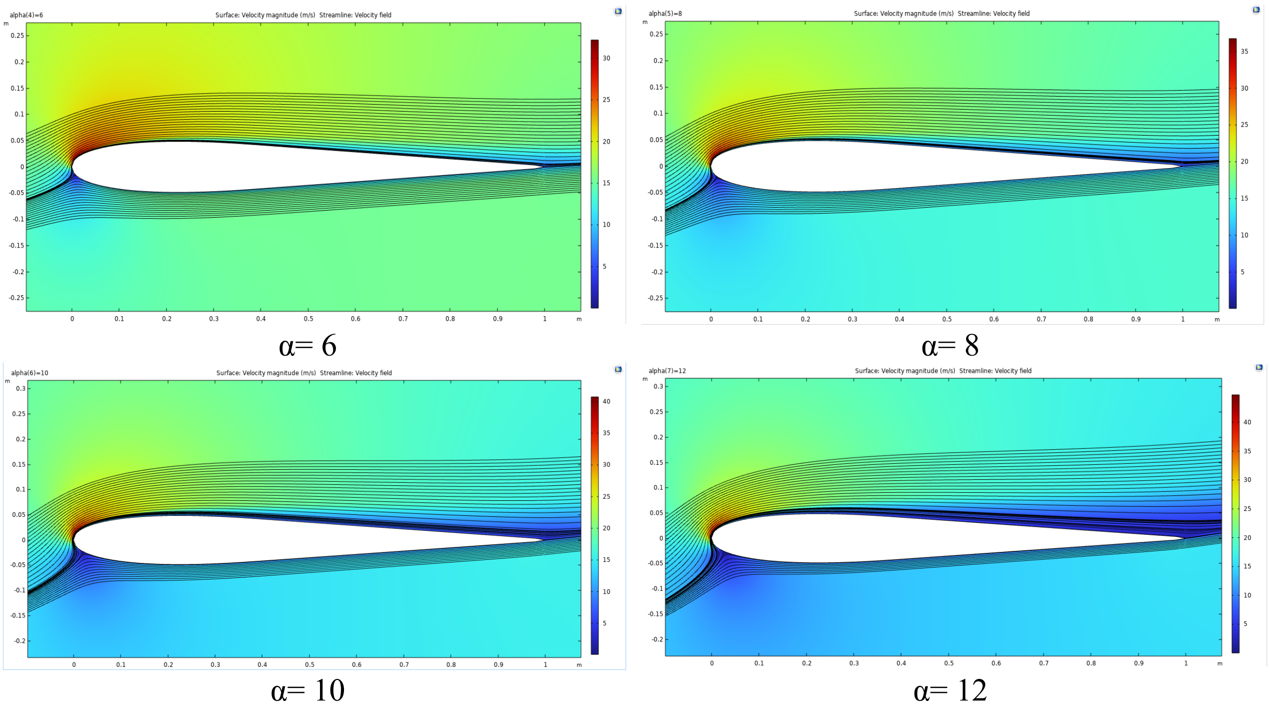


Figure 4. Isolines of flow velocity at different angles of attack.

The turbulence model $k-\varepsilon$ is considered to be the best semi-empirical model available today.

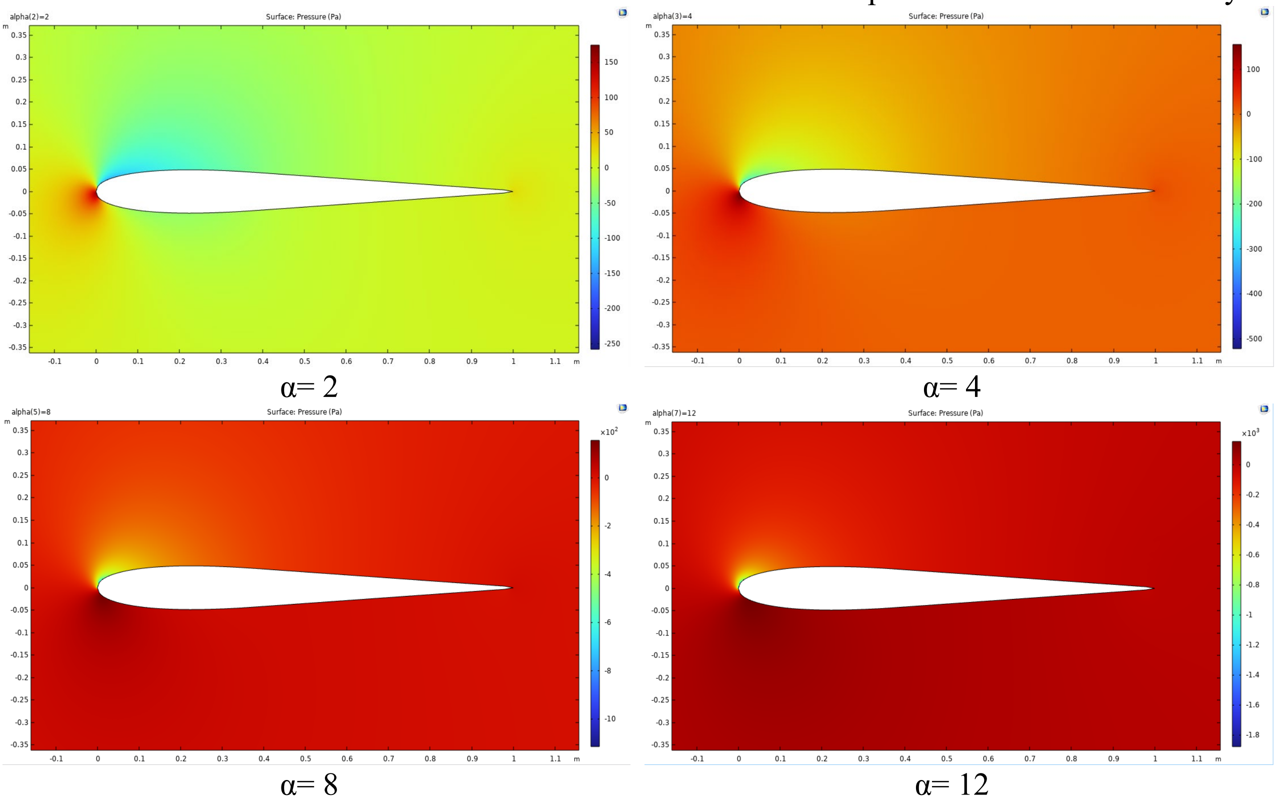


Figure 5. Isolines of the pressure field at different angles of attack.

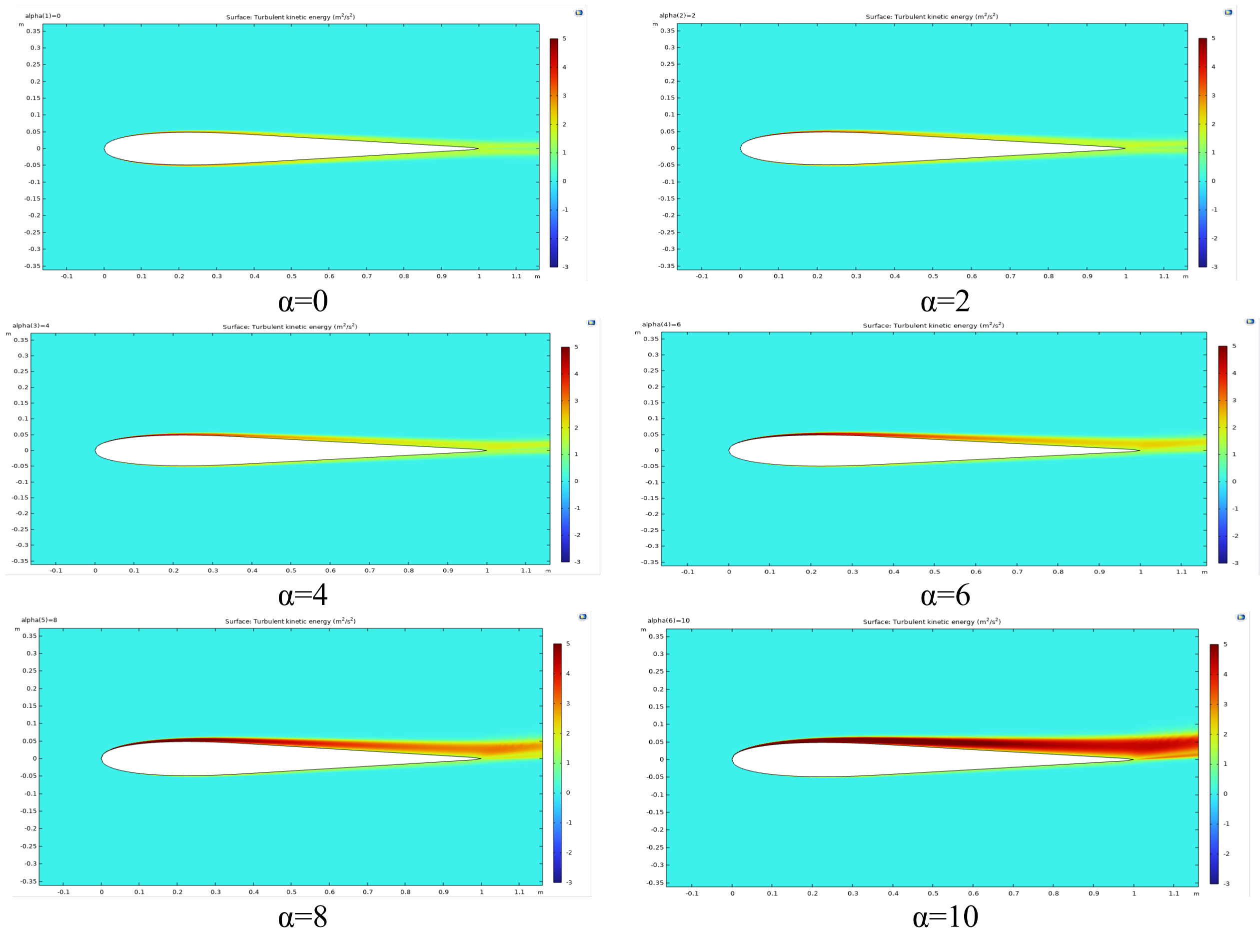


Figure 6. Isolines of the turbulent kinetic energy field at different angles.

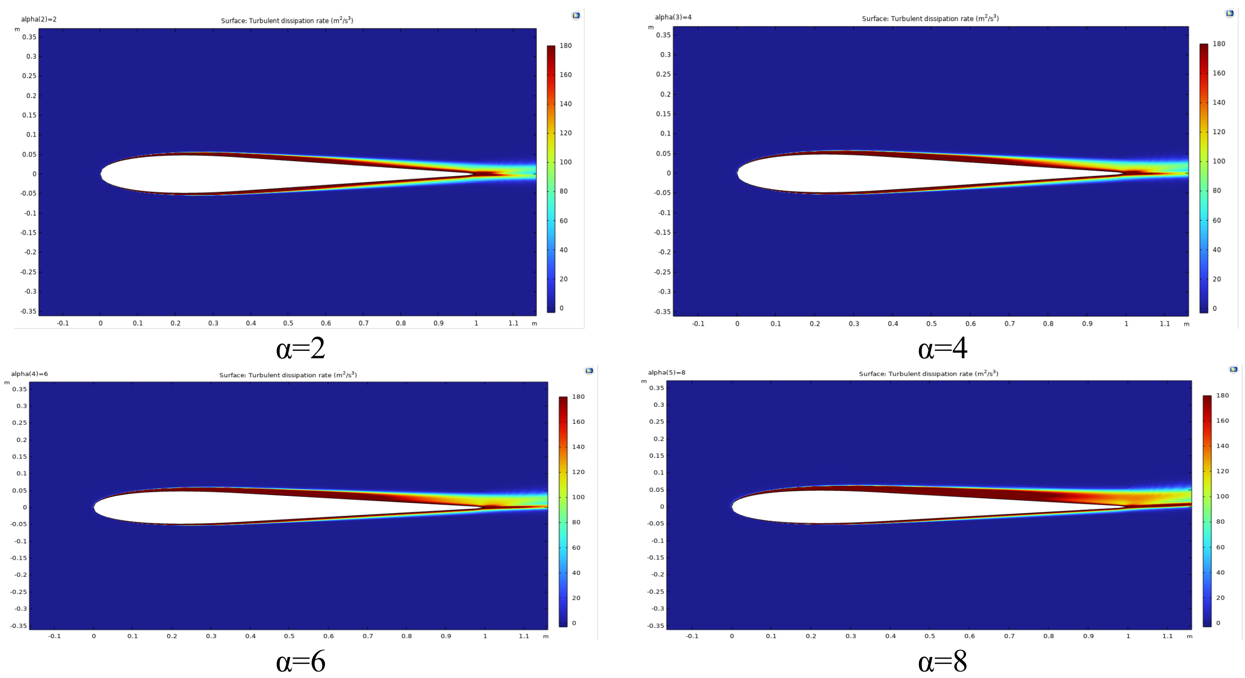


Figure 7. Isolines of the turbulent dissipation field at different angles

4 Conclusion

Studying the flow around the BELL 540 airfoil using CFD techniques provides useful data to engineers and designers. The data can be used to improve aerodynamic efficiency, optimize airfoil shapes and create more efficient wind turbines. Understanding the aerodynamic characteristics of an airfoil at the level of numerical modeling helps in the development of innovative technologies aimed at renewing industrial energy.

The article reviewed shows the results of a standard turbulence model $k-\varepsilon$ in the Comsol software package Multiphysics , which uses the finite element method. To validate the model, $k-\varepsilon$ the problems of flow around the BELL 540 airfoil were considered. From the results obtained, it is clear that $k-\varepsilon$ the model has high accuracy for this problem.

References

[1]. *Orozco Murillo, W., Palacio-Fernande, J. A., Patiño Arcila, I. D., Zapata Monsalve, J. S., & Hincapié Isaza, J. A.* (2020). Analysis of a Jet Pump Performance under Different Primary Nozzle Positions and Inlet Pressures using two Approaches: One Dimensional Analytical Model and Three Dimensional CFD Simulations. Journal of Applied and Computational Mechanics, 6(Special Issue), 1228-1244.

[2] *Hadad, K., Eidi, H. R., & Mokhtari, J.* (2017). VOC level control by ventilation improvement of Flexography printing room using CFD modeling. Journal of Applied and Computational Mechanics, 3(3), 171-177.

[3] *Tsega, E. G., & Katiyar, V. K.* (2019). A Numerical Simulation of Inspiratory Airflow in Human Airways during Exercise at Sea Level and at High Altitude. Journal of Applied and Computational Mechanics, 5(1), 70-76.

[4] *Sentyabov A.V., Gavrilov A.A., Dekterev A.A.* Investigation of turbulence models for computation of swirling flows. Thermophysics and aeromechanics. 18:1, 73-85, 2011.

[5] *Menter F.R.*"Zonal two-equation k- ω turbulence models for aerodynamic flows." AIAAPaper 1993-2906.

[6] *Menter F.R.*, Kuntz M., and Langtry R. "TenYears of Industrial Experience with the SST Turbulence Model." Turbulence, Heat and Mass Transfer 4, ed: K. Hanjalic, Y. Nagano, and M. Tummers, Begell House, Inc., 2003, pp. 625 - 632.

[7] *Pasha A. A.* (2018). Study of parameters affecting separation bubble size in high speed flows using k-ω turbulence model. Journal of Applied and Computational Mechanics, 4(2), 95-104.

[8] *Malikov, Z. M., & Madaliev, M. E.* (2021). Numerical study of a swirling turbulent flow through a channel with an abrubt expansion. Vestnik Tomskogo Gosudarstvennogo Universiteta. Matematika i Mekhanika, (72), 93-101.

[9] *Malikov, Z. M., & Madaliev, M. E.* (2021). Mathematical modeling of a turbulent flow in a centrifugal separator. Vestnik Tomskogo Gosudarstvennogo Universiteta. Matematika i Mekhanika, (71), 121-138.

[10] *Erkinjon M.M.* Numerical Calculation of an Air Centrifugal Separator Based on the SARC Turbulence Model, *J. Appl. Comput. Mech* ., 7(2), 2021, x–xx. https://doi.org/10.22055/JACM.2020.31423.1871

[11] *Spalart P.R., Jou W.H., Strelets M., and Allmaras S.R.*, "Comments on the Feasibility of LES for Wings and on a Hybrid, RANS/LES Approach," Advances in DNS/LES, Proceedings of 1st AFOSR International Conference on DNS/LES, Vol. 1, Greyden Press, Columbus, 1997, pp. 137–147

[12]"Turbulence modeling Resource. NASA Langley Research Center," http://turbmodels.larc.nasa.gov .

[13] *Ladson C.L.*, "Effects of Independent Variation of Mach and Reynolds Numbers on the Low-Speed Aerodynamic Characteristics of the NACA 0012 Airfoil Section," NASA TM 4074, October 1988.

[14] *Khujaev I. , Jumayev J. , Hamdamov M.* Modeling of Combustion Processes in Cylindrical Chambers Using Modern Package Programs. AIP Conference Proceedings, 2024, 3004(1), 060015

[15] *Khujaev I. , Toirov O. , Jumayev J. , Hamdamov M.* Modeling of vertical axis wind turbine using Ansys Fluent package program. E3S Web of Conferences, 2023, 401, 04040

[16] *Hamdamov M. , Bozorov B. , Mamataliyeva H. , Ergashov D.* Numerical modeling of wind turbine with vertical axis using turbulence model k - ω in ANSYS FLUENT. E3S Web of Conferences, 2023, 401, 02024

[17] *Hamdamov MM , Ishnazarov AI , Mamadaliev K.A.* Numerical Modeling of Vertical Axis Wind Turbines Using ANSYS Fluent Software. Lecture Notes in Computer Science (including subseries Lecture Notes in Artificial Intelligence and Lecture Notes in Bioinformatics), 2023, 13772 LNCS, pages 156–170

[18] *Hamdamov M. , Bozorov B. , Mamataliyeva H. , Ergashov D.* Numerical modeling of wind turbine with vertical axis using turbulence model k - ω in ANSYS FLUENT . E3S Web of Conferences, 2023, 401, 02024

[19] *Mathew S.*, "Wind energy: fundamentals, resource analysis and economics," New York, Springer-Verlag Berlin Heidelberg, 2006.

[20] *Goudarzi A., Ahmadi A.* "Intelligent Analysis of Wind Turbine Power Curve Models," In Computational Intelligence Applications in Smart Grid (CIASG), IEEE, pp. 1-7, 2014.

[21] *Carrillo C., Montan A., Cidras J., Dıaz-Dorado E.* "Review of powercurve modeling for wind turbines," Renewable and SustainableEnergy Reviews, vol. 21, pp. 572–581, 2013.

[22] *Sohoni V., Gupta S., and Nema R.* "A Critical Review on WindTurbine Power Curve Modeling Techniques and Their Applications in Wind Based Energy Systems," Journal of energy, pp.1-18, 2016.

[23] *Alhassan A.* Teyabeen, "Statistical Analysis of Wind Speed Data," 6th International Renewable Energy Congress (IREC), pp. 1-6. IEEE2015.

[24] *Alhassan A.* Teyabeen, "Selection of appropriate statistical model for wind speed data, and selecting suitable turbine generator at four locations in Libya." [Master's thesis]; Dept. of EE; University of Tripoli, Libya, 2017.