A Numerical Study of Turbulent Flows Around a NASA 0012 Profile for Wind Turbines Using the K E Model

Muzaffar Hamdamov
Institute of Mechanics and Seismic Resistance of Structures
Academy of Sciences of the Republic of Uzbekistan
Tashkent, Uzbekistan
mmhamdamov@mail.ru

Rabim Fayziev
Department of Econometrics
Tashkent State University of Economics
Tashkent, Uzbekistan
r.fayziyev@tsue.uz

Khayotjon Aminov
Department of Econometrics
Oriental university
Tashkent, Uzbekistan
1muzaffarovs9@gail.com

Sardorbek Muzaffarov
Institute of Mechanics and Seismic Resistance of Structures
Academy of Sciences of the Republic of Uzbekistan
Tashkent, Uzbekistan
muzaffarovs9@gail.com

Abstract

Currently, the development of renewable energy sources is one of the most important issues. An environmentally friendly and cheap energy is wind energy. Therefore, it is necessary to study turbulent flows around aerodynamically shaped profiles when creating new models of wind turbines. 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 NACA0012 airfoil with angles of attack from 0 to 20 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, Comsol Multiphysics, NACA 0012, renewable energy

ACM Reference Format:
Muzaffar Hamdamov, Rabim Fayziev, Khayotjon Aminov, and Sardorbek Muzaffarov. 2024. A Numerical Study of Turbulent Flows Around a NASA 0012 Profile for Wind Turbines Using the K E Model. In *The 8th International Conference on Future Networks & Distributed Systems (ICFNDS '24), December 11, 12, 2024, Marakech, Morocco.* ACM, New York, NY, USA, 6 pages. https://doi.org/10.1145/3726122.3726151

1 INTRODUCTION

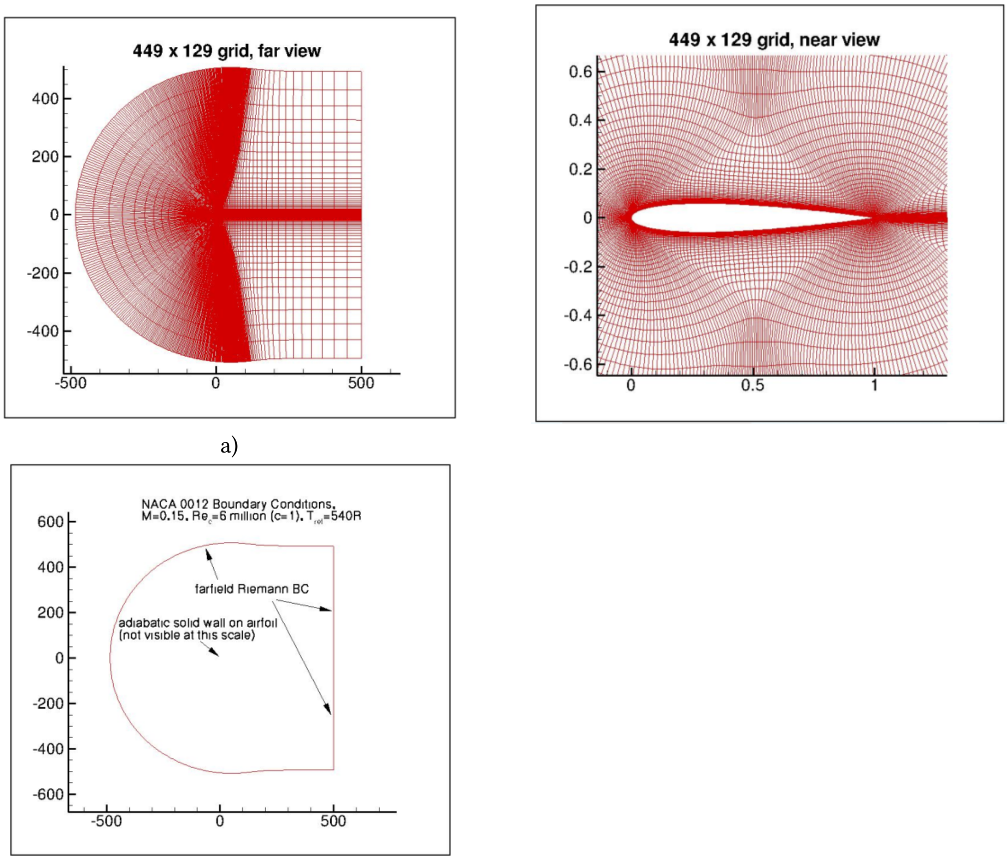
As shown in the works [1-3], the existing energy supply of the Republic of Uzbekistan does not meet the needs. The President of the Republic of Uzbekistan has set the task of doubling the volume of energy supply of the country by 2030. In this regard, the works [1-3] propose ways to implement them. One of the ways to solve the problem is to increase the production of energy supply using renewable sources.

One of the most important issues in the development of renewable energy sources is the creation of efficient wind turbines. Because, at present, until 2030, the issue of providing a significant part of energy with renewable energy is set. Therefore, it is appropriate to continue the research in this field and study the movements of the parts that are important for wind turbines. Turbulent flows are formed in renewable energy installations.

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 [8, 9]. 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

ICFNDS '24, Marakech, Morocco

ACM ISBN 979-8-4007-1170-1/2024/12
https://doi.org/10.1145/3726122.3726151



a)

b)

Figure 1: 2D Profile NACA 0012. a) computational mesh b) boundary conditions [13].

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 [10-13].

In this study, we focus on the numerical simulation of turbulent flow around the NACA0012 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. NACA 0012 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 [11-14].

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 and , to solve the Navier-Stokes equations and turbulence equations [7, 14-17].

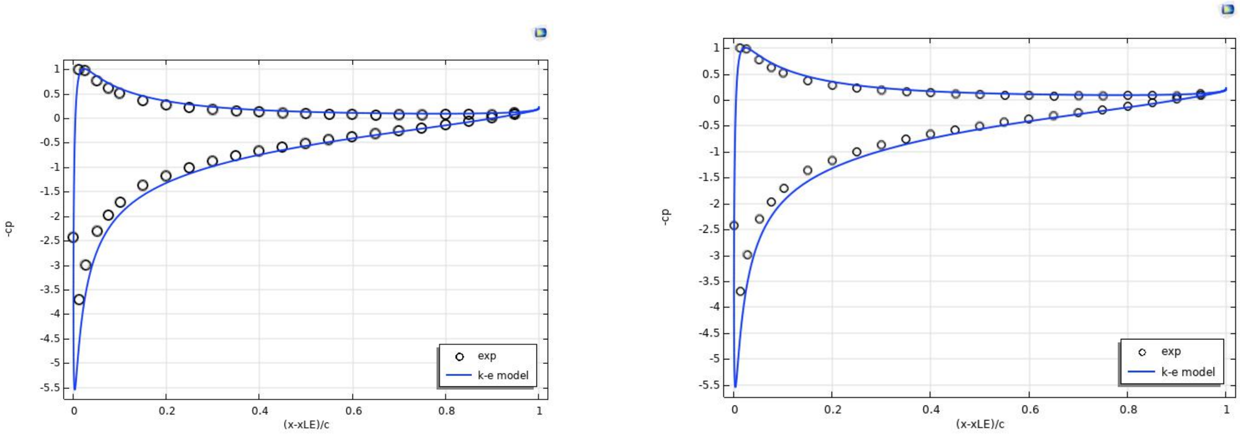
The main goal of this study is to verify the adequacy of the proposed turbulence model by comparing the obtained numerical data with the results of experimental measurements. Successful comparison of these results will confirm the applicability of our model to real engineering problems and increase the 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..

1.1 Physical and mathematical formulation of the problem

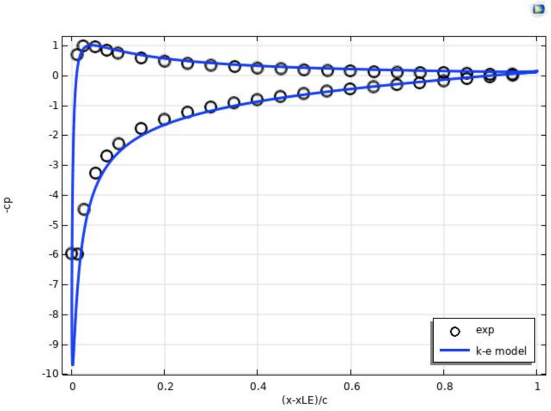
The NACA 0012 turbulent airfoil must be operated in virtually incompressible conditions. Reynolds number per chord Re = 5 million. In Fig. 1 shows the computational mesh and boundary conditions [17-20].

2 Mathematical model

The Reynolds-averaged Navier-Stokes (RANS) equations and the NACA 0012 profiles are employed. These differential equations, which characterize changes in pressure and velocity in a liquid medium across time and space, serve as the foundation for the mathematical description of the dynamics of an incompressible



$\alpha = 5$ $\alpha = 10$



$\alpha = 15$

Figure 2: Pressure coefficient varying depending on the angle of attack of the profile surface [13]

Fluid. 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 [4-7]:

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

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

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

Where

$\bar{u}_i$- components of the average velocity field, $\bar{p}$- 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 NACA 0012 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 NACA 0012 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) [17, 18].

3 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-e model, also known as k-epsilon, is a two-differential equation model that describes the turbulent kinetic energy (k) and its dissipation rate (ϵ). Because this model can simulate turbulent flows with sufficient accuracy at relatively low computational costs, it has proven to be a reliable tool for solving many engineering problems.

The system consists of two nonlinear diffusion equations that describe the turbulent energy mass density k and the turbulent energy dissipation rate ϵ. This model was proposed by Launder

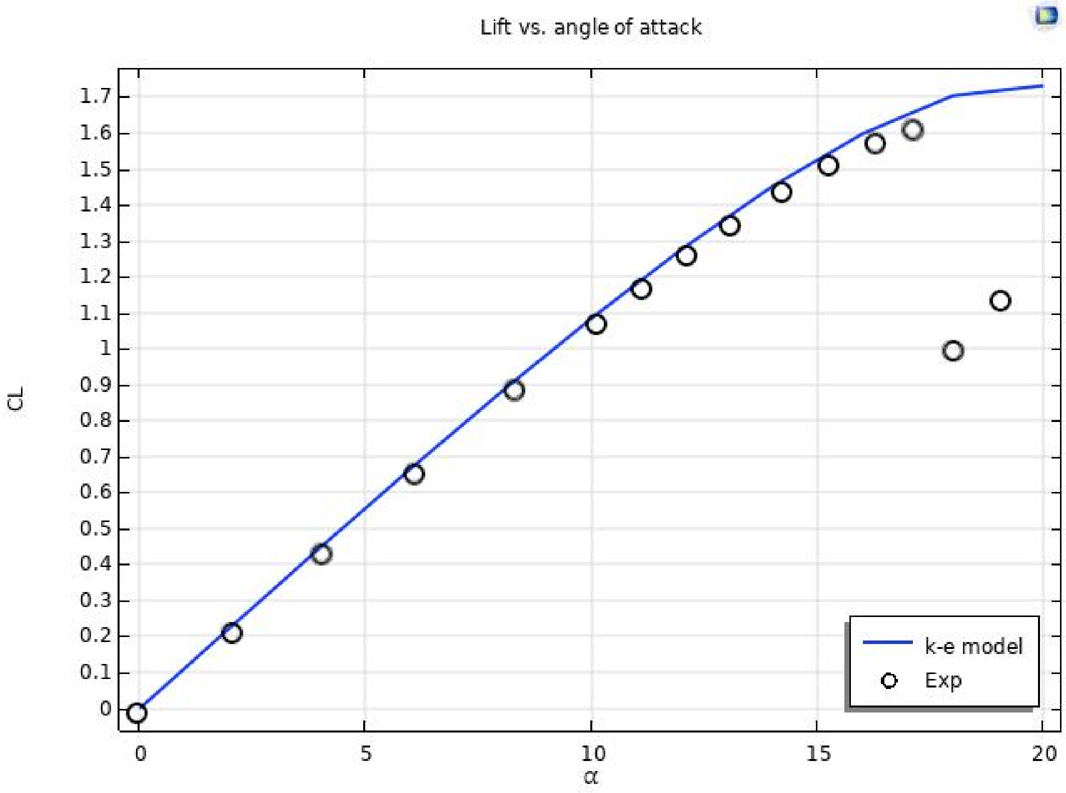


Figure 3: The influence of angles of attack on lift coefficients [13].

and Spalding more than 30 years ago and has become the basis for many engineering calculations.

These nonlinear diffusion equations serve as the basis for modeling turbulent flows in many applications. This model has become available and widely used in various engineering problems due to the simplification proposed by Launder and Spalding.This model is currently very popular and is included in many CFD packages[19, 20].

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

The notation used here is $C_1 = \max[0.43, \frac{\eta}{\eta+5}]$, $\eta = S\frac{k}{\varepsilon}$, $S = \sqrt{2S_{ij}S_{ij}}$, $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$, $C_\mu = \frac{1}{A_0+A_S\frac{kU^*}{\varepsilon}}$, $U^* \equiv \sqrt{S_{ij}S_{ij} + \tilde{\Omega}_{ij}\tilde{\Omega}_{ij}}$, $\Omega_{ij} = \bar{\Omega}_{ij} - 2\varepsilon_{ijk}\omega_k$, $A_S = \sqrt{6}\cos\phi$, $\phi = \frac{1}{3}\cos^{-1}(\sqrt{6}W)$, $W = \frac{S_{ij}S_{jk}S_{ki}}{\tilde{S}^3}$, $\tilde{S} = \sqrt{S_{ij}S_{ij}}$, $S_{ij} = \frac{1}{2}(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j})$, $G_k = -\rho\overline{u'_i u'_j}\frac{\partial u_j}{\partial u_i}$, $S \equiv \sqrt{2S_{ij}S_{ij}}$, $G_b = \beta g_i \frac{\mu_t \partial T}{\mathrm{Pr}_t \partial x_i}$, $a_0 = 1/\mathrm{Pr} = k/\mu c_p$, $\beta = -\frac{1}{\rho}(\frac{\partial \rho}{\partial T})$, $G_b = -g_i \frac{\mu_t}{\rho \mathrm{Pr}_t}\frac{\partial \rho}{\partial x_i}$,$a = \sqrt{\gamma RT}$, $\mathrm{Pr}_t = 1/a_t$, $M_t = \sqrt{\frac{k}{a^2}}$.

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

4 Solution method

For the standard turbulence model $k - \varepsilon$ standard COMSOL Multiphysics solvers were used [19-23].

5 RESULTS AND DISCUSSION

The change in pressure on the channel wall depending on the distance is called the distribution of the surface pressure coefficient [21].

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

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.

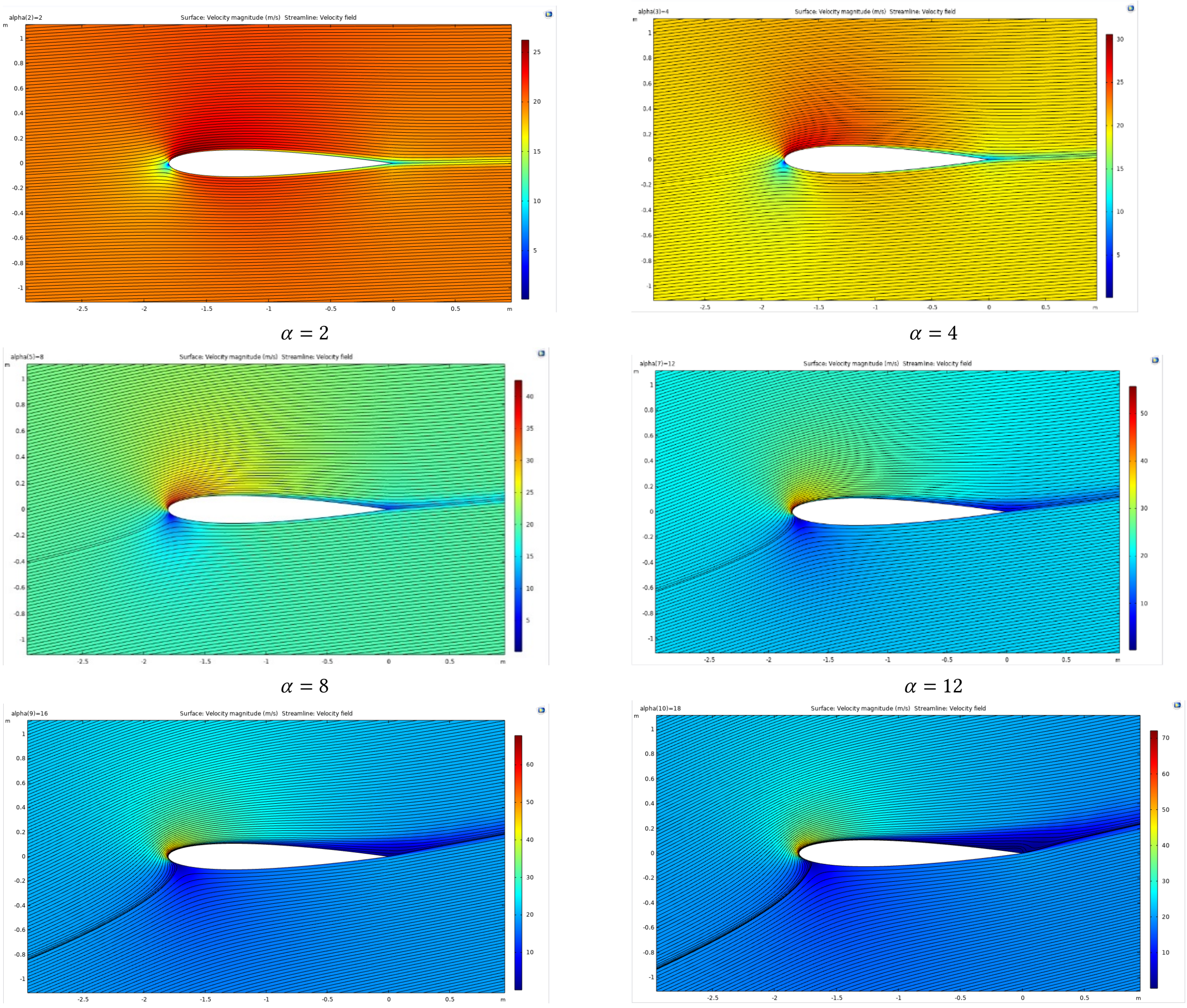
Comparisons of the obtained numerical results with known experimental data are shown below. In Fig. 2 shows the pressure coefficients and experimental results for various angles of attack of the profile surface.

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

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.

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



$\alpha = 2$ $\alpha = 4$

$\alpha = 8$ $\alpha = 12$

$\alpha = 16$ $\alpha = 18$

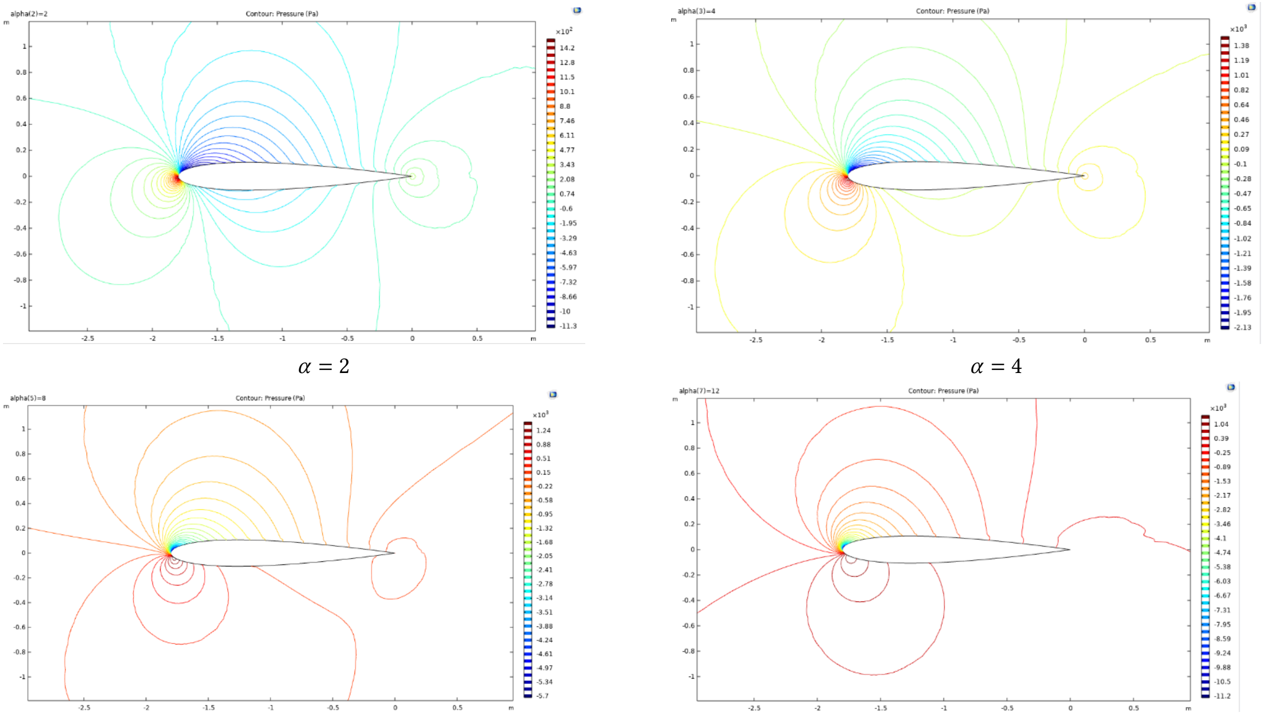
Figure 4: Isolines of flow velocity at different angles of attack.

6 CONCLUSION

More than 80 percent of electricity produced in our country is obtained by burning gas. But these gases can expire with time and release large amounts of CO toxic gases into the environment. Therefore, we should switch to a clean source of electricity as much as possible. Nowadays, the demand for renewable energy sources is increasing day by day. Therefore, wind energy is a clean and cheap source of renewable energy. The movement of aerodynamic wings is very important in converting the mechanical energy of the wind into electrical energy. Therefore, this paper investigates the turbulent flow around the NACA 0012 airfoil, which is widely used in wind turbines. Results were obtained for lift force and power coefficient at different wind speeds. These results were compared with the results of experiments at the NACA base.

Studying the flow around the NACA 0012 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 NACA 0012 airfoil were considered. From the results obtained, it is clear that $k-\varepsilon$the model has high accuracy for this problem.



$\alpha = 2$ $\alpha = 4$

$\alpha = 8$ $\alpha = 12$

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

References

[1] Fayziev R.A., Kurbanov F.M. Modeling and Forecasting of Net In-come from the Country's Electricity Supply. ACM International Conference Proceeding Series, 2021, pp. 407–412.

[2] Fayziev R.A., Kurbanov F.M. Mathematical modeling and forecasting of electricity production in enterprises of the energy system of Uzbekistan AIP Conference Proceedings 29 August 2022; 2656 (1): 020015.

[3] <number>[3]</number Rabim Fayziev, Feruz Kurbanov, Akmal Mirzoev. AmathematicalmodeltoattractinvestmentsforpowersupplyintheRepublicofUzbekistan. E3S Web of Conferences 541, 02002 (2024) WFCES 2024.

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

[5] Law of the Republic of Uzbekistan. On the farm. — Tashkent, 1.04.2021. lex.uz/docs/5351489

[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] <number>[7]</number 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] <number>[10]</numberhttps://turbmodels.larc.nasa.gov/naca0012_val.html .

[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] 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.

[13] 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.

[14] 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.

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

[16] 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.

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

[18] 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.

[19] HamdamovM. , BozorovB., MamataliyevaH., ErgashovD. Numerical modeling of wind turbine with vertical axis using turbulence model k - ω in ANSYS FLUENT. E3S Web of Conferences, 2023, 401, 02024.

[20] Hamdamov,M.M, IshnazarovA.I, MamadalievK.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.

[21] 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