Journal of Construction and Engineering Technology

# NUMERICAL STUDY OF FLOW AFTER NASA 0012 AERODYNAMIC PROFILE BASED ON THE SST TURBULENCE MODEL

**Murodil Madaliev[1,2*], Sergey Chernyshev[3,4], Alexander Ionov[3]**

[1]Institute of Mechanics and Seismic Resistance of Structures of the Academy of Sciences of the Republic of Uzbekistan, Tashkent, Uzbekistan
[2]Fergana polytechnic institute, Fergana, Uzbekistan
[3]Central Aero-Hydrodynamic Institute named after Professor N.E. Zhukovsky, Zhukovsky, Moscow Region, Russia Russia,
[4]Moscow Institute of Physics and Technology (MIPT), Dolgoprudny, Moscow Region, Russia
*Correspondence: m.e.madaliyev@ferpi.uz

## Abstract

The article presents a study of the SST turbulence model in the Comsol Multiphysics software package for the problem of subsonic flow around a NACA0012 profile with angles of attack from 0 to 15 degrees, respectively. In this work, the finite element method is used for the numerical implementation of the turbulence equations. To stabilize the discretized equations, stabilization using the Galerkin least squares method was used. The results obtained are compared with the results of experimental data. The results showed good convergence of the SST turbulence model with experimental data.

**Key words:** Navier–Stokes equations, separated flow, SST model, Comsol Multiphysics, NASA.

## 1. Introduction

Aerodynamic research is an important part of aircraft development and optimization. One widely used airfoil is the NASA 0012. Numerical methods such as computational fluid dynamics (CFD) provide an effective means to study the flow around a given airfoil and determine its aerodynamic characteristics.

Review of numerical methods:

CFD Simulation:

CFD studies allow you to conduct virtual experiments by simulating flow around an airfoil under different conditions.

Solving the Navier-Stokes equations and the turbulence equations using various approaches (for example, k-ε, k-ω models) allows one to estimate aerodynamic parameters.

Profile geometric characteristics:

Analyzing the thickness and camber in different sections of the chord allows us to understand how geometric features affect aerodynamic properties.

Pressure distribution:

Studying the pressure distribution along the airfoil surface helps to identify high and low pressure zones, which is important for understanding the lift force generated.

Lift and drag:

Evaluation of lift and drag at different angles of attack allows us to determine the optimal conditions for using the profile.

Border Effects:

The study of the influence of the proximity of the profile to a solid surface on the aerodynamic characteristics, as well as the effects of the wing tip, is intended to more accurately take into account the conditions of real application.

Computational Fluid Dynamics, or CFD (Computational Fluid Dynamics), is the science of numerically solving systems of equations that describe the movement of liquids and gases. This discipline finds application in modeling a variety of hydrodynamic processes, such as aerodynamic flows around aircraft, turbulent flows, heat transfer, mixing of fluids and many others. The basis of CFD is the Navier-Stokes equations, which describe the conservation of mass, momentum and energy in a liquid or gas.

Direct analytical solution of the Navier-Stokes equations is possible only for simple geometric objects

Published online: 24 June 2023
https://doi.org/10.5281/zenodo.10521424
© FarPI Hitech Engineer

and flows. In most cases, a division of space into a grid and a numerical solution at each node of this grid are required. Numerical CFD methods, such as finite difference, finite element or finite volume methods, can solve the Navier-Stokes equations over the entire domain, taking into account the boundary and initial conditions.

The use of CFD in various engineering fields allows the design and analysis of systems, taking into account the physical laws of the operation of liquids and gases. For example, in the aviation industry, CFD can optimize the shape of a wing or fuselage, reduce drag, and improve aerodynamic efficiency. CFD capabilities have also been successfully applied in the design of automobiles, ships, turbines, air conditioning systems and other areas. Thus, CFD is an important discipline in engineering, enabling more efficient design and analysis of a variety of systems associated with the movement of liquids and gases [1-2]. The Navier-Stokes equations are a system of differential equations that describe the motion of an incompressible fluid:

$$\begin{cases} \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\nabla\mathbf{v} = -\frac{\nabla p}{\rho} + \nu\nabla^2\mathbf{v} + \mathbf{F} \\ \nabla\mathbf{v} = 0 \end{cases} \quad (1)$$

where:

v is the fluid velocity vector,

t - time,

p - pressure,

ϱ - density,

ν - kinematic viscosity,

F - external force acting on the fluid,

∇ is the nabla operator that determines the gradient and divergence of the vector field.

The first equation - the Navier-Stokes equation of motion - describes the change in fluid velocity over time, under the influence of external forces, viscosity and pressure. The second equation - the continuity equation - guarantees the conservation of the mass of the liquid.

The Navier-Stokes equations are nonlinear equations and are difficult to solve. They are used, for example, to simulate fluid flows in various fields of science and technology, such as aerodynamics, hydrodynamics, oceanography, etc. The Navier-Stokes equation is a mathematical model of laminar flow. Very many flows are turbulent, so it is necessary to model the Navier-Stokes equation so that this equation has a turbulent character [5–13].

NASA Airfoil 0012 is an airfoil that was developed by the US National Aerospace Administration (NASA). This profile is widely used in aerodynamic research and engineering calculations. Here, "0012" denotes the profile designation format, where the first digit (0) indicates no angle of attack (zero angle of attack), and the last three digits (012) represent the profile shape characteristics. The main parameters of the NASA 0012 profile include thickness, camber (camber) and distribution of the profile along the chord (the longitudinal line running from tip to tail). In this case, the thickness of the profile increases evenly from the front edge to the rear, and the camber characterizes the bend of the profile. An important property of an airfoil is its ability to produce lift at atmospheric pressure, which is especially important in aviation and aerospace applications. A graphical representation of a NASA 0012 airfoil typically includes characteristics such as thickness and camber distribution along the chord, as well as plots of lift, drag, and moment about the rotation point (usually relative to the leading edge of the airfoil). If you have specific questions about NASA Profile 0012 or if you need any specific data, please clarify your request and I will try to provide more detailed information.

The purpose of this article is to study the SST turbulence model for problems of turbulent flow around the NASA 0012 airfoil. The obtained numerical results are compared with known experimental data, which are presented on the NASA Turbulence Modeling Resource (TMR) website [14].

## 2. SST turbulence model.

Menter's shear stress transfer (SST) model [5-6] is a combination of the k-ε and k-ω models. For the wall layer, k-ω is used, for the outer region - k-ε. This model is currently very popular and is included in many CFD packages.

$$\begin{cases} (\mathbf{U}\cdot\nabla)k = \nabla[(\nu + \sigma_k \nu_t)\nabla k] + P - \beta^* \omega k, \\ (\mathbf{U}\cdot\nabla)\omega = \nabla[(\nu + \sigma_\omega \nu_t)\nabla \omega] + \frac{\gamma}{\nu_t} P - \beta\omega^2 + 2(1 - F_1)\frac{\sigma_{\omega 2}}{\omega}\nabla\omega\nabla k. \end{cases} \quad (1)$$

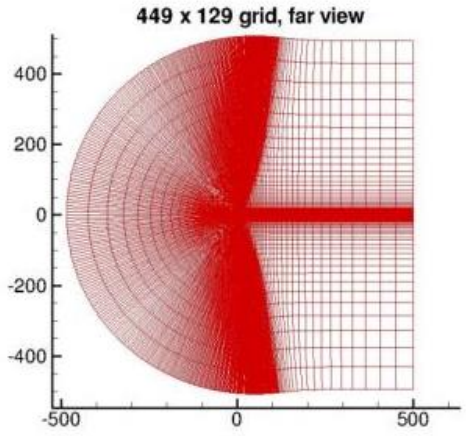
Here k is the specific turbulent kinetic energy ($m^2 s^{-2}$), ω is the specific rate of turbulent dissipation ($s^{-1}$). Other values are presented in works [5-6].

## 3. Solution method

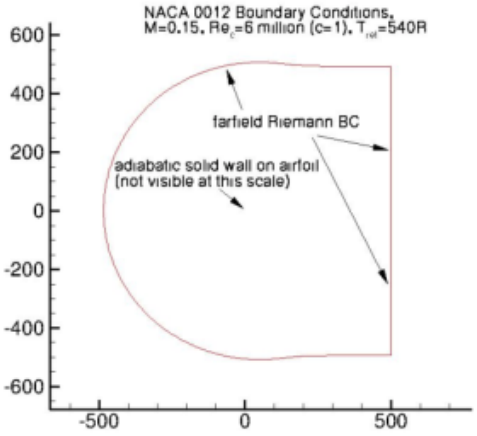
Standard COMSOL Multiphysics solvers were used for the standard SST turbulence model.

## 4. 2DN00: 2D NACA 0012 Airfoil Validation Case

The NACA 0012 turbulent airfoil must be operated under virtually incompressible conditions (recommended here to use M = 0.15 in compressible CFD codes). Reynolds number per chord Re = 6 million. The boundary layers should be completely turbulent over most of the profile of Figure 1. The inflow conditions for the turbulence variables should be specified[14].




a)




b)

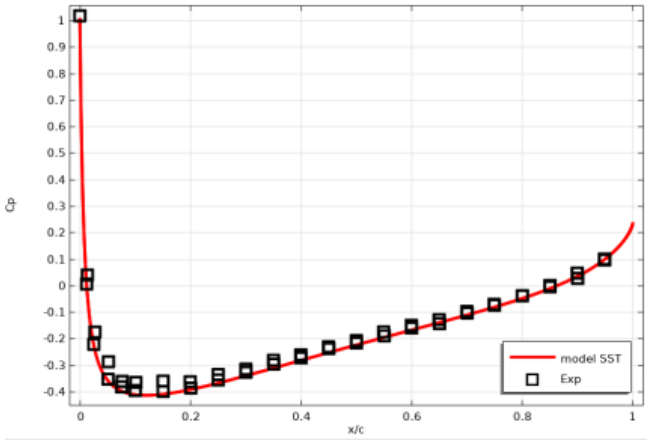
Fig. 1. 2D NACA 0012 Airfoil. a) computational mesh and b) boundary conditions.

The distribution of the surface pressure coefficient on the channel wall is characterized by a change in pressure on its surface depending on the distance from a certain point.

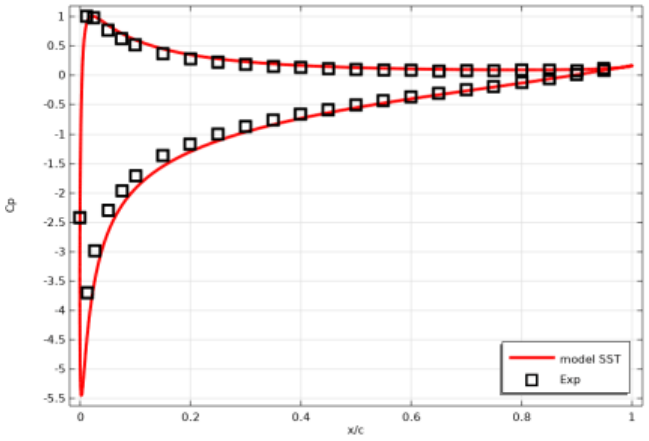
$$C_p = \frac{p - p_\infty}{0.5\rho U_0^2}$$

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, U0 is the speed of the free flow.

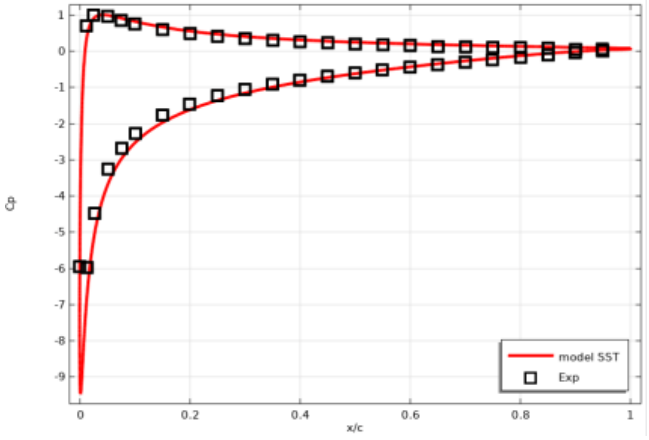
Below are comparisons of the obtained numerical results with known experimental data. Figure 2 shows the pressure coefficient at different angles of attack of the surface profiles as well as the experimental results.




$\alpha$=0




$\alpha$=10




$\alpha$=15

Fig.2 Pressure coefficient at different angles of attack of surface profiles.

Figure 3 shows the effect of angles of attack on lift coefficients.

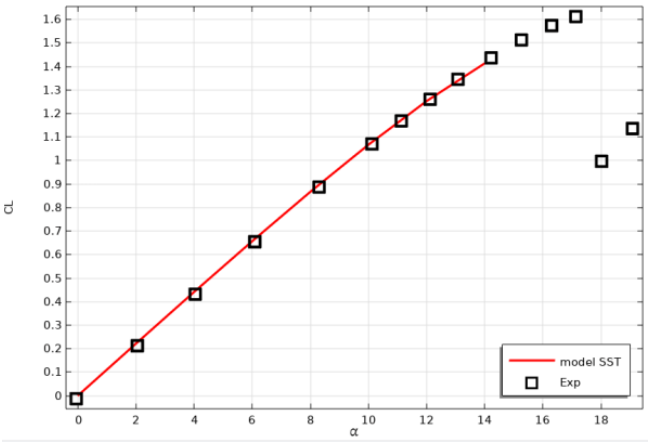
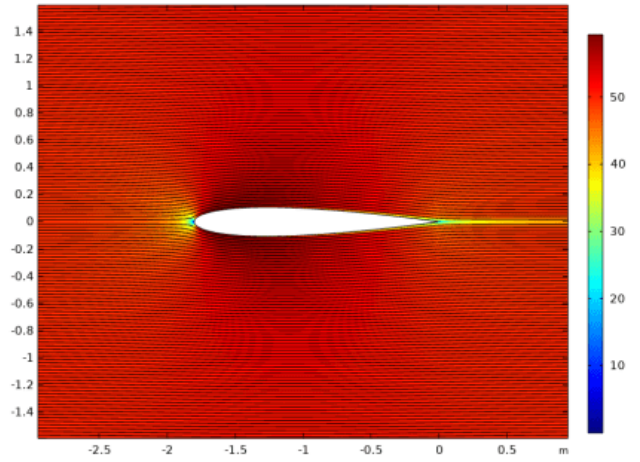


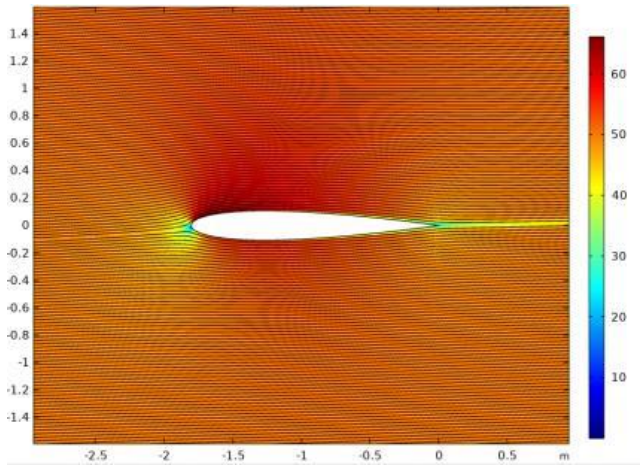

Fig. 3. The influence of angles of attack on lift coefficients.

As can be seen from Figures 2-3, the results of the SST model are close to the experimental results.

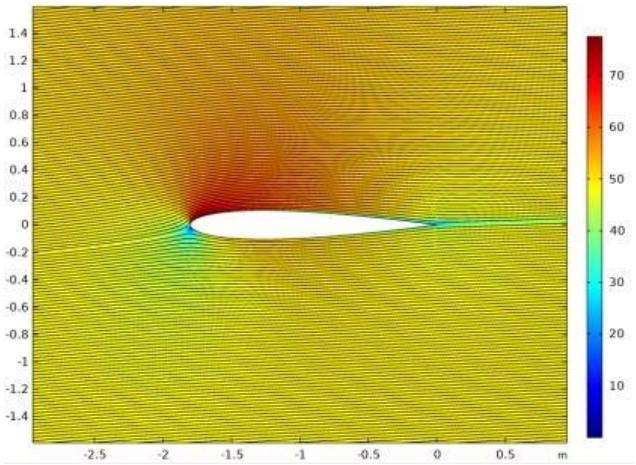
In Fig. Figure 4 shows isolines of the flow velocity at different angles of attack.



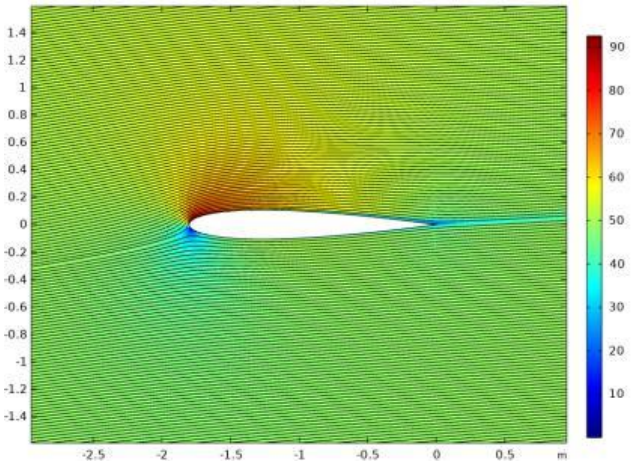
α=0



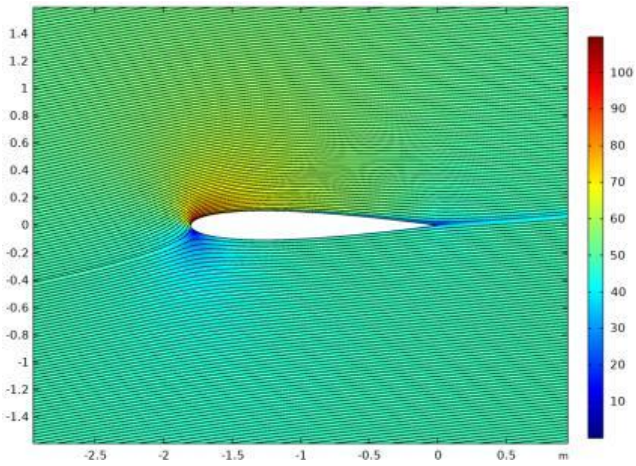
α=2



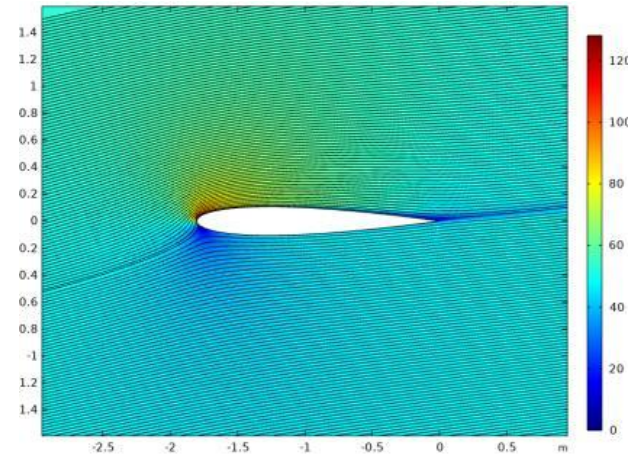
α=4



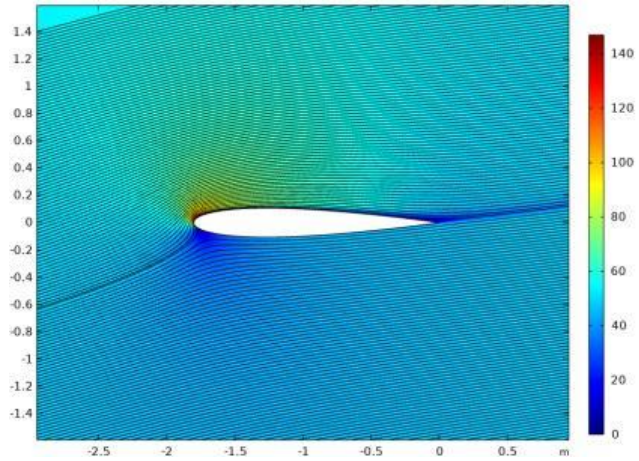
α=6



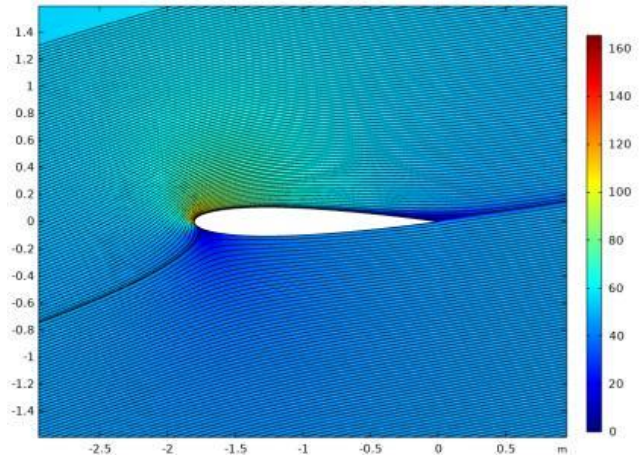
α=8



$\alpha$=10



$\alpha$=12



$\alpha$=15

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

The SST turbulence model is considered to be the best semi-empirical model available today.

## 5. Conclusions

A numerical study of the flow around the NASA 0012 airfoil using CFD techniques provides valuable data for engineers and designers. The results obtained can be used to optimize the profile shape, improve aerodynamic efficiency and develop more efficient aircraft. Understanding the aerodynamic characteristics of an airfoil at the level of numerical simulation contributes to the development of advanced technologies in the field of aviation and aerospace.

This article shows the results of a standard SST turbulence model in the Comsol Multiphysics software package, which uses the finite element method. To validate the SST model, the problems of flow around the NASA 0012 airfoil were considered. From the results obtained, it is clear that the SST 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. [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. [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 son 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. Madaliev, M., Qurbanova, N., & Rustamova, X. (2023). Numerical decisions of the problem of the centrifugal separator based on the sarc turbulence model. Journal of Construction and Engineering Technology, 1(1), 1-6.
13. Rajabov, A. (2023). Numerical simulation of separated flows based on non-stationary navie-stokes equations. Journal of Construction and Engineering Technology, 1(1), 1-4.
14. "Turbulence modeling Resource. NASA Langley Research Center", http://turbmodels.larc.nasa.gov.
15. 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.