Computational Study of Aerodynamic Flow Past Profile S818-NR Based on Advanced Turbulence

Sardorbek Muzaffarov 1,a) and Muzaffar Hamdamov 1, b)

1Institute of Mechanics and Seismic Stability of Structures named after M.T.Urazbaev, Uzbekistan Academy of Sciences, 100125, Tashkent, Dormon yuli, 40, Uzbekistan.

a) Corresponding author: [sardorbekmuzaffarov50@gmail.com](mailto:sardorbekmuzaffarov50@gmail.com)  
b) [mmhamdamov@mail.ru](mailto:mmhamdamov@mail.ru)

**Abstract.** This paper presents the results of a numerical study of the flow around the S818-NR airfoil by a subsonic airflow at angles of attack from 0° to 20°. The simulation was performed by the finite element method in the COMSOL Multiphysics software environment using the SST k-ω (Shear Stress Transport), which allows taking into account the complex nature of turbulent flow. In the course of the work, data were obtained on the distribution of pressure, longitudinal and transverse velocity components, and lift. A comparative analysis of the results was carried out for different values of the Reynolds number, which made it possible to more deeply reveal the influence of flow parameters on the structure of the airfoil flow. The numerical results showed good agreement with the experimental data, which confirms the correctness of the selected turbulence model and the adequacy of the numerical method used. Particular attention is paid to the choice of modeling parameters and methods for processing the obtained data. The results obtained contribute to a deeper understanding of the features of turbulent flow around aerodynamic profiles and can find practical application in the development of reliable engineering solutions in aerodynamics and the design of wind turbines.

# Introduction

Turbulence is one of the most complex and multifaceted phenomena in aerodynamics and hydrodynamics. It arises as a result of nonlinear interactions between gas or liquid particles, causing chaotic and hardly predictable motions. Such a nature of turbulence significantly complicates its mathematical description and numerical modeling.

Understanding turbulent flows is of great importance for various fields of engineering. Turbulence affects the lift, drag, and stability of aircraft, as well as the efficiency of wind turbines. The study of turbulent flows in the blade region of wind turbines is particularly important, since their performance and the optimality of the aerodynamic shape directly depend on it.

In hydrodynamics, turbulent flows determine the nature of water motion in rivers, seas, and pipelines, which is important for the design of hydraulic structures and water supply systems. Turbulence has a significant impact on the performance of various technical devices, such as turbomachinery, pumps, and fans, where high efficiency and reliability are required.

Despite the progress achieved, many aspects of turbulent flows remain insufficiently studied, which highlights the need for further research. Thus, turbulent flows represent a key subject of modern scientific inquiry and have fundamental importance for the advancement of engineering technologies.

At present, due to the rapid development of numerical methods and the increase in computer processing power, the demand for computational fluid dynamics (CFD) methods in the design of aircraft, wind energy systems, and their structural components has grown significantly [1, 2]. CFD technologies are not only more cost-effective compared to full-scale or model experiments, but also provide more comprehensive and detailed information. In many cases, this approach is the only effective way to obtain the necessary scientific data.

To date, problems related to the flow around an aerodynamic profile by an air stream have been studied in a number of scientific works.

In [3], the improvement of wind turbine efficiency through numerical modeling of blade aerodynamics is examined. It is shown that selecting the optimal shape and angles of attack of the profiles is a key factor. Numerical simulations of flow around the blades under unsteady conditions with variable velocity made it possible to obtain parameter fields and streamlines, which can be used to improve wind turbine design.

In [4], mathematical modeling of the flow around small aircraft with low-aspect-ratio wings at moderate Reynolds numbers was carried out. A comparative analysis of numerical simulations and experiments was performed for individual components and small aircraft equipped with a propeller. Calculations based on the RANS equations with the SST turbulence model demonstrated good agreement with experimental data, confirming the reliability of the method.

In [5], the results of numerical simulations of the aerodynamic characteristics of helicopter profiles NACA0012, NACA23012, VR12, and HH02 within the operating range of angles of attack and Mach numbers are presented. RANS-based calculations using ANSYS Fluent took into account transition, stall, and flow compressibility. The obtained data (pressure coefficient distributions and integral characteristics) showed good agreement with the experiment. A comparative analysis of the profiles was carried out, revealing patterns of aerodynamic changes that can be used for blade shape optimization and CFD-based calculations.

In [6], numerical simulations of transonic flow around the NACA 0012 profile were carried out using the RANS equations and the Spalart–Allmaras turbulence model. A wide range of Mach numbers and angles of attack was considered, which made it possible to characterize the features of the flow in the near-sonic regime. The results show the structure of steady and unsteady fields, including shock-wave oscillations, and refine the conditions for transitions between transonic flow regimes.

In [7], using ANSYS Fluent, the influence of the geometry of S-series profiles (S809, S811, S814, S818) on the vortex flow region was analyzed. At Reynolds numbers and angles of attack ranging from 0° to 20°, it was found that the S811 and S818 profiles exhibit the best aerodynamic efficiency, providing high lift and low drag. They are also characterized by a weakened vortex wake and reduced turbulent energy, which makes them promising candidates for wind turbine blades.

In [8], synthetic jet technology was used to control flow separation on the S809 profile at high Reynolds numbers. Using CFD, the aerodynamic characteristics at different angles of attack were analyzed. Optimization of jet parameters was performed by the orthogonal design method, comparing single- and double-jet configurations. It was shown that the synthetic jet effectively suppresses separation and improves aerodynamic efficiency, with the double-jet system proving more effective, particularly due to the higher jet velocity.

In [9], numerical simulations were carried out for the NREL S809 aerodynamic profile. The flow was computed using an unsteady incompressible solver of the Reynolds-averaged Navier–Stokes (RANS) equations. Two turbulence models were applied for RANS closure: the SST Mentor model and the RNG model. All simulations were performed under the assumption of fully turbulent flow. The flow field was analyzed at angles of attack from 0° to 20°. Lift and drag forces were determined by integrating pressure and shear stress over the profile surface. The effectiveness of both turbulence models was compared, and the influence of free-stream turbulence intensity was examined. The results confirm that the SST Mentor model demonstrates satisfactory accuracy in simulating turbulent flow around the aerodynamic profile.

Zhang et al. [10] performed a numerical analysis of the S809 aerodynamic profile using a two-dimensional (2D) finite-difference solver for steady and unsteady, compressible and viscous flows based on the Reynolds-averaged Navier–Stokes (RANS) equations. The study compared four two-parameter turbulence models, both linear and nonlinear. Simulations were conducted at a Reynolds number of and angles of attack from 0° to 70°. However, the modeling results did not match the experimental data in the post-stall region.

Bertagnolio et al. [11] performed steady numerical simulations using the two-dimensional EllipSys2D solver for incompressible Navier–Stokes equations, applied to a wide range of wind turbine aerodynamic profiles. Their results showed that the code is sensitive to the profile shape and demonstrates good agreement with experimental data when the laminar-to-turbulent transition location is accurately determined.

Du and Selig [12], as well as Hu et al. [13], investigated the three-dimensional rotational effects for a blade based on the S809 aerodynamic profile (while Chaviaropoulos and Hansen [14] studied blades based on NACA63-4XX and NACA63-2XX profiles). The results showed that flow separation is delayed on the rotating blade. Hu et al. [13] concluded that Coriolis and centrifugal forces play a key role in the mechanism of three-dimensional stall delay.

Although numerous scientific studies have already been conducted in this area, continuing research remains one of the current important scientific challenges.

The main objective of this work is to investigate the airflow around helicopter and wind turbine blade profiles using the Reynolds-averaged Navier–Stokes (RANS) equations, as well as to determine the distributed and integral aerodynamic characteristics over the operating range of angles of attack.

From a practical point of view, the most interesting aerodynamic profiles are those with a relative thickness of 10–18%, since such profiles are typical for the mid-sections of helicopter and wind turbine blades. These sections make up the main part of the blade span and largely determine the aerodynamic behavior of the entire blade under all flight or operational conditions.

The S818-NR aerodynamic profile was developed by the U.S. National Renewable Energy Laboratory (NREL) and is primarily intended for use in small- and medium-scale wind energy installations. This profile is optimized for efficient operation at low and medium wind speeds, providing high aerodynamic efficiency and stable electricity generation under variable wind conditions.

# Let us focus on the main characteristics of the S818-NR profile

The S818-NR aerodynamic profile possesses several important characteristics. In particular, its maximum relative thickness is approximately 18%, which contributes to high structural strength. This profile is distinguished by a high lift coefficient and a low drag coefficient, making it more efficient compared to other profiles.

Thanks to its high resistance to flow separation, the profile is capable of operating efficiently even at large angles of attack. Furthermore, the S818-NR design accounts for the transition between laminar and turbulent regions, which promotes flow stability and efficient energy conversion.

Nevertheless, in the existing scientific literature, the maximum lift values, flow separation regions, aerodynamic behavior at different Reynolds numbers, and the profile’s stability at various angles of attack have not been studied comprehensively and in depth.

In this regard, the aim of the present study is to perform a numerical analysis of these parameters under various conditions. The obtained results will allow the assessment of the aerodynamic efficiency of the S818-NR profile and the development of practical recommendations for its application in the design of wind energy installations to improve their performance.

In this study, we focus on the numerical simulation of turbulent flow around the S818-NR profile. This profile is widely used in aerodynamic research due to its simplicity and favorable lift characteristics. We will investigate the flow around the profile at various angles of attack, ranging from 0° to 20°. For numerical modeling, we use the COMSOL Multiphysics software package, which provides extensive capabilities for solving a wide range of continuum mechanics problems, including the simulation of turbulent flows.

Aerodynamic studies are an important component in the development and optimization of wind turbines. The S818-NR is one of the most widely used aerodynamic profiles. Efficient methods, such as computational fluid dynamics (CFD), can be employed to study the flow around the profile and determine its aerodynamic characteristics.

Studies using computational fluid dynamics (CFD) methods allow virtual experiments by simulating airflow around an aerodynamic profile under various operating conditions. This approach is particularly relevant in the field of wind energy, where the airflow exhibits a wide range of velocities and angles of attack.

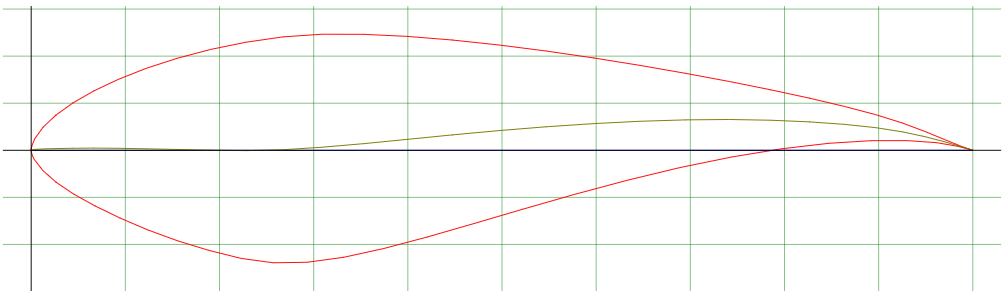
Various turbulence models and numerical methods are used to evaluate aerodynamic parameters. One of the most effective is the SST (Shear Stress Transport) turbulence model, which is employed to solve the Navier–Stokes equations describing the behavior of viscous fluids or gases. This model allows for the consideration of the transition between laminar and turbulent flow regimes, as well as accurate reproduction of flow characteristics near the profile surface and in potential flow separation regions.

The main goal of this study is to assess the adequacy of the chosen turbulence model by comparing numerical results with experimental data. Successful agreement between these results will confirm that the model is suitable for solving practical engineering problems and will increase confidence in numerical methods in the field of aerodynamics.

In the following sections of the article, the research methodology will be presented in detail, the obtained results will be reported, and their practical significance and potential applications will be analyzed.

# Physical and Mathematical Formulation of the Problem

The turbulent S818-NR wing profile is intended to operate under practically incompressible conditions. The Reynolds number based on the chord is.  Fig 2. shows the computational mesh.



**FIGURE 1.** 2D S818-NR Profile

|  |  |
| --- | --- |
|  |  |
| **FIGURE 2.** Computational Mesh | |

In the present study, commercial software COMSOL Multiphysics was used to model the flow around a two-dimensional aerodynamic profile. The calculations were performed based on the unsteady Reynolds-averaged Navier–Stokes (RANS) equations, assuming a steady, incompressible, and two-dimensional (2D) flow. These equations form the basis of the mathematical description of incompressible fluid dynamics and represent a system of differential equations that model the variations of velocity and pressure in a fluid over time and space.

Since the flow is not fully turbulent, the SST  turbulence model, based on four equations, was chosen for a more accurate description of the flow characteristics. A fully coupled scheme was used to account for the relationship between pressure and velocity. A second-order discretization scheme was applied for the pressure and momentum equations [26].

The main equations used in the study are as follows:

The continuity equation was solved using a steady-state approach:

, (1)

Time-averaged momentum equation:

 (2)

# Turbulence Model

The SST  (Shear Stress Transport ) model is an improved turbulence model developed by Frank Menter for more accurate calculation of flows with strong pressure gradients, particularly near walls and under flow separation conditions. It is widely used in aerodynamics, including the simulation of wind turbine blades, wings, diffusers, and similar applications.

The SST Menter model represents a combination of the  and  models. The method employs the  model near the wall, but then (through a special function) switches to the  model away from the wall (i.e., closer to the outer edge of the boundary layer).

Equation  is transformed into the  form (see [18]).

This approach makes it possible to exploit the advantages of both models:

The  model does not require wall corrections and provides a more accurate description of flows with an adverse pressure gradient. However, the solution depends on the values of the turbulent variables in the external flow, particularly on  [27].

The  model is less sensitive to these variables, but requires modification near the wall.

It has been demonstrated that Menter’s SST  model provides significantly better results in simulating flows with a strong adverse pressure gradient compared to the original  and  models (Kral [19], Catalano and Amato [20], Tulapurkara [21]).

According to Kral [19], this improvement is attributed to the introduction of the turbulent viscosity limiter proposed by Menter. As reported by Catalano and Amato [20], compared to other models, it is specifically Menter’s SST  model that is capable of predicting the sharp flow separation observed in experiments. The results obtained are comparable to those of nonlinear models, while the SST model is less rigid and less computationally demanding.

This model has been successfully applied to numerical simulations of isolated wing flows (Guilmineau et al. [22], Sorensen and Michelsen [23]), as well as in the aerodynamic analysis of wind energy systems (Sorensen et al. [24]).

The model is based on the following two transport equations:

Turbulent kinetic energy :

 (3)

Specific dissipation :

 (4)

The following formula is used to determine the turbulent viscosity in the SST  model:

 (5)

The coefficient  indicates the rate at which turbulent energy 𝑘 is generated in the flow as a result of velocity shear or flow deformation:

 (6)

The SST  model employs two key switching functions -  and  that determine which turbulence model is applied in different flow regions.

 - is active in the near-wall regions and ensures the use of the  model close to the wall. This function is particularly important for the accurate simulation of laminar-to-turbulent transition and shear layers, where the  model provides more precise results:

 (7)

 begins to dominate in regions away from the wall, facilitating a smooth transition to the  model. This is important in free-stream regions, where the  model more accurately represents turbulence:

. (8)

In (7) and (8) *ρ* – denotes the density; *U* – is the velocity vector; μ – represents the molecular (dynamic) viscosity;  – turbulent viscosity; – source term of turbulent kinetic energy, *y* – distance to the nearest wall (m), *α, β, β∗, σk, σω,  σω2*- model constants; – modified diffusion term:

,

where  is the scalar product of the gradients of *k* and .

The closure of the system requires the use of the equation of state for a perfect gas.

, (9)

where *R* is the specific gas constant for air. *T=293,15 K* – air temperature.

For the considered cases, the Reynolds number was

,

which corresponds to a developed turbulent flow regime over a solid surface. Thus, a value of  was adopted in the calculations. Accordingly, the turbulence model was selected.

The use of Reynolds-averaged Navier–Stokes equations makes it possible to account for turbulence effects and their influence on the flow around the S818-NR airfoil. These equations are solved using numerical methods, such as the finite element method, with the application of specialized software packages like COMSOL Multiphysics. This approach provides detailed data on the flow characteristics and its impact on the airfoil.

The study of the SST turbulence model for turbulent flow around the S818-NR airfoil is the objective of this article. The numerical data obtained are compared with the well-known experimental data available on the NASA Turbulence Modeling Resource (TMR) website [17].

The boundary conditions on the semicircle and on the boundaries (at the  boundary, Fig. 2) are specified depending on the angle of attack.

, , (10)

where - is the velocity magnitude of the incoming free-stream flow;  is the angle between the flow direction and the OX coordinate axis.

At this boundary, the value of the excess pressure is equal to zero:

 (11)

Accordingly, for the boundary







the conditions were specified .

At the outlet of the computational domain  (), a smooth matching condition was applied to determine the velocities.

. (12)

# Solution Method

For the standard SST turbulence model, the standard solvers of COMSOL Multiphysics were used.

The computations were carried out for 50000<Re<1000000 at various angles of attack within the range of .

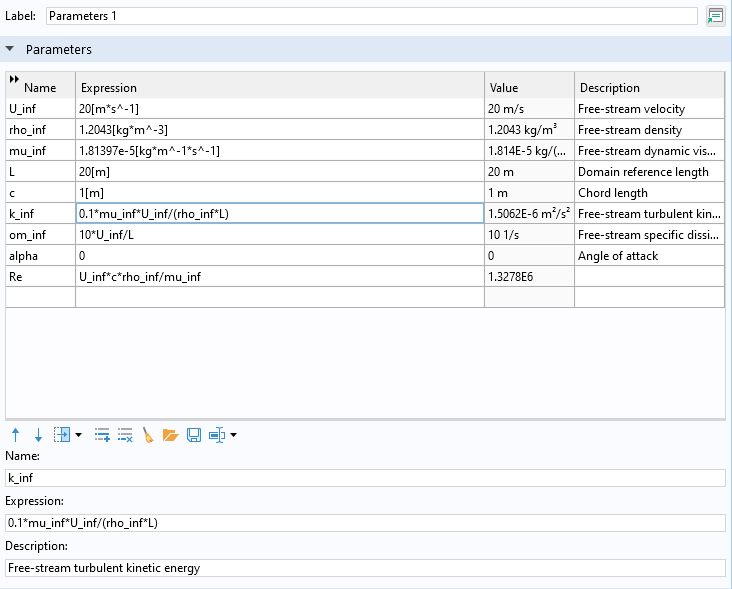
In the calculations, local mesh refinement was applied near the surface of the *f1(x)* and *f2(x)* profile, as shown in Fig. 1.

For the flow domain around the profile, a mesh consisting of 50,600 elements was used. To ensure stable convergence of the calculations, the relaxation coefficient was set to .

The steady-state method was used to solve the problem. The results of potential flow were employed as the initial approximation. The principle of operation of the finite volume method is that, for a fixed time, the velocity fields are first determined. These are then used to compute pressure increments, with respect to which the Poisson equation is formulated.

On average, the program ran for two or more hours. The outputs included fields of longitudinal and transverse velocities, pressure fields, turbulent kinetic energy and its dissipation, as well as aerodynamic derivatives.

The input data for the calculation included the following quantities: wind velocity in  , angular velocity in rad/s, air density in , and pressure in  (see Fig. 3).



**FIGURE 3.** Input data window with dimensions and initial values

# Results and Discussion

The pressure coefficient (or relative pressure coefficient) is a dimensionless quantity that shows the ratio of the difference between the local pressure on the surface of a body and the pressure of the incoming flow to the dynamic pressure of the flow.

The change in excess pressure along the channel wall depending on the distance to the top is called the distribution of the surface pressure coefficient:

|  |  |
| --- | --- |
| . | (13) |

Here p is the pressure at a point on the profile surface; *p∞* − free flow pressure; *ρ* − free flow density; *U0* − free flow velocity.

Comparisons of the obtained numerical results with known experimental data are shown below. Fig. 4 shows the pressure coefficients for different angles of attack of the profile surface.

Dependence of the pressure coefficient *Cp* from the angle of attack is an important tool for the analysis and optimization of airfoils. It allows one to evaluate the efficiency of the airfoil under different conditions and use this information to design more efficient aerodynamic shapes.

|  |  |
| --- | --- |
|  |  |
| α=50 | α=100 |
|  | **FIGURE 4.** Distribution of the surface pressure coefficient *Cp* at different angles of attack |
| α=150 |

In the area of the leading edge (x≈0–5 mm) a sharp drop in the pressure coefficient *Cp* is observed. This indicates an acceleration of the air flow along the upper surface of the profile, which causes a significant decrease in static pressure. As a result, a strong vacuum zone appears on the upper surface, which causes the formation of a large lifting force.

In the upper surface area in the range 5 mm <x< 70 mm, the pressure coefficient retains strongly negative values, but gradually increases, indicating partial pressure recovery. However, the low pressure zone remains wide, indicating a strong aerodynamic effect and simultaneously increasing the risk of flow separation. The slow pressure recovery may be due to a decrease in the pressure gradient, due to which the inertial flow is difficult to maintain on the profile surface.

On the lower surface, on the contrary, the pressure is relatively higher (in some places the pressure coefficient even takes positive values), which indicates a slower movement of the air flow and, accordingly, a higher pressure. A significant difference in pressure between the upper and lower surfaces of the profile leads to the emergence of a significant lifting force.

At the rear of the profile (x≈90–100 mm), the *Cp* values on the upper and lower surfaces approach each other, but do not completely equalize. This indicates that the air flow around the profile is not completely restored, and residual vortex phenomena and energy losses in the flow are possible.

|  |  |
| --- | --- |
|  |  |
| α=20 | α=60 |
|  |  |
| α=100 | α=140 |
|  |  |
| α=180 | α=200 |
| **FIGURE 5.** Field of transverse velocity at | |

In the area of the leading edge of the profile, especially near the upper surface, a zone with a high positive value of the transverse (*y*-component) velocity is observed (Fig. 5), which is reflected as a red-green color on the flow map. This indicates an accelerated upward movement of the air mass along the upper surface of the profile. In this area, an intensive decrease in pressure occurs, a rarefaction zone is formed, which contributes to the formation of a significant lifting force.

On the upper surface of the profile, the flow is deflected upward in the y -direction, indicating that it is adjacent to the surface. The color range varies from green to yellow and red, indicating a high value of the y -component of the velocity. This, in turn, corresponds to a section with increased flow velocity and reduced pressure.

On the lower surface of the profile, blue zones are observed, indicating a downward transverse flow. Here, the flow is relatively slower, and the y -component of the velocity is negative and has a small value. This indicates the presence of a region with higher pressure below the profile.

The flow over the upper and lower surfaces has a multidirectional movement and does not completely level off towards the trailing edge. This indicates that the flow has not yet been fully restored, and there is a possibility of separation zones or low flow energy.

For the S818 airfoil at the calculated Reynolds number , the maximum lift coefficient is 1.68.

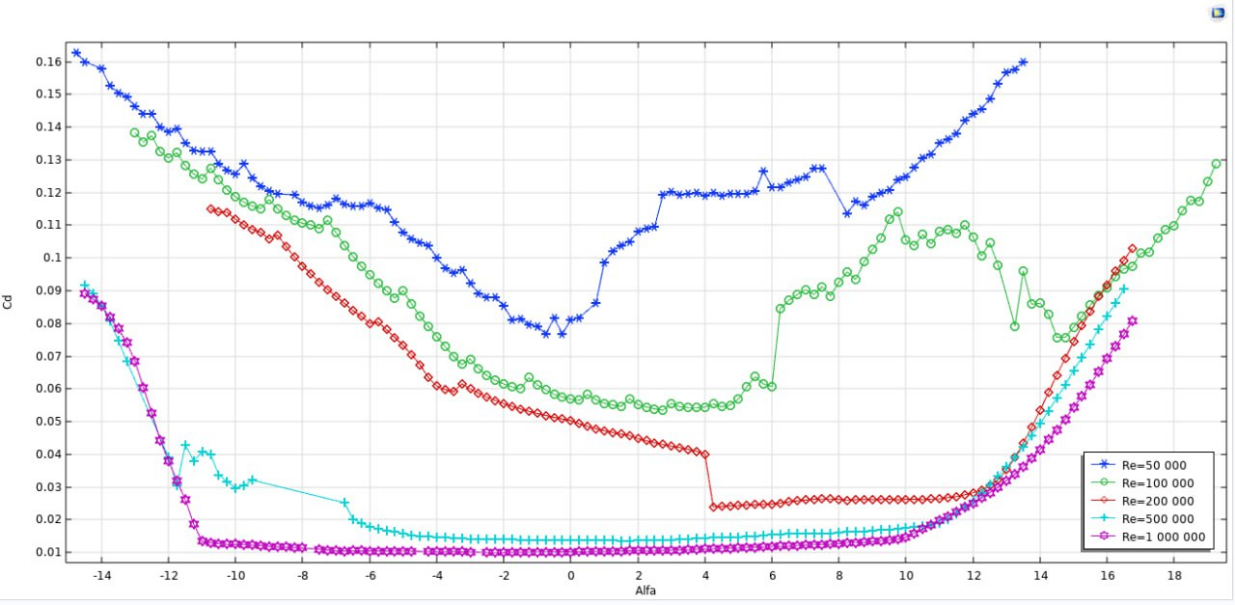
Such behavioral characteristics are particularly important when selecting a profile for both low-speed vehicles and high-speed installations such as wind turbines or large unmanned aerial vehicles.

The drag coefficient (𝐶𝐷) is also one of the key parameters when assessing the aerodynamic characteristics of the profile. This coefficient reflects the degree of resistance of the profile to the air flow. In combination with the lift coefficient (𝐶𝐿), the coefficient 𝐶𝐷 plays a key role in the analysis of the overall aerodynamic efficiency of the profile. The ratio CL/CD is especially important, the maximum value of which serves as a criterion for determining the optimal operating mode.

The drag coefficient CD is a dimensionless value that characterizes the ability of a body (for example, an aerodynamic profile, wing or body) to resist air movement. It is determined by the formula:

|  |  |
| --- | --- |
|  | (14) |

Here CD is the drag coefficient (dimensionless value); D is the aerodynamic drag force (N); V – air flow velocity (m/s), S – wing projection area (profile) (m²).



**FIGURE 6.** Analysis of the dependence of the angle of attack on the drag coefficient (CD ) at different Reynolds numbers

Below is an analysis of the dependence of the drag coefficient CD of the S818-NR aerofoil on the angle of attack α at different Reynolds numbers (Re).

In this range Re=50,000 – 100,000 The CD values are relatively high, especially in the range of 00–60. This is due to the early laminar flow separation, in which the flow around the profile becomes unstable and separation zones are formed in the upper layer. As the angle of attack increases, the drag coefficient CD increases, which reduces the overall aerodynamic efficiency of the profile. When approaching the stall angle – at approximately 6–80 – there is a sharp increase in CD due to complete flow separation and the emergence of pronounced turbulence.

In the range of Re = 200,000 – 500,000, the initial values of CD are significantly lower, indicating stable and smooth flow around the profile surface. With increasing angle of attack, CD increases, but no sharp jumps are observed until the flow stalls. It is especially worth noting the behavior at Re = 500,000, where CD is weakly sensitive to changes in the angle of attack and the profile maintains high stability and aerodynamic efficiency.

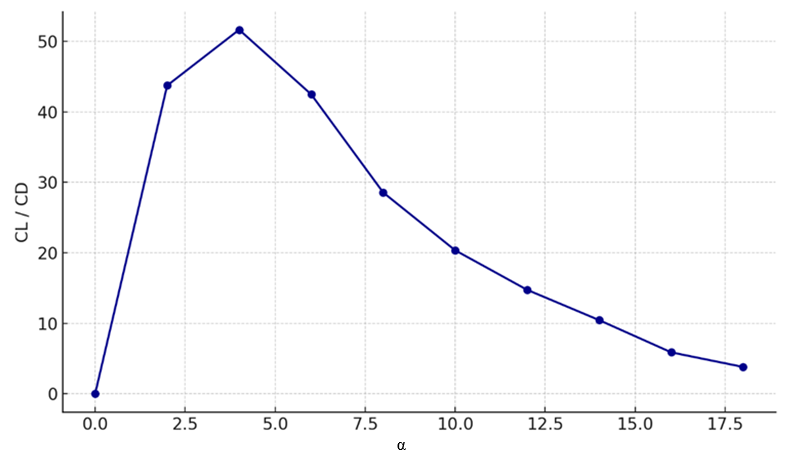
At Re ≥ 1000000, the lowest initial values of CD are observed, since the flow is completely near-wall, and there are no laminar separations. As the angle of attack increases, CD increases very slowly, and even when a separation occurs, there is no sharp jump in CD, which indicates high stability and flow quality.

the Reynolds number increases, the CD coefficient decreases, indicating improved wall flow and reduced drag. At low Re, laminar separation occurs earlier, increasing CD and degrading aerodynamic performance. At high Re (≥ 500,000), the airfoil exhibits low drag, stable behavior, and delayed flow separation, making it particularly suitable for wind power and aircraft applications.

In aerodynamic optimization problems, the relationship between the lift coefficient (𝐶𝐿) and the drag coefficient (𝐶𝐷) is of significant importance when

CL/CD → maximum.

This value determines the aerodynamic quality of the profile and helps to determine the most advantageous angle of attack for efficient flight.



**FIGURE 7.** Graph of aerodynamic quality CL/CD depending on the angle of attack α

Fig. 7 shows the change in the aerodynamic quality CL/CD depending on the angle of attack α for the S818 airfoil calculated using the SST k-ω turbulence model. At small angles of attack (2–60), a sharp increase in CL/CD is observed – the maximum value (~52) is achieved at an angle of 50 This indicates the most efficient aerodynamic operation of the airfoil. After the peak, a decrease in CL/CD occurs – at α=8–120, the efficiency drops significantly, which is associated with an increase in drag and, possibly, the beginning of the transition to separated flow. At angles above 140, a sharp drop in quality is observed – to a value of 5 at α=180, which indicates flow separation and a significant increase in drag.

Thus, **the S818 profile operates optimally in the angle of attack range of 40 to 80**, especially at medium and high Reynolds numbers. This range should be taken into account when designing wind turbine blades to ensure maximum efficiency.

# Conclusion

The article presents the results of numerical simulation of the flow around the S818-NR airfoil using the standard turbulent model SST k-ω, implemented in the COMSOL Multiphysics environment based on the finite element method. During the study, the velocity distributions, pressure coefficients and other key aerodynamic parameters were studied.

The application of the SST k-ω model showed correct reproduction of the lift and drag characteristics for the S818-NR profile. The greatest agreement with the experimental results is observed at attack angles up to 12–14°, which confirms the reliability of the chosen method. Therefore, this approach can be effectively used for numerical analysis of aerodynamics in the range of medium attack angles.

The results obtained provide engineers and designers with valuable data that can be used to improve aerodynamic efficiency, optimize airfoil geometry, and develop more advanced wind turbines and aircraft. Numerical aerodynamic studies contribute to the development of innovative technologies in the field of renewable energy.

The S818-NR profile is characterized by low aerodynamic resistance, which helps reduce fuel costs and improve the speed characteristics of aircraft. Its design is aimed at minimizing turbulent effects, providing stability and improved controllability.

Due to its high aerodynamic performance, the S818-NR profile is considered a promising option for designing wind turbine blades. Its geometry provides significant lift with low resistance, making it the optimal choice for modern power plants.

# ACKNOWLEDGMENTS

This study was made possible by budget funding from the Uzbekistan Academy of Sciences. We express our sincere gratitude to the Academy for its ongoing support and commitment to advancing scientific research.

# References

1. W. Orozco Murillo, J. A. Palacio-Fernande, I. D. Patiño Arcila, J. S. Zapata Monsalve and J. A. Hincapié Isaza, Journal of Applied and Computational Mechanics, **6**, 1228–1244 (2020).
2. K. Hadad, H. R. Eidi and J. Mokhtari, Journal of Applied and Computational Mechanics, **3** (3), 171–177 (2017).
3. D. Yu. Evseev and O. K. Ovchinnikova, Aerospace Engineering and Technologies, **1** (1), 117–130 (2023).
4. O. G. Buzykin, A. V. Kazakov, and A. V. Shustov, TsAGI Science Journal, **XLI** (5), (2010).
5. A. L. Tarasov, MAI Proceedings, **131**, (2023). DOI: 10.34759/trd-2023-131-13.
6. A. A. Prikhodko, O. B. Polevoy, A. A. Pilipenko, I. I. Lipatov, and R. Ya. Tugazakov, Modern Science: Research, Ideas, Results, Technologies, Issue **2**(10), 138–143 (2012).
7. M. J. Hamzah, L. Al-Sadawi, A. Khudhair and T. Biedermann, Engineering and Technology Journal, **41**, (2023).
8. J. Feng, Y. Lin, G. Zhu and X. Luo, Sādhanā, **44**, 1–10 (2019). https://doi.org/10.1007/s12046-019-1173-2
9. O. Guerri, K. Bouhadef and A. Harhad, Wind Engineering, **30** (4), 287–302 (2006).
10. S. Zhang, X. Yuan and D. Ye, Wind Engineering, **25**, 41–53 (2001).
11. F. Bertagnolio, N. Sørensen, J. Johansen and P. Fuglsang, Wind Turbine Airfoil Catalogue, RISOE-R-1280(EN), RISO (2001).
12. Z. Du and M. S. Selig, Renewable Energy, **20** (2), 167–181 (2000).
13. D. Hu, O. Hua and Z. Du, Renewable Energy, **31**, 821–836 (2006).
14. P. K. Chaviaropoulos and M. O. L. Hansen, ASME Journal of Fluids Engineering, **122**, 330–336 (2000).
15. P. Fuglsang and C. Bak, “Development of the Riso wind turbine airfoils,” Wind Energy, **7**, 145–162 (2004).
16. X. Li, K. Yang, L. Zhang, J. Bai and J. Xu, Journal of Renewable and Sustainable Energy, **6**, 033110 (2014).
17. Turbulence Modeling Resource, NASA Langley Research Center. URL: http://turbmodels.larc.nasa.gov
18. F. R. Menter, AIAA Journal, **32** (8), 1598–1605 (1994).
19. L. D. Kral, Progress in Aerospace Sciences, **34**, 481–541 (1998).
20. P. Catalano and M. Amato, Aerospace Science and Technology, **7**, 493–509 (2003).
21. E. G. Tulapurkara, Progress in Aerospace Sciences, **33**, 71–165 (1997).
22. E. Guilmineau, J. Piquet and P. Quentey, Computers and Fluids, **26**, 135–162 (1997).
23. N. N. Sørensen and J. A. Michelsen, ASME Journal of Solar Energy Engineering, **126**, 1011–1016 (2004).
24. N. N. Sørensen, J. A. Michelsen and S. Schreck, Wind Energy, **5**, 151–169 (2002).
25. M. M. Hamdamov, A. I. Ishnazarov and K. A. Mamadaliev, Lecture Notes in Computer Science, **13772 LNCS**, 156–170 (2023).
26. M. Hamdamov and S. Muzaffarov, “Simulation of turbulent flow around a thin rectangle using k–ε turbulence and environment models in COMSOL Multiphysics,” in *ESSM2024*, AIP Conference Proceedings 3265, edited by R.Abirov (AIP Publishing, Melville, NY, 2025), 060013.
27. M. M. Hamdamov, R. A. Fayziyev and S. S. Muzaffarov, “Numerical simulation of wind turbines conducted using COMSOL software,” in E3S Web of Conferences 541, edited by D. Nazarov *et al.* (EDP Sciences, Almaty, Kazakhstan, 2024), 01001