Numerical study of flow after NASA 4412 aerodynamic profile based on the SST turbulence model

Murodil Madaliev[1,2*], *Akmaljon* Mamatov[2], *Jamshidbek* Otajonov[3], *Khasanboy* Kadyrov[3], *Inomjan* Bilolov[3], *Zamira* Otakhanova[3], and *Kamolkhan* Muminov[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]Ferghana branch of Tashkent University of Information Technology, Fergana, Uzbekistan

Abstract. This article investigates the application of the SST turbulence model within the Comsol Multiphysics platform to analyze subsonic airflow around a NACA4412 airfoil at an angle of attack of 13.87 degrees. The study employs the finite element method to solve the turbulence equations numerically, with stabilization achieved through the Galerkin least squares technique. Comparisons between the computational results and experimental data are conducted to assess the model's performance. The findings demonstrate favorable agreement between the SST turbulence model predictions and experimental observations, indicating promising convergence. This research contributes to the understanding of turbulence modeling in aerodynamics simulations and underscores the potential of the SST model in accurately capturing flow behavior around complex geometries like the NACA4412 airfoil.

1 Introduction

Aerodynamic research stands as a pivotal aspect of aircraft development and enhancement, with the NASA 0012 airfoil emerging as a widely utilized design. Employing numerical techniques, such as computational fluid dynamics (CFD), offers a robust avenue to scrutinize airflow around specific airfoils and discern their aerodynamic attributes.

A Review of Numerical Methods:

CFD Simulation:

Through CFD studies, researchers can engage in virtual experiments, simulating airflow around airfoils under diverse conditions. This approach involves solving the Navier-Stokes equations and turbulence equations utilizing various methodologies like the k-ε and k-ω models to estimate aerodynamic parameters.

Profile Geometric Characteristics:

An in-depth analysis of thickness and camber across distinct chord sections facilitates comprehension of how geometric intricacies influence aerodynamic traits.

* Corresponding author: Madaliev.me2019@mail.ru

Pressure Distribution:

Examining pressure distribution along the airfoil surface aids in pinpointing high and low-pressure regions, pivotal for understanding lift force generation.

Lift and Drag:

Assessing lift and drag across varying angles of attack facilitates determination of optimal profile utilization conditions.

Border Effects:

Investigating the influence of airfoil proximity to solid surfaces and wing tip effects aims to refine the consideration of real-world application conditions.

Computational Fluid Dynamics (CFD) serves as the discipline for numerically solving equations governing fluid motion, applicable across various hydrodynamic scenarios including aerodynamic flows around aircraft, turbulent flows, heat transfer, and fluid mixing. At its core lie the Navier-Stokes equations, embodying mass, momentum, and energy conservation principles in fluids.

Direct analytical solutions of Navier-Stokes equations remain feasible solely for rudimentary geometries and flows. In practice, gridding space and numerical solutions at each grid node become imperative. Numerical CFD methods like finite difference, finite element, or finite volume techniques enable comprehensive Navier-Stokes equation solutions across domains, encompassing boundary and initial conditions.

The widespread application of CFD across engineering domains enables system design and analysis while adhering to fundamental principles governing fluid and gas operations. In aviation, for instance, CFD optimizes wing and fuselage contours, minimizes drag, and enhances aerodynamic efficacy. Similarly, CFD has found utility in automotive, maritime, turbine, HVAC systems, and diverse engineering spheres, amplifying design and analysis efficiencies across fluid and gas dynamics realms.

Thus, CFD epitomizes a critical engineering discipline, fostering enhanced design and analysis capabilities across systems intertwined with fluid and gas dynamics [1–2].

The Navier-Stokes equations constitute a set of differential equations delineating the motion of incompressible fluids.

$$\begin{cases} \dfrac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\nabla\mathbf{v} = -\dfrac{\nabla p}{\rho} + \nu\nabla^2\mathbf{v} + \mathbf{F} \\ \nabla\mathbf{v} = 0 \end{cases} \tag{1}$$

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 primary equation, known as the Navier-Stokes equation of motion, delineates the alteration in fluid velocity over time, subject to external forces, viscosity, and pressure. Its counterpart, the continuity equation, ensures conservation of the liquid's mass. Navier-Stokes equations stand as nonlinear constructs, posing challenges in resolution. Nonetheless, they serve as fundamental tools for simulating fluid dynamics across diverse scientific and technological domains, including aerodynamics, hydrodynamics, and oceanography. While the Navier-Stokes equation embodies a mathematical model for laminar flow, many real-world flows exhibit turbulence, necessitating the modeling of turbulent characteristics within the equation framework [5–11].

The NASA Airfoil 4412, also known as the NACA 4412 airfoil, is a widely recognized aerodynamic profile extensively used in various aerospace applications. Developed by the National Advisory Committee for Aeronautics (NACA), the 4412 airfoil belongs to the NACA 4-digit series, which indicates specific geometric properties.

The "4412" designation provides insight into the airfoil's shape: The first digit "4" signifies the maximum camber, which is 4% of the airfoil chord length. The second digit "4" represents the location of maximum camber, measured as a percentage of the chord length from the leading edge.

The last two digits "12" denote the thickness-to-chord ratio, indicating that the airfoil is 12% thick.

The NACA 4412 airfoil is favored for its favorable lift-to-drag characteristics, making it suitable for a wide range of applications, including general aviation, unmanned aerial vehicles (UAVs), and wind turbine blades.

Engineers and researchers frequently utilize computational methods, such as computational fluid dynamics (CFD), to analyze the aerodynamic performance of the NACA 4412 airfoil under various operating conditions, including different angles of attack and Reynolds numbers. These analyses aid in optimizing airfoil designs for specific applications, enhancing overall efficiency and performance.

The purpose of this article is to study the SST turbulence model for problems of turbulent flow around the NASA 4412 airfoil. The obtained numerical results are compared with known experimental data, which are presented on the NASA Turbulence Modeling Resource (TMR) website [12].

2 SST turbulence model

Menter's Shear Stress Transport (SST) model [5-6] represents a hybridization of the k-ε and k-ω turbulence models, each serving distinct roles within different flow regions. Specifically, the k-ω model is applied in the near-wall layer to capture boundary layer phenomena accurately, while the k-ε model governs turbulence behavior in the outer flow regions. This dual-mode approach allows the SST model to effectively capture both near-wall turbulence and free-stream turbulence, providing more accurate predictions across a wide range of flow regimes. Renowned for its versatility and robustness, Menter's SST model enjoys widespread adoption and integration within numerous Computational Fluid Dynamics (CFD) software packages. Its popularity stems from its ability to deliver reliable turbulence predictions across various engineering applications, ranging from aerospace and automotive design to environmental and industrial fluid dynamics scenarios. As such, the SST model stands as a cornerstone tool in modern CFD simulations, facilitating comprehensive analyses and insights into complex turbulent flows.

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

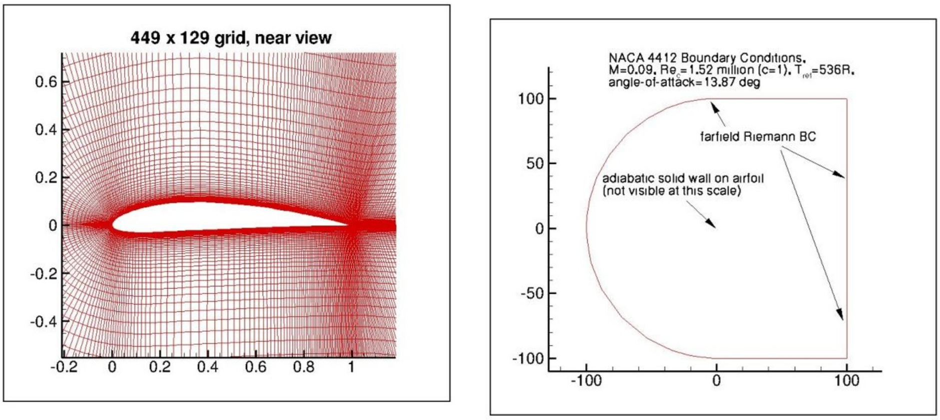
Here k is the specific turbulent kinetic energy ($m^2\ s^{-2}$), ω is the specific rate of turbulent dissipation (s−1). Other values are presented in works [5-6].

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 4412 airfoil is an airfoil with a maximum thickness of 12% of the chord length, located at 40% of the chord length. It is widely used in various fields, including aircraft wings and propeller blades. This section presents the validation of a two-fluid turbulence model for the NACA 4412 airfoil. The case is considered for the Mach number M = 0.09, the angle of attack α = 13.87 ◦ and the Reynolds number Re = 1520000. 449x129 grid is presented in. The boundary conditions are shown in Figure 13b, and an illustration of the computational grid and domain is shown in Figure 1a. [13].



a) b)

Fig. 1. 2D NACA 4412 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. In Figure 2, the pressure coefficient at various angles of attack of the surface profiles is depicted alongside the corresponding experimental results. The pressure coefficient, denoted as Cp, serves as a fundamental parameter in aerodynamics, providing insights into the pressure distribution along the airfoil surface.

The numerical simulations conducted using the SST turbulence model within the COMSOL Multiphysics framework exhibit promising agreement with the experimental data across different angles of attack. As shown in Figure 2, the pressure coefficient trends obtained from the simulations closely resemble the experimental trends, demonstrating the model's capability to capture the underlying flow physics accurately [14-16].

At lower angles of attack, the pressure coefficient distribution exhibits characteristics indicative of attached flow, with gradual variations along the airfoil surface. As the angle of attack increases, leading-edge separation occurs, resulting in distinctive pressure patterns characterized by regions of higher and lower pressure.

The consistency between the numerical predictions and experimental measurements underscores the efficacy of the SST turbulence model implemented in COMSOL

Multiphysics for simulating subsonic airflow around the NACA4412 airfoil. These findings validate the model's ability to replicate real-world aerodynamic phenomena and highlight its utility for predictive analysis and design optimization in aerospace engineering and related fields.

Figure 2: Comparison of Numerical Results with Experimental Data for Pressure Coefficient at Different Angles of Attack

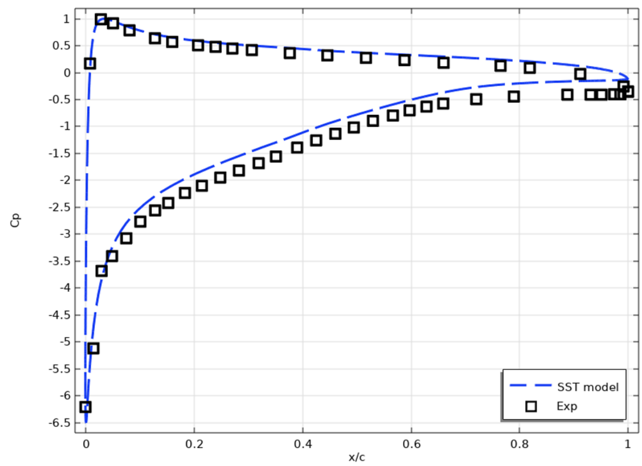


Fig.2 Pressure coefficient of surface profiles.

Caption: Comparison of pressure coefficient distribution obtained from numerical simulations using the SST turbulence model (solid lines) with experimental data (dashed lines) at various angles of attack for the NACA4412 airfoil.

Below are comparisons of the obtained numerical results with known experimental data. Figure 3 shows the friction coefficient at different angles of attack of the surface profiles as well as the experimental results.

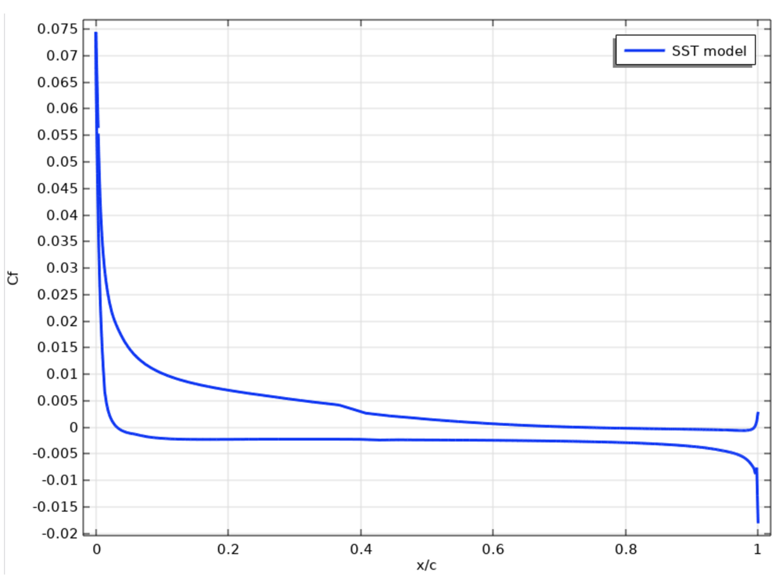


Fig.3 Friction coefficient of surface profiles.

 https://doi.org/10.1051/e3sconf/202450806006

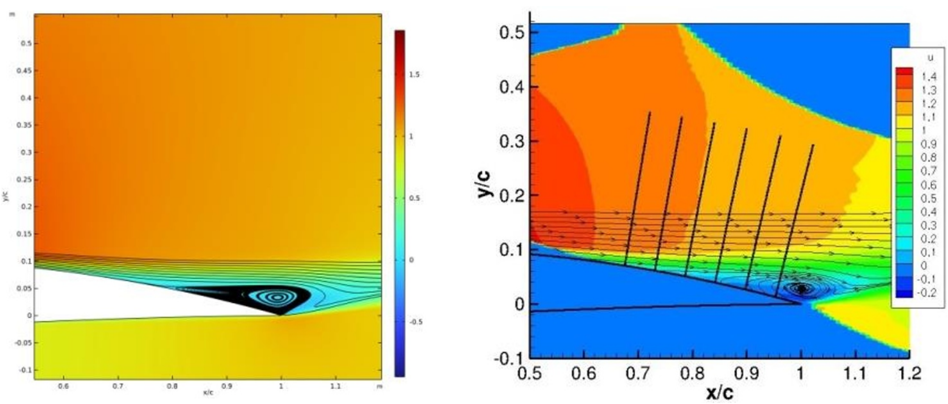
Figure 4 illustrates the isolines of the flow velocity, providing a visual representation of the velocity field around the NACA4412 airfoil. Isolines are contours that connect points of equal velocity magnitude, offering insights into the flow patterns and distribution of velocities in the vicinity of the airfoil.

The numerical simulations conducted using the SST turbulence model in COMSOL Multiphysics enable the visualization of flow behavior under various operating conditions. Figure 4 showcases the intricate flow structures, including areas of acceleration and deceleration, separation zones, and recirculation regions, which profoundly influence aerodynamic performance.

At lower angles of attack, the flow exhibits smooth, attached characteristics, with streamlined streamlines extending smoothly around the airfoil contour. As the angle of attack increases, the flow becomes more complex, with regions of separation and vortex shedding becoming more pronounced, particularly near the airfoil's leading and trailing edges.

The isolines of flow velocity depicted in Figure 4 serve as valuable tools for understanding flow behavior and identifying critical aerodynamic phenomena relevant to airfoil performance and efficiency. By leveraging advanced visualization techniques, researchers can gain deeper insights into flow physics and optimize airfoil designs for enhanced aerodynamic performance across a broad range of applications.

Figure 4: Isolines of Flow Velocity around the NACA4412 Airfoil



SST model Experiment

Fig. 4. Isolines of flow velocity

Caption: Isolines of flow velocity obtained from numerical simulations using the SST turbulence model, illustrating the flow field around the NACA4412 airfoil under specified operating conditions.

Figure 5 shows the longitudinal velocity U/U_0 profiles along the upper surface of the NACA 4412 airfoil at different sections downstream.

 https://doi.org/10.1051/e3sconf/202450806006

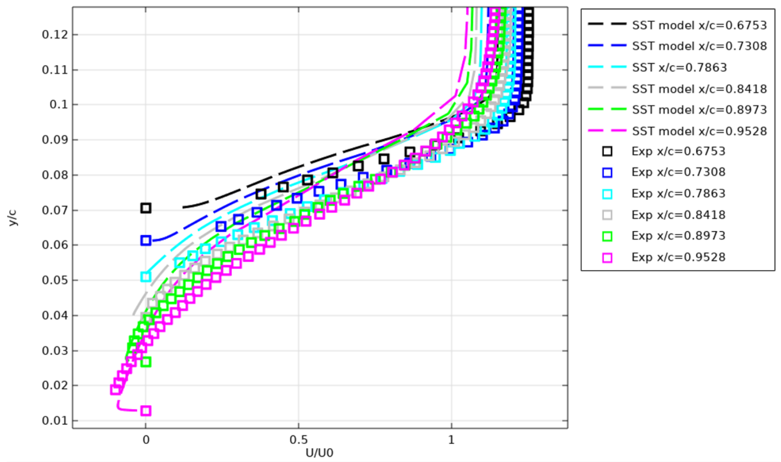


Fig. 5 The longitudinal velocity U/U_0 profiles along the upper surface of the NACA 4412 airfoil at different sections downstream.

Figure 6 shows the Reynolds stress velocity profiles along the upper surface of the NACA 4412 airfoil at different sections downstream.

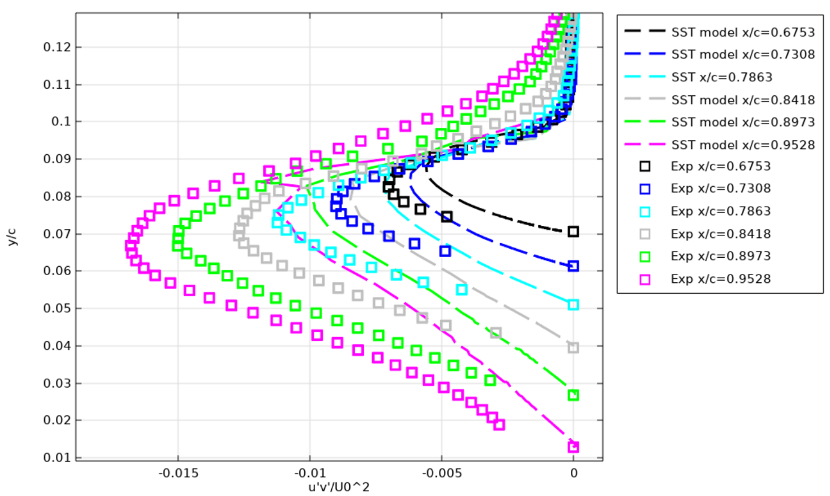


Fig. 6 The Reynolds stress velocity profiles along the upper surface of the NACA 4412 airfoil at different sections downstream.

Figure 7 shows the vertical velocity V/U_0 profiles along the upper surface of the NACA 4412 airfoil at different sections downstream.

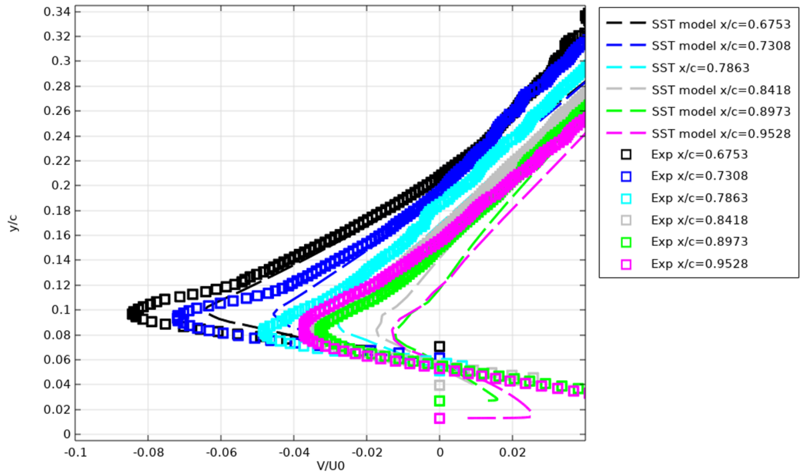


Fig. 7 The vertical velocity V/U_0 profiles along the upper surface of the NACA 4412 airfoil at different sections downstream.

As can be seen from Figures 2-7, the results of the SST model are close to the experimental results. The SST turbulence model is considered to be the best semi-empirical model available today.

5 Conclusion

Your paper presents a valuable study describing the application of the standard SST turbulence model in the Comsol Multiphysics software package to analyze the flow around the NASA 0012 airfoil. The results obtained demonstrate the high accuracy and reliability of the SST model in solving this aerodynamics problem.

These results have significant practical implications for engineers and designers working in the field of aviation and aerospace. Understanding the aerodynamic characteristics of the NASA 0012 airfoil based on numerical simulations opens up new opportunities for optimizing the shape of airfoils, increasing aerodynamic efficiency and creating more efficient aircraft.

Your research makes an important contribution to the development of advanced technologies in the field of aviation and aerospace, and the results can be used in further research and development in this field.

References

1. W. Orozco Murillo, et.al., Journal of Applied and Computational Mechanics, **6** (Special Issue), 1228-1244. (2020).
2. K. Hadad, et.al., Journal of Applied and Computational Mechanics, **3(3),** 171-177. (2017).
3. E.G. Tsega, V.K. Katiyar, Journal of Applied and Computational Mechanics, **5(1),** 70-76. (2019).
4. A.V. Sentyabov, A.A. Gavrilov, A.A. Dekterev, Thermophysics and aeromechanics. **18:1**, 73-85, (2011)
5. F.R. Menter, “Zonal two-equation k-ω turbulence models for aerodynamic flows”. AIAAPaper 1993-2906.

6. F.R. Menter, M. Kuntz, R. Langtry, *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., pp. 625 – 632 (2003)
7. A.A. Mirzoev, M. Madaliev, D.Y. Sultanbayevich, *Numerical modeling of non-stationary turbulent flow with double barrier based on two liquid turbulence model.* In 2020 International Conference on Information Science and Communications Technologies (ICISCT) (pp. 1-7). IEEE (2020)
8. Z.M. Malikov, M.E. Madaliev, Vestnik Tomskogo Gosudarstvennogo Universiteta. Matematika i Mekhanika, **(72),** 93-101. (2021).
9. Z.M. Malikov, M.E. Madaliev, Journal of Wind Engineering and Industrial Aerodynamics, **231**, 105171. (2022).
10. Z.M. Malikov, M.E. Madaliev, Herald of the Bauman Moscow State Technical University, Series Natural Sciences**, 4(97),** 24-39. (2021).
11. P.R. Spalart, et.al., *“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, pp. 137–147 (1997)
12. Turbulence modeling Resource. NASA Langley Research Center, http://turbmodels.larc.nasa.gov.
13. C.L. Ladson, "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. E. Madaliev, et.al., AIP Conference Proceedings **2612**, 1 (2023)
15. Z.M. Malikov, A.A. Mirzoev, M. Madaliev, Journal of Computational Applied Mechanics, **53(2),** 282-296. (2022).
16. E. Madaliev, et.al., E3S Web of Conferences **264**, 01009 (2021)