Journal of Construction and Engineering Technology
Journal homepage: www.journal.fstu.uz

NUMERICAL SIMULATION OF AERODYNAMIC FLOW AROUND A BUILDING USING OPENFOAM SOFTWARE

Kurbonova Nilufar

[1]Affiliation 1; Fergana State Technical University, Uzbekistan
*Correspondence: Madaliev.me2019@mail.ru

Abstract

This article examines one of the pressing issues of modern urban development – the impact of high-speed air flows on buildings and surrounding infrastructure. The impact of wind loads plays an important role in the design of high-rise buildings, bridges and other construction projects, as it directly affects their stability, comfort and safety. To solve this problem, OpenFOAM software, an open platform for numerical modeling of continuum mechanics problems, was used. In this work, the finite volume method was used to discretize the transport equations and calculate the RANS (Reynolds-Averaged Navier-Stokes) equations. To ensure the accuracy of the calculations, the SIMPLE algorithm was used, which allows you to relate the velocities and pressures at each step of the iterative process. In addition, the SST (Shear Stress Transport) turbulence model was used to model turbulence in flows. This model has proven itself in calculating problems related to aerodynamics, as it combines the advantages of the k-ω and k-ε models, which allows to effectively describe both laminar and turbulent flow regions. The study simulated the flow around a building of various shapes and heights, which made it possible to identify key features of the distribution of speeds, pressures and turbulent characteristics in the vicinity of the building. The calculation results make it possible to assess the potential risks of structural failure, identify critical areas and optimize architectural solutions to reduce the negative impact of wind loads. Thus, the use of OpenFOAM and modern turbulence models, such as SST, provides a powerful tool for analyzing and predicting the aerodynamic characteristics of building objects, which is an important step in the development of sustainable urban development technologies.

Keywords: Navier-Stokes equations, SST model, OpenFOAM, RANS, SIMPLE.

1. Introduction

Modern urban planning faces many complex challenges related to ensuring the sustainability, reliability and safety of buildings, especially under high wind loads. The influence of air flows on buildings and adjacent infrastructure plays a key role in the design of building projects, from architectural solutions to engineering analysis of structures. Given the growing density of urbanization and the increase in the height of buildings in large cities, wind loads are becoming one of the central issues that must be taken into account to prevent the destruction of structures and ensure environmental comfort. High-speed air flows have different effects on buildings depending on their shape, height and location. This may include the formation of high-pressure zones on the windward side, turbulent zones on the leeward side and even strong vortices that can affect the stability of structures and create uncomfortable conditions for people nearby. Incorrect design of buildings from the point of view of aerodynamics can lead to serious consequences, including the destruction of facades, excessive vibrations of structures, a decrease in the energy efficiency of buildings, as well as unfavorable conditions for pedestrians, such as the formation of strong air flows near the base of the building. Traditional methods of studying aerodynamic processes, such as wind tunnel tests, although providing high accuracy, have a number of limitations. They require significant financial costs, large time resources and complex preparation of experimental stands. In addition, physical experiments have their limitations in reproducing some complex scenarios, such as the interaction of air flows between several buildings or taking into account small structural details. In such conditions, numerical modeling methods have become an important alternative and complement to experimental studies. At the current stage of development of computing technologies,

Received: 20 January 2025
Revised: 22 January 2025
Accepted: 2 October 2025
https://doi.org/10.24412/2181-4473-2025.3.202

numerical methods based on solving the Navier-Stokes equations have become widespread. One of the most powerful tools for such calculations is the OpenFOAM (Open Field Operation and Manipulation) software. OpenFOAM is a flexible and universal open-source platform that allows you to solve a wide range of problems in continuum mechanics, including problems of air flow around buildings. The software provides the user with the ability to customize and modify calculation models, which makes it popular among both researchers and practicing engineers [1–4]. The finite volume method (FVM) was used to perform calculations in this study, which allows discretizing the equations of fluid and gas motion, ensuring high accuracy and stability of results. The SST (Shear Stress Transport) turbulence model was used to close the RANS (Reynolds-Averaged Navier-Stokes) equations. This model combines the advantages of two widely used turbulence models: $k-\omega$ for regions with a high velocity gradient and $k-\varepsilon$ for fully developed turbulent flow. This approach allows taking into account complex flow behavior, including transition zones between laminar and turbulent regions. The SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm was used to link the velocity field and the pressure field, ensuring reliable convergence of the calculation process [5–7]. As part of the work, numerical simulation of air flow past a building was carried out taking into account various geometric characteristics and flow conditions. This allowed us to study in detail the distribution of pressure and speeds around the building, identify high-stress zones and determine the most critical areas exposed to wind loads [8–10].

Thus, the main goal of the study is to develop and implement an approach to numerical modeling of aerodynamic flow around buildings using OpenFOAM. The results of the work can be useful in designing buildings with improved aerodynamic characteristics, which will reduce wind loads, reduce energy consumption and increase the overall resistance of buildings to external influences. In addition, this study is an important step towards the introduction of numerical analysis methods in modern design of urban development objects, which contributes to the development of sustainable and safe construction technologies [11–17].

2. Physical and mathematical formulation of the problem

The dimensions of the fluid domain were determined according to the recommendations for numerical modeling to ensure a complete representation of the flow structures. This approach avoids the influence of boundary conditions on the behavior of the air flow. For all considered cases, the dimensions of the computational domain were height (h = 140 m), width (a = 280 m), and length (L = 350 m), which is clearly shown in Figure 1.

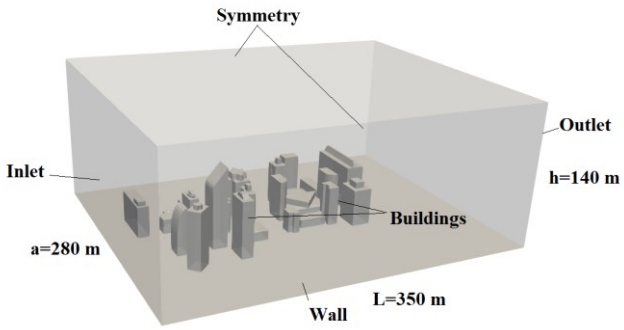


Fig. 1. Dimensions of the computational domain.

The following conditions are set at the inlet boundary of the computational domain: the air flow velocity is 20 m/s, the turbulence intensity is set at 5%, and the length of the turbulence scale parameter is 10 mm. The inlet conditions were chosen in accordance with recommendations from [21], which are widely used in building aerodynamics simulations. The Reynolds number calculated based on the height of the structure under study (h) and the specified flow velocity is $Re_h = 2.0 \times 10^8$. The configuration of the computational grid is shown in Figure 2.

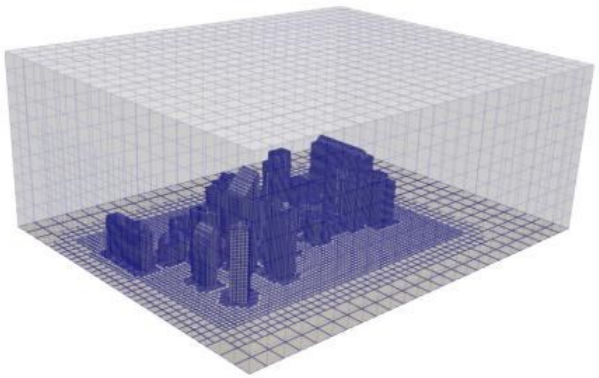


Fig. 2. Non-structural computational grid.

Although the buildings have different shapes and sizes, the governing equations (RANS) and boundary conditions are the same for the whole computational domain. Therefore, a single simulation provides a general solution, while the post-processing stage allows analyzing local aerodynamic characteristics (velocity, pressure, turbulence) separately for each building.

2.1 Mathematical model

The Reynolds-averaged Navier–Stokes (RANS) equations were used to analyze the aerodynamic

characteristics of buildings. This system is the basis for the mathematical description of the dynamics of an incompressible fluid and is a set of differential equations describing the spatial and temporal changes in velocity and pressure in a flow [14–17].

A feature of the averaged form of the Navier–Stokes equations is that it takes into account the influence of turbulent flows by introducing additional terms characterizing turbulent stresses. These terms reflect the effects caused by the interaction of fluctuating velocity components, which allows describing complex flow structures that arise around objects. In particular, this concerns such phenomena as the formation of turbulence zones, recirculation, and vortex structures, which play a key role in the aerodynamics of buildings.

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

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

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

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

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

These equations together form the basis for numerical modeling of complex turbulent flows around buildings, which is an important step in studying aerodynamic characteristics and designing sustainable architectural solutions.

2.2 Turbulence models.

The Reynolds-averaged Navier–Stokes (RANS) equations are used to numerically simulate turbulent flows around complex-shaped objects. These equations include additional terms describing turbulent stresses that arise due to velocity fluctuations. To take them into account correctly, it is necessary to use a turbulence model that ensures the closure of the system of equations. In this study, the Shear Stress Transport (SST) turbulence model [18–20] was chosen, which combines the advantages of two popular approaches: the *k–ω* and *k–ε* models. This approach allows for efficient flow simulation both in the boundary layer regions, where precise detailing of velocity gradients is important, and in flow separation zones, characterized by intense turbulence. Due to its versatility and high accuracy, the SST model is an optimal choice for studying the aerodynamic characteristics of objects with complex geometry, including buildings and infrastructure structures.

$$\begin{cases} (\mathbf{U}\cdot\nabla)k = \nabla\left[(\nu + \sigma_k \nu_t)\nabla k\right] + P - \beta^* \omega k, \\ (\mathbf{U}\cdot\nabla)\omega = \nabla\left[(\nu + \sigma_\omega \nu_t)\nabla \omega\right] + \frac{\gamma}{\nu_t} P - \beta\omega^2 + 2(1-F_1)\frac{\sigma_{\omega 2}}{\omega}\nabla\omega\nabla k. \end{cases} \tag{3}$$

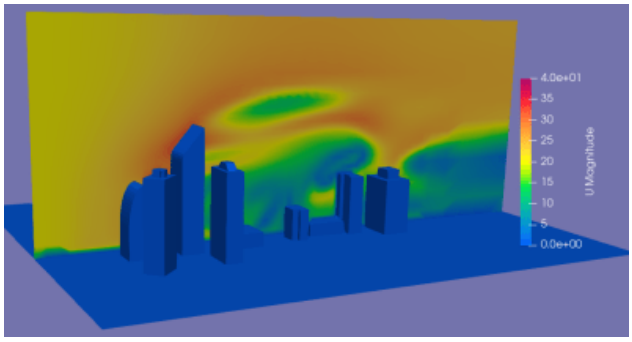
Here k is the specific turbulent kinetic energy ($m^2\ s^{-2}$), ω is the specific rate of turbulent dissipation (s^{-1}). Other values are presented in [18–19].

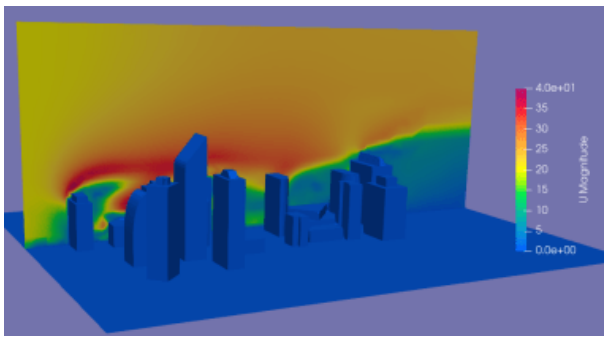
3 Solution method

For numerical modeling of aerodynamic characteristics of buildings in the OpenFOAM software package, the control volume method was applied, which is one of the most common and effective methods in computational fluid dynamics. This method underlies modern approaches to solving problems of continuum mechanics due to its versatility, accuracy and ability to adapt to various geometric and physical conditions.

4. Results and discussion

Figure 3 shows the velocity isolines displayed in three vertical planes of geometry. These visualizations provide a clear analysis of the flow velocity distribution and the ability to determine the characteristic features of its behavior in the flow area, including the zones of velocity change.





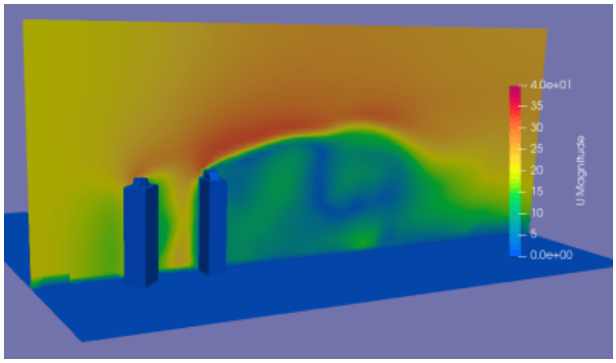
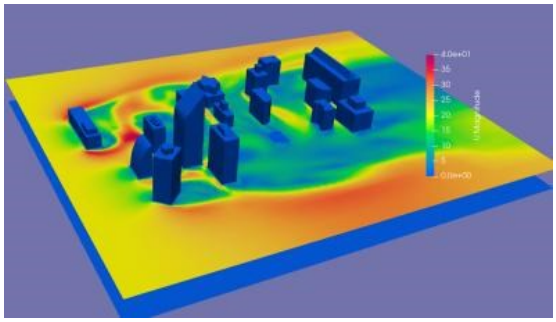
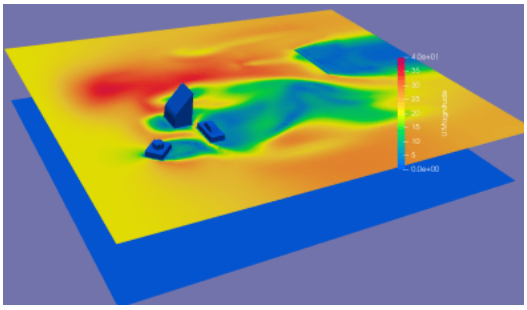


Fig. 3 Velocity isolines displayed in three vertical planes of geometry

Figure 4 shows the velocity contours plotted in three horizontal planes of geometry.





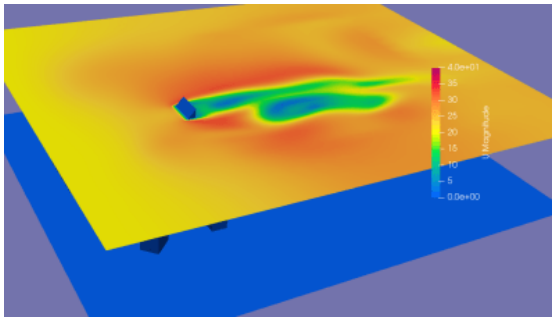
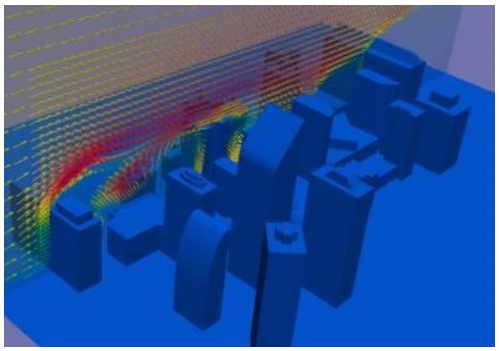
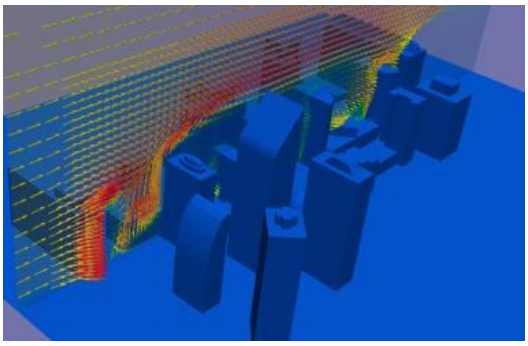
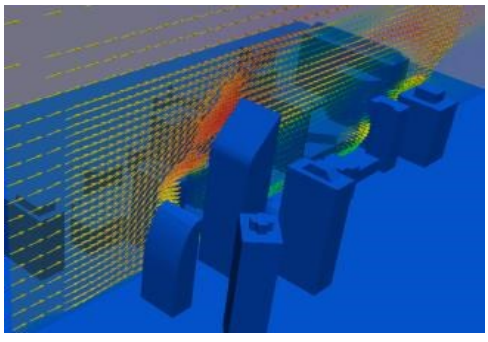


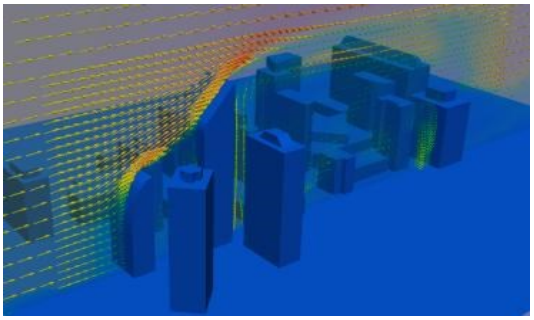
Fig. 4. Velocity isolines displayed in three horizontal planes of geometry

Figure 5 shows velocity vectors that clearly demonstrate the direction and magnitude of the flow in the study area.









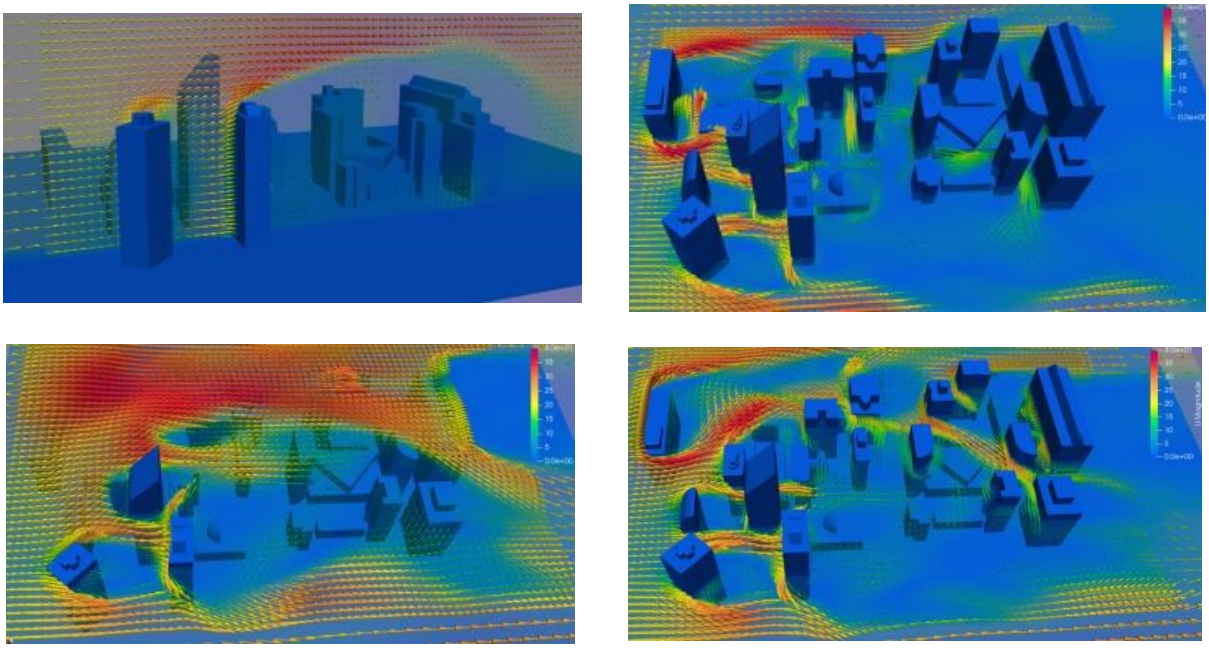
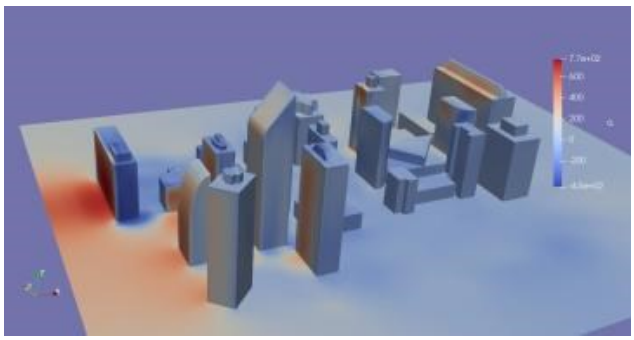


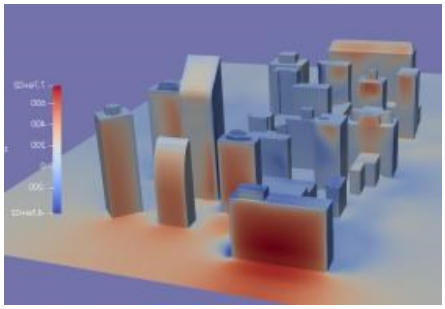
Fig. 5. Velocity vectors that clearly demonstrate the direction and magnitude of the flow in the area under study.

This visualization is an important tool for analyzing aerodynamic characteristics, as it allows one to obtain a complete picture of the flow behavior. From the presented data, it is possible to determine key features of air movement, including the main flow directions, its intensity, and to identify areas with recirculation, turbulence, and significant velocity changes. In addition, the vector field allows one to study in detail the interaction of the air flow with the geometric surfaces, including areas with increased velocity and areas where the flow slows down or deflects. Such data are of particular value for understanding the aerodynamic loads on an object and optimizing its shape. Visualization of vector fields also allows one to more effectively interpret the numerical results obtained during the simulation and to evaluate the correctness of the selected turbulence models [20].

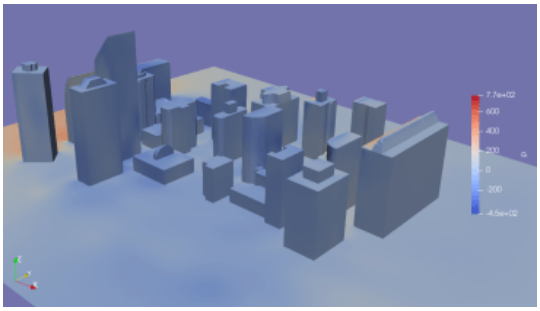
Figure 6 shows the pressure isolines on the surface of the building walls.



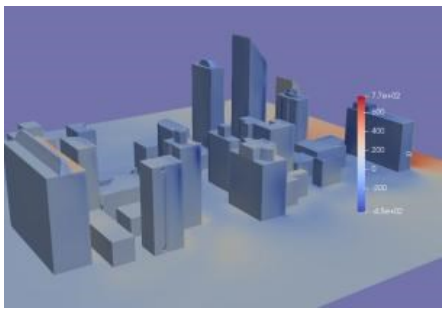
a)



b)



c)



d)

Fig. 6. Pressure contours in the walls of the building.

This visualization allows you to determine the pressure distribution over the entire surface of the object, which is a key aspect for analyzing aerodynamic loads. Isolines show areas with maximum and minimum pressure values, and also allow you to identify areas where critical loads may occur that can affect the stability of the structure. Isoline data also allows you to analyze the effect of air flow on various sections of the walls, including areas of high pressure gradients, which are most often located in the area of flow around corners and edges of the building. Such information is useful for optimizing architectural solutions and increasing the overall resistance of the building to wind effects.

5. Conclusions

In conclusion, the numerical modeling of the aerodynamic characteristics of the building using the OpenFOAM software package and the control volume method allowed us to obtain detailed results reflecting the behavior of the air flow around the object. The use of the Reynolds-averaged Navier-Stokes (RANS) equations and the Shear Stress Transport (SST) turbulence model ensured high accuracy in predicting the distribution of speeds, pressure and turbulent effect zones. The study identified key zones with high aerodynamic loads, which is of great importance for further improvement of design solutions and increasing the resistance of buildings to wind effects. The vector fields of velocity and pressure isolines obtained in the modeling process allowed us to better understand the features of air flows and their impact on the surfaces of objects. These studies can be used to optimize the architectural forms of buildings, improve their aerodynamic characteristics and develop new solutions aimed at improving the energy efficiency and safety of buildings in extreme weather conditions. The results of the work also confirm the effectiveness of using the control volume method and the SST turbulence model for analyzing complex flows, which makes this approach suitable for wide application in the field of computational aerodynamics.

References

1. Piechna, J. A Review of Active Aerodynamic Systems for Road Vehicles. Energies 2021, 14, 7887.
2. Szudarek, M.; Piechna, A.; Piechna, J. Feasibility Study of a Fan-Driven Device Generating Downforce for Road Cars. Energies 2022, 15, 5549.
3. Szudarek, M.; Kamieniecki, K.; Tudruj, S.; Piechna, J. Towards Balanced Aerodynamic Axle Loading of a Car with Covered Wheels-Inflatable Splitter. Energies 2022, 15, 5543.
4. Piechna, J.R.; Kurec, K.; Broniszewski, J.; Remer, M.; Piechna, A.; Kamieniecki, K.; Bibik, P. Influence of the Car Movable Aerodynamic Elements on Fast Road Car Cornering. Energies 2022, 15, 689.
5. Broniszewski, J.; Piechna, J.R. Fluid-Structure Interaction Analysis of a Competitive Car during Brake-in-Turn Manoeuvre. Energies 2022, 15, 2917.
6. Broniszewski, J.; Piechna, J. A fully coupled analysis of unsteady aerodynamics impact on vehicle dynamics during braking. Eng.Appl. Comput. Fluid Mech. 2019, 13, 623–641.
7. Kurec, K.; Kamieniecki, K.; Piechna, J. Influence of Different Plates Arrangements on the Car Body. Energies 2022, 15, 619.
8. Palanivendhan, M.; Nagpal, D.; Rao, D.; Philip, J.; Ganapathi, M. Design and analysis of an aerodynamic kit for a two wheeled race motorcycle. Mater. Today Proc. 2021, 45, 7239-7246.
9. Angeletti, M.; Sclafani, L.; Bella, G.; Ubertini, S. The Role of CFD on the Aerodynamic Investigation of Motorcycles; SAE Technical Papers; SAE International: Warrendale, PA, USA, 2003.
10. Bella, G.; Ubertini, S.; Desideri, U. Experimental and Computational Analysis of the Aerodynamic Performances of a Maxi-Scooter; SAE Technical Papers; SAE International: Warrendale, PA, USA, 2003.
11. Malikov, Z. M., Mirzoev, A. A., & Madaliev, M. (2022). Numerical simulation of the mixing layer problem based on a new two-fluid turbulence model. Journal of Computational Applied Mechanics, 53(2), 282-296.
12. Malikov, Z. M., Madaliev, M. E., Navruzov, D. P., & Adilov, K. (2022, October). Numerical study of an axisymmetric jet based on a new two-fluid turbulence model. In AIP Conference Proceedings (Vol. 2637, No. 1). AIP Publishing.
13. Madaliev, M., Yunusaliev, E., Usmanov, A., Usmonova, N., & Muxammadyoqubov, K. (2023). Numerical study of flow around flat plate using higher-order accuracy scheme. In E3S Web of Conferences (Vol. 365, p. 01011). EDP Sciences.
14. Madaliev, E., Madaliev, M., Raxmankulov, S., & Raxmonkulova, S. (2023). Turbulent mixing of two plane flows based on the SST turbulence model. In E3S Web of Conferences (Vol. 452, p. 02012). EDP Sciences.
15. Malikov, Z. M., & Madaliev, M. E. (2025). Modeling of Turbulent Two-Phase Flow based on a Two-Fluid Approach. Journal of Applied and Computational Mechanics, 11(1), 204-222.
16. Madaliev, M., Yunusaliev, E., Abdulkhaev, Z., Otakulov, B., Yusupov, S., Ergashev, I., & Tohirov, I. (2024). Comparison of numerical results of linear and nonlinear turbulence models based on the rans approach. In E3S Web of Conferences (Vol. 587, p. 01003). EDP Sciences.
17. 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.
18. Menter F. R."Zonal two-equation k-ω turbulence models for aerodynamic flows". AIAAPaper 1993-2906.

19. 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.
20. Kurbonova, N. (2024). Aerodynamic Study of the Characteristics of Pedestrian Overstands under Wind Load. Journal of Construction and Engineering Technology, 7-13.
21. Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., & Shirasawa, T. (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. Journal of wind engineering and industrial aerodynamics, 96(10-11), 1749-1761.