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

ANALYSIS OF CFD MODELING OF THE KARKIDON RESERVOIR SPILLWAY OPERATION USING THE SST TURBULENT MODEL

Shuhratjon O‘tbosarov[1*], Oybek Muminov[1]

[1] Fergana polytechnic institute, Fergana, Uzbekistan
*Correspondence: sh.utbosarov@ferpi.uz

Abstract

In this article, mathematical modeling of the drainage system of the Karkidon reservoir was carried out in order to analyze the hydrodynamic characteristics of the flow in the spillway. The study examined the behavior of water masses in the outlet part of the reservoir using RANS equations to describe the overall flow movement and SST equations to model turbulent phenomena. The calculations made it possible to estimate the distribution of velocities and pressure in various sections of the water outlet, as well as to identify zones of possible formation of vortices and turbulent structures. The results obtained contribute to a deeper understanding of the dynamics of water flows in engineering structures of this type and can be used to optimize their design and operation.

Keywords: Navier–Stokes equations, SST model, Comsol Multiphysics, Karkidon reservoirs.

1. Introduction

Water resource management is one of the key challenges in modern conditions, especially in regions where water supply plays a critical role in supporting the livelihoods of the population and industry. Water is an essential resource for most sectors of the economy, including agriculture, energy industry and municipal needs. With increasing water consumption and a changing climate leading to more frequent and intense droughts and floods, the need for effective water management has become even more urgent. Reservoirs, as large hydraulic structures, serve as important elements of water supply, flow regulation and flood prevention. They play a critical role in providing a stable and reliable water supply to human settlements, as well as maintaining ecological balance in aquatic ecosystems. In addition, reservoirs are used for hydropower, recreation and fisheries, making them multifunctional assets that require careful management and monitoring. The effective operation of reservoirs directly depends on the reliability and functionality of their components, such as spillways and drainage systems. Spillways are designed to safely drain excess water from the reservoir, preventing possible destruction of the dam and flooding of adjacent areas. They must be designed and operated to handle maximum design volumes of water while minimizing risks to infrastructure and the environment [1–5].

Reservoir drainage systems also play an important role in maintaining the structural integrity of hydraulic structures. They prevent excess pressure from building up at the base of the dam, reducing the risk of seepage and erosion. Therefore, the reliability and efficiency of drainage systems are critical factors for the overall safety and durability of reservoirs [6–9].

Modern numerical modeling techniques such as Computational Fluid Dynamics (CFD) provide powerful tools for detailed analysis and visualization of fluid flows in complex engineering structures. CFD modeling allows you to study the hydrodynamic processes occurring inside spillways and drainage systems with a high degree of accuracy. This is especially important for identifying areas of turbulence, high speeds and pressures that can lead to structural failure or reduced efficiency [10–14].

The use of turbulence models such as SST (Shear Stress Transport) allows the effects of turbulent mixing to be taken into account and more accurate predictions of flow characteristics to be obtained. These models are widely used in engineering applications to optimize the design and improve the reliability of hydraulic structures.

In this work, mathematical modeling of the drainage system of the Karkidon reservoir was carried

Received: 20 November 2024
Revised: 17 March 2024
Accepted: 20 June 2025

https://doi.org/10.24412/2181-4473-2025.3.101

out using CFD. The main attention is paid to the analysis of the movement of water in the spillway and the study of turbulent structures that arise during the operation of the spillway. The Reynolds equations (RANS) were used to describe the flow motion, and the SST model was used to simulate turbulence. The purpose of this study is to evaluate the performance of the weir and identify potential problem areas where turbulent structures and increased loads may form. The results obtained can be used to optimize the design of the weir and improve its reliability, which ultimately contributes to improving water resource management in the region [15–18].

Thus, the research presented in this article contributes to the development of methods for managing and optimizing the operation of reservoirs, ensuring their reliable and safe operation in a changing climate and increasing demands on water resources [19-25].

2. Physical and mathematical formulation of the problem

Water comes out of a pipe with a diameter of 2000 mm under a pressure of 5 atm (Fig. 1. A cone valve is installed at the outlet, water enters the cone valve and exits into the air.

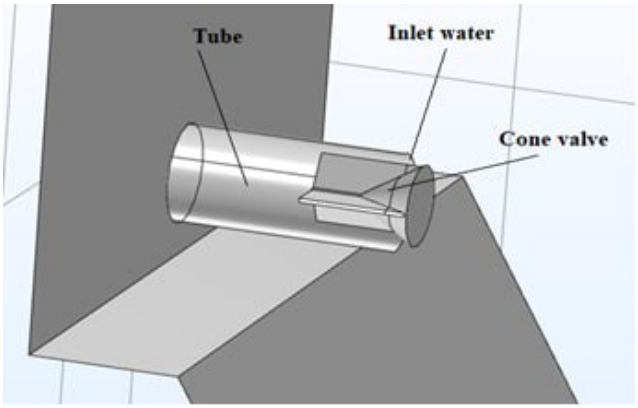
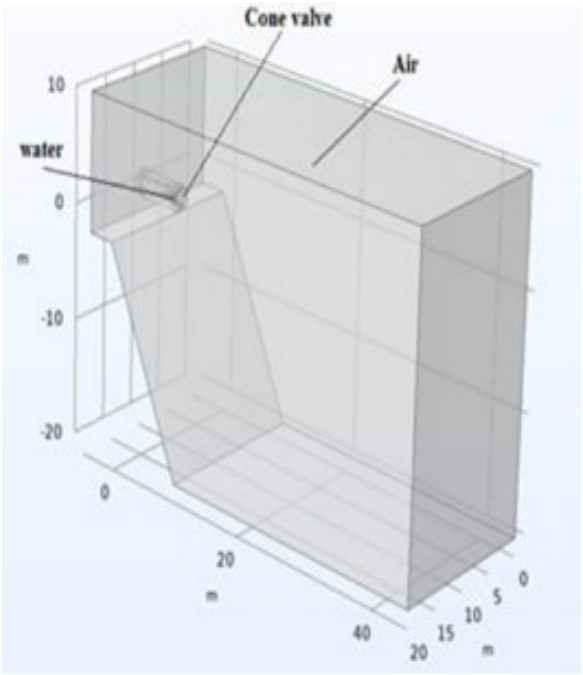
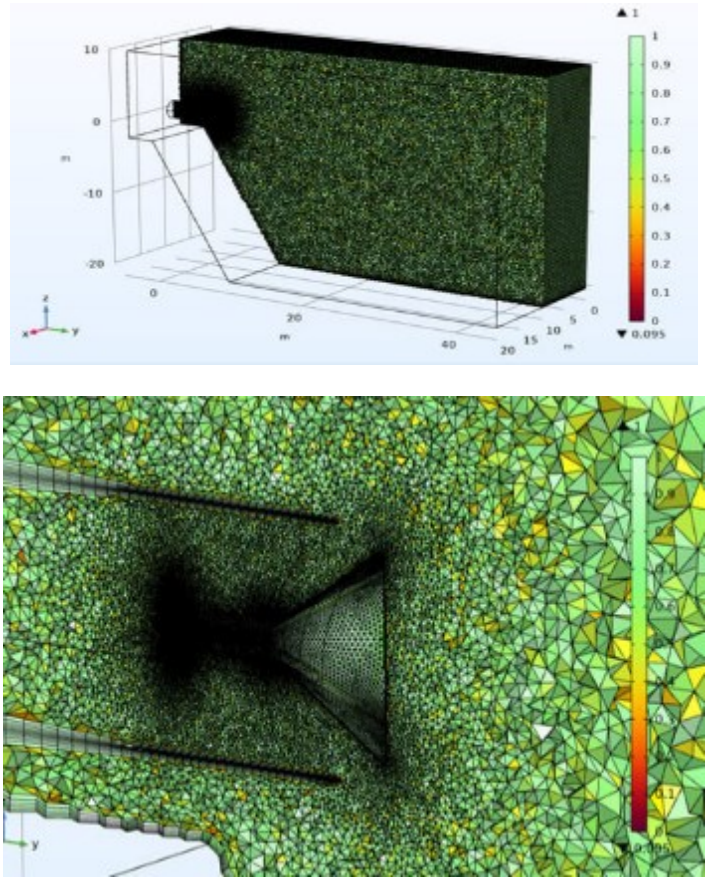


Fig.1. Gate input geometry.

The general geometry of the problem and the computational grid are shown in Figure 2, where all dimensions are indicated in meters. The problem used 8,979,132 non-structural meshes.





a) b)

Fig. 2. a) geometry of the outlet b) non-structural computational mesh

3. Mathematical model

To study this problem, the Reynolds-averaged Navier-Stokes equations were used. These equations serve as the basis for the mathematical description of the movement of an incompressible fluid and represent a system of differential equations that describe the change in speed and pressure in the fluid over time and in spatial coordinates [25].

The averaged Navier-Stokes equations take into account turbulent flows and are represented by the following system of equations:

Mass conservation equation (continuity equation), describing the law of conservation of mass in the computational domain:

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

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

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

where

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

The use of Reynolds-averaged Navier-Stokes equations allows one to take into account turbulent effects in the flow. These equations are solved by numerical methods, such as the finite element method, using specialized software packages such as Comsol Multiphysics. This allows you to obtain detailed data on the characteristics of the flow and its impact on the profile [15–24].

4. Turbulence models

The SST (Shear Stress Transport) model [26–29] is one of the most common and effective turbulence models used in the numerical modeling of aerodynamic flows. It combines the advantages of two turbulence models: the k-omega model, which is used near the wall to accurately describe velocity gradients in the boundary layer, and the k-epsilon model, used in the main part of the flow to model turbulent dissipation. This combination allows the SST model to provide more accurate and stable predictions of turbulent flow characteristics under different conditions.

$$\begin{cases} (\mathbf{U}\cdot\nabla)k = \nabla\left[(\nu + \sigma_k \nu_t)\nabla k\right] + P - \beta^* \omega k, \\ (\mathbf{U}\cdot\nabla)\omega = \nabla\left[(\nu + \sigma_\omega \nu_t)\nabla \omega\right] + \frac{\gamma}{\nu_t} P - \beta\omega^2 + 2(1-F_1)\frac{\sigma_{\omega 2}}{\omega}\nabla\omega\nabla k. \end{cases} \tag{4}$$

Here: k – specific turbulent kinetic energy ($m^2 c^{-2}$), ω – specific rate of turbulent dissipation (c^{-1}). Other values are presented in the works [26–29].

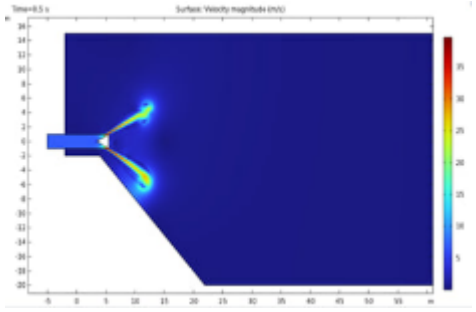
The SST model provides more accurate prediction of turbulent effects in a wide range of flow regimes, including both laminar and turbulent flows. It is especially useful when modeling free surfaces and other complex geometries where various flow regimes are present. The SST model takes into account both the influence of viscosity in the near layer of the boundary and the influence of energy dissipation in the distant layer of the boundary, which makes it possible to more accurately describe turbulent structures in the flow. This makes the SST model one of the preferred choices for engineers and researchers involved in numerical simulation of aerodynamic flows.

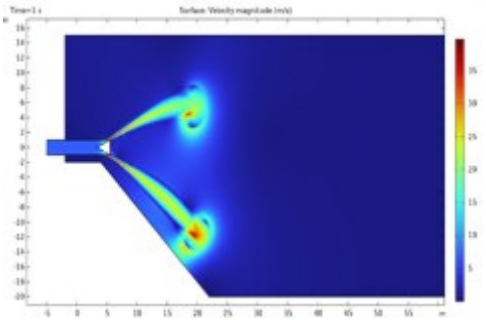
5. Solution method

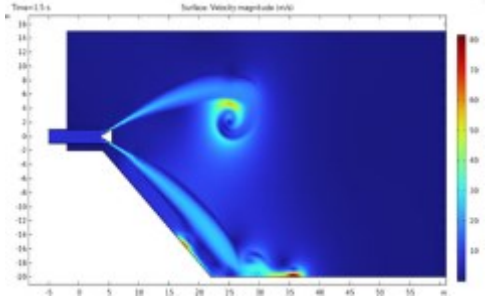
To analyze turbulent flows in our study, we used the SST turbulence model available in the Comsol Multiphysics software package. Comsol Multiphysics offers a wide range of solvers designed to simulate a variety of physical phenomena. These tools allow you to perform numerical simulations with high accuracy and efficiency, and provide the flexibility to adapt to different types of problems and geometries [30–40].

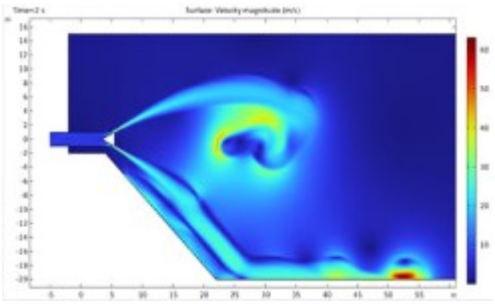
6. Results and discussion

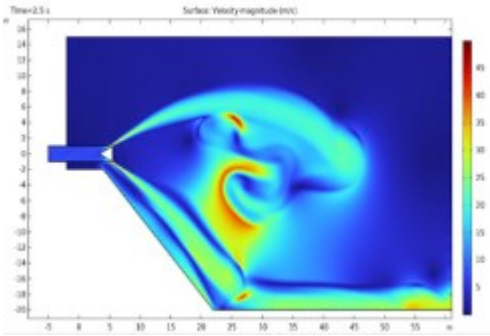
Figure 3 shows in detail the change in speed contours recorded every second. This graph allows you to observe the dynamics of the flow, demonstrating how the velocity distribution changes over time. Velocity contours show areas with different speed characteristics, allowing you to visualize flow movements and identify key areas of high and low speeds. This information is critical to understanding the behavior of flow under different conditions and to further analyze its interaction with surfaces and other elements of the system.

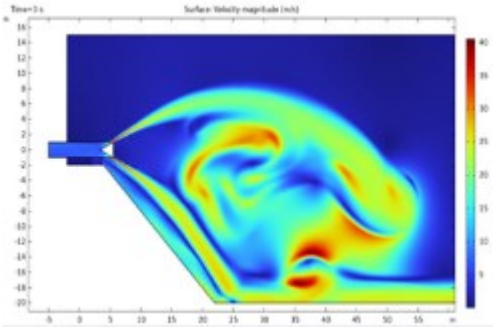

Time=0.5 s
Surface: Velocity magnitude (m/s)

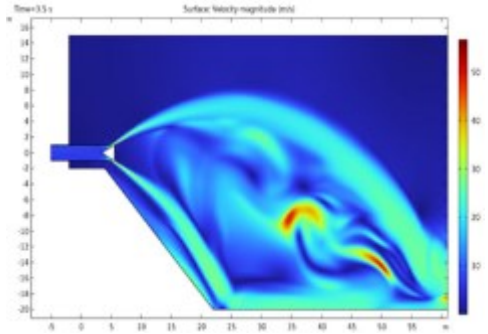

Time=1 s
Surface: Velocity magnitude (m/s)

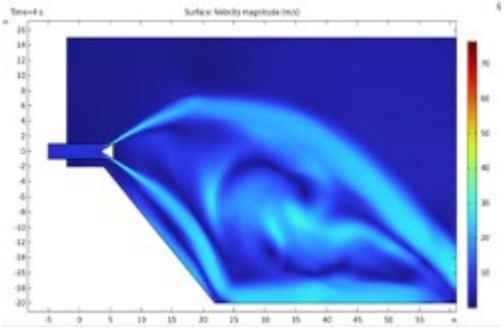

Time=1.5 s
Surface: Velocity magnitude (m/s)

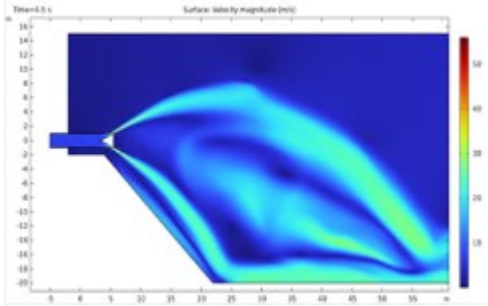

Time=2 s
Surface: Velocity magnitude (m/s)

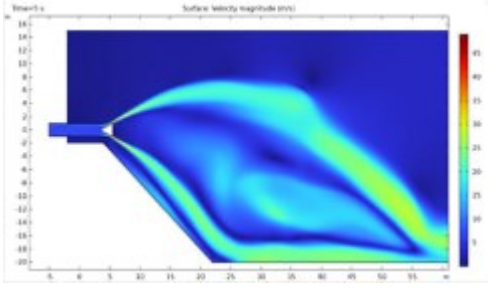

Time=2.5 s
Surface: Velocity magnitude (m/s)


Time=3 s
Surface: Velocity magnitude (m/s)


Time=3.5 s
Surface: Velocity magnitude (m/s)


Time=4 s
Surface: Velocity magnitude (m/s)


Time=4.5 s
Surface: Velocity magnitude (m/s)


Time=5 s
Surface: Velocity magnitude (m/s)

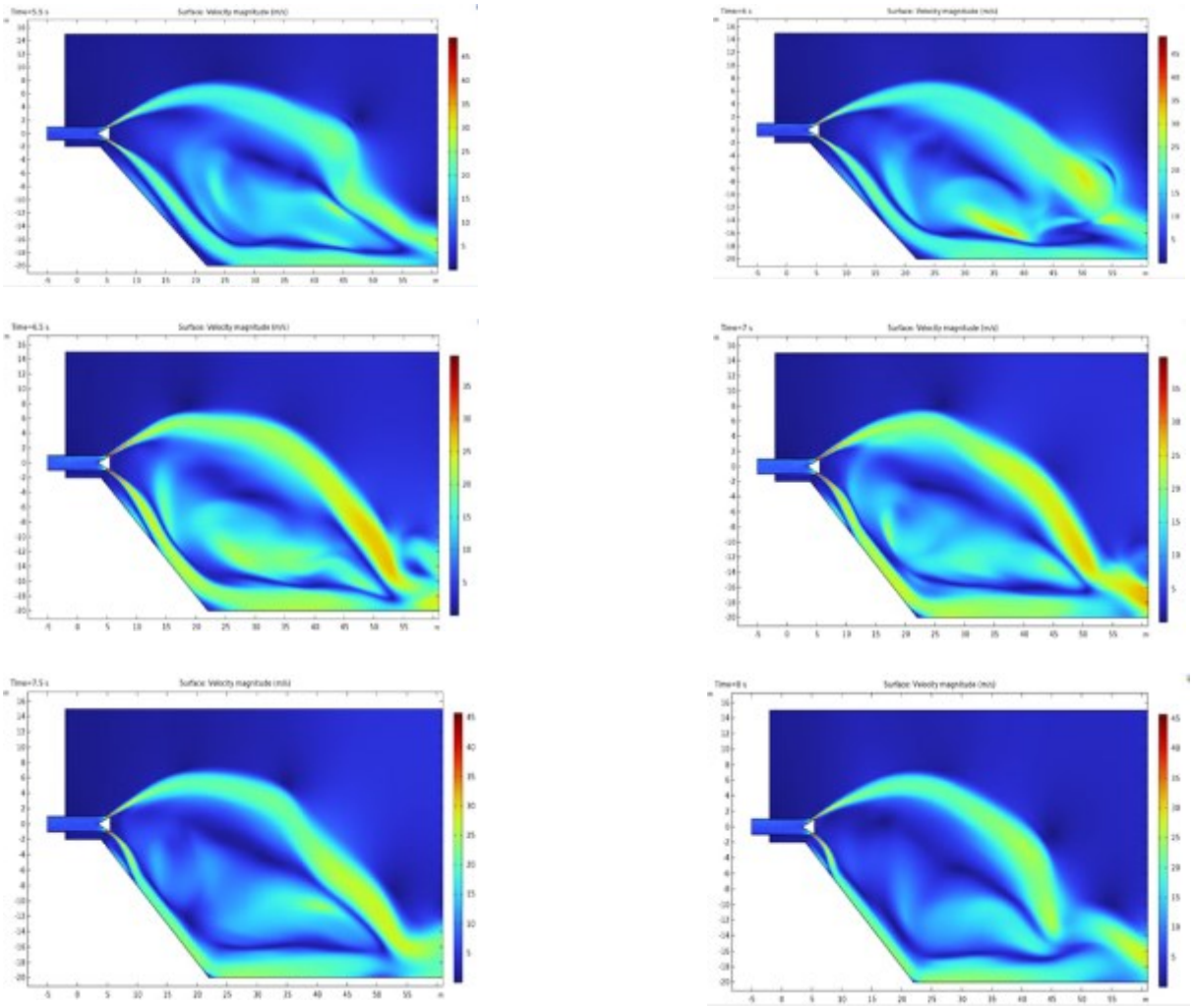


Fig. 3. The change in velocity contours is shown in detail

Figure 4 shows the changes in pressure exerted by the liquid on the valve. This graph illustrates how pressure is distributed across the valve surface and changes over time. Changes in pressure can be caused by various factors, such as changes in flow rate, turbulent effects, or variable conditions at the inlet and outlet of the system. Analysis of this data allows us to understand the load on the valve in various operating modes, which is important for its optimal design and reliable operation.

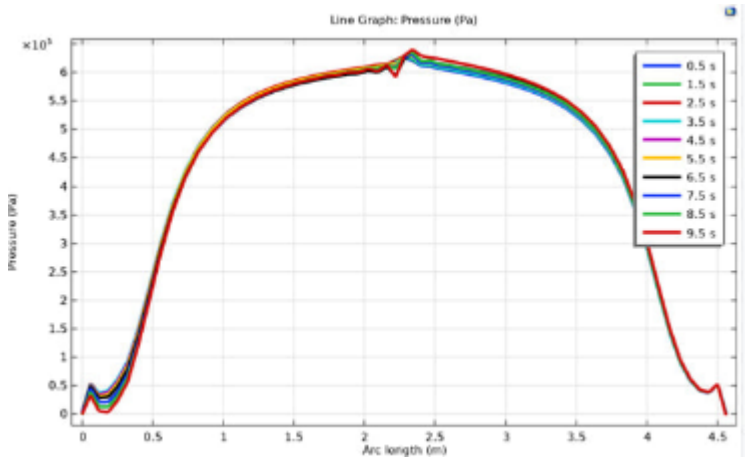


Fig. 4. Pressure changes demonstrated.

Figure 5 shows 5 sections of water at a distance of 10 meters at the outlet.

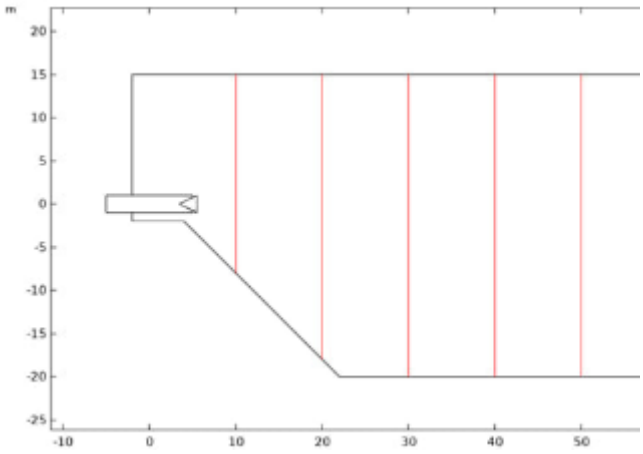
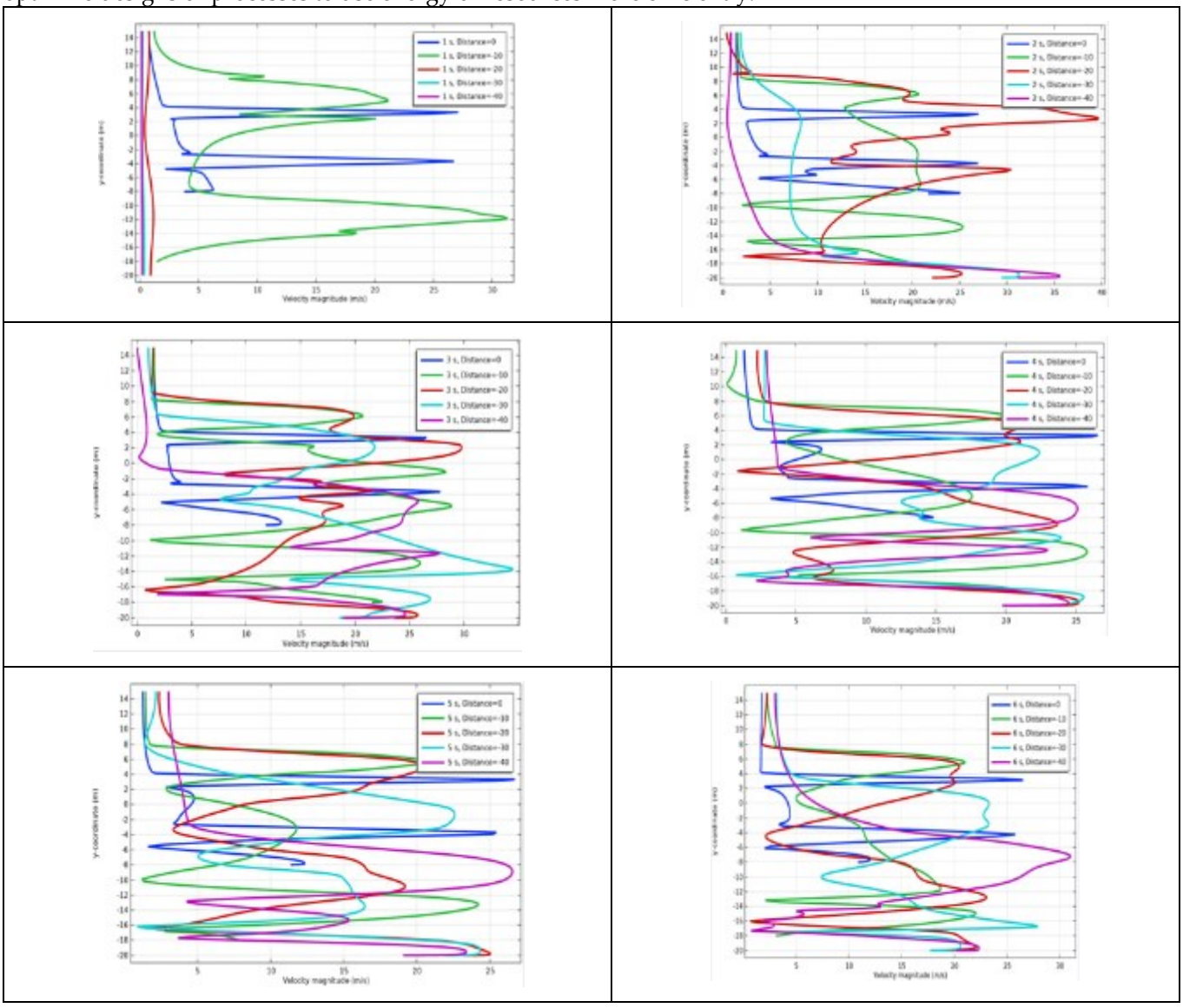


Fig. 5. Five sections of water at a distance of 10 meters at the outlet.

Figure 6 shows the speed changes in the five sections listed. This graph allows you to visually compare flow rates in different areas of the system or model. Analyzing velocity data in various sections helps understand the distribution and intensity of flow at various points in the system, identify possible areas of high or low velocity, and optimize designs or processes to use energy or resources more efficiently.



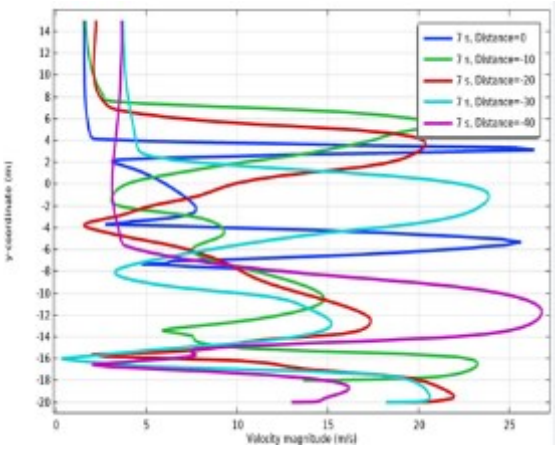


Fig. 6. Speed changes are presented in five specified sections

6. Conclusion

In this study, turbulent flows around a conical valve were analyzed using the SST (Shear Stress Transport) model in the Comsol Multiphysics software package. The results obtained allow us to draw the following conclusions:

The SST model has proven to be an effective tool for analyzing turbulent flows. Its use made it possible to obtain detailed information about the distribution of speed and pressure around the valve.

Changing the surface pressure of a cone valve has a significant impact on its operation and efficiency. Data analysis showed that the pressure distribution is non-uniform and depends on the operating mode of the valve.

Variations in speed across different parts of the system are a key parameter for evaluating efficiency and optimizing design. Observed speed changes may indicate possible bottlenecks or areas of increased flow resistance.

The results of the study can be used to improve the design of cone valves and optimize their performance under various operating conditions.

In summary, our study has contributed to the understanding of turbulent flows in bevel valve systems and provides valuable information for further research and development in this field.

References

1. Aghaei, Y., Kilanehei, F., Faghihirad, S., & Nazari-Sharabian, M. (2021). Dynamic Pressure at Flip Buckets of Chute Spillways: A Numerical Study. International Journal of Civil Engineering, 1-12.
2. Kumcu, S. Y., & Ispir, K. (2022). Experimental and numerical modeling of various energy dissipater designs in chute channels. Applied Water Science, 12(12), 266.
3. Hurtig, K., Croockewit, J., & Vasquez, J. A. (2013). Verification of CFD simulations of flip bucket spillway performance using physical model results. In CANADIAN DAM ASSOCIATION 2013 Annual Conference, Montréal, Québec, Canada.
4. Aydin, M. C., Isik, E., & Ulu, A. E. (2020). Numerical modeling of spillway aerators in high-head dams. Applied water science, 10(1), 42.
5. Salmasi, F., & Samadi, A. (2018). Experimental and numerical simulation of flow over stepped spillways. Applied Water Science, 8(8), 229.
6. Li, D., Yang, Q., Ma, X., & Dai, G. (2018). Case study on application of the step with non-uniform heights at the bottom using a numerical and experimental model. Water, 10(12), 1762.
7. Lee, J. H., Julien, P. Y., & Thornton, C. I. (2019). Interference of dual spillways operations. Journal of Hydraulic Engineering, 145(5), 04019014.
8. Kumcu, S. Y. (2017). Investigation of flow over spillway modeling and comparison between experimental data and CFD analysis. KSCE Journal of Civil Engineering, 21, 994-1003.
9. Almawla, A. S., Kamel, A. H., & Lateef, A. M. (2021). Modelling of Flow Patterns over Spillway with CFD (Case Study: Haditha Dam in Iraq). International Journal of Design & Nature and Eco dynamics, 16(4), 373-385.
10. Madaliev, E., Madaliev, M., Akramov, A., Umurqulov, S., & Qurbonova, S. (2024, November). Numerical simulation of the flow in a flat suddenly expanding channel based on a two-fluid turbulence model. In E3S Web of Conferences (Vol. 508, p. 06001). EDP Sciences.
11. Malikov Z. "Mathematical Model of Turbulence Based on the Dynamics of Two Fluids". Applied Mathematical Modeling. V.82, P. 409-436. 2020 https://doi.org/10.1016/j.apm.2020.01.047
12. Malikov Z.M. Mathematical model of turbulent heat transfer based on the dynamics of two fluids. Applied Mathematic Modeling. 91 2021 186–213. https://orcid.org/0000-0001-9038-5407
13. Malikov, Z. M., Madaliev, M. E., Chernyshev, S. L., & Ionov, A. A. (2024). Validation of a two-fluid turbulence

model in comsol multiphysics for the problem of flow around aerodynamic profiles. Scientific Reports, 14(1), 2306.
14. Dong, Z., Wang, J., Vetsch, D. F., Boes, R. M., & Tan, G. (2019). Numerical simulation of air–water two-phase flow on stepped spillways behind x-shaped flaring gate piers under very high unit discharge. Water, 11(10), 1956.
15. Malikov Z.M., Navruzov D.P., Adilov K., Jurayev S,R. Numerical study of a circular jet based on a modern turbulence model // «Computer Applications for Management and Sustainable Development of Production and Industru» (CMSD2021). doi:10.1117/12.2631607.
16. Z.M. Malikov., D.P. Navruzov., X. Djumayev. Models results Comparison of different approaches to turbulence for flow past a heated flat plate" E3S Web of Conferences 264, 01008 (2021)
17. Menter F. R."Zonal two-equation k-ω turbulence models for aerodynamic flows". AIAAPaper 1993-2906.
18. Malikov, Z., Mirzoev, A., Madaliev, M., Yakhshibayev, D., & Usmonov, A. (2021, November). Numerical simulation of flow through an axisymmetric two-dimensional plane diffuser based on a new two-fluid turbulence model. In 2021 International Conference on Information Science and Communications Technologies (ICISCT) (pp. 1-4). IEEE.
19. 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.
20. Madaliev, E., Madaliev, M., Mullaev, I., Sattorov, A., & Ibrokhimov, A. (2023, March). Numerical simulation of the layer mixing problem based on a new two-fluid turbulence model. In AIP Conference Proceedings (Vol. 2612, No. 1). AIP Publishing.
21. Malikov, Z. M., & Madaliev, M. E. (2022). Numerical simulation of separated flow past a square cylinder based on a two-fluid turbulence model. Journal of Wind Engineering and Industrial Aerodynamics, 231, 105171.
22. Malikov, Z. M., & Madaliev, M. E. (2021). New two-fluid turbulence model-based numerical simulation of flow in a flat suddenly expanding channel. Herald of the Bauman Moscow State Technical University, Series Natural Sciences, 4(97), 24-39.
23. 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.
24. Mirzoev, A. A., Madaliev, M., & Sultanbayevich, D. Y. (2020, November). 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.
25. Madaliev, E., Madaliev, M., Adilov, K., & Pulatov, T. (2021). Comparison of turbulence models for two-phase flow in a centrifugal separator. In E3S Web of Conferences (Vol. 264, p. 01009). EDP Sciences.
26. 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.
27. Menter F. R."Zonal two-equation k-ω turbulence models for aerodynamic flows". AIAAPaper 1993-2906.
28. 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.
29. 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.
30. Galenko P. K., Gomez H., Kropotin N. V., Elder K. R. Unconditionally stable method and numerical solution of the hyperbolic phase-field crystal equation // Physical Review E.– 2013.– Vol. 88.– No. 1.– 013310. https://doi.org/10.1103/PhysRevE.88.013310 ↑68
31. COMSOL Multiphysics® v. 6.0, http://www.comsol.com.– Stockholm: COMSOL AB. ↑68, 76
32. Alnaes M., Blechta J., Hake J., Johansson A., Kehlet B., Logg A., Richardson C., Ring J., Rognes M. E., Wells G. N. The FEniCS Project Version 1.5 // Archive of Numerical Software.– 2015.– Vol. 3.– No. 100.– pp. 9–23. https://doi.org/10.11588/ans.2015.100.20553 ↑68, 76
33. Balay S., Gropp W. D., McInnes L. C., Smith B. F. Efficient management of parallelism in object-oriented numerical software libraries // Modern Software Tools for Scientific Computing, eds. E. Arge, A. M. Bruaset, H. P. Langtangen.– Birkh¨auser Press.– 1997.– pp. 163–202. https://doi.org/10.1007/978-1-4612-1986-6_8 ↑68
34. Xudaykulov, S., Madaliev, M., & Muminov, O. (2023). Turbulent flow out of a convex curve in a channel using the SST turbulence model. In E3S Web of Conferences (Vol. 452, p. 02011). EDP Sciences.
35. Abdukarimov, B., O'tbosarov, S., & Abdurazakov, A. (2021). Investigation of the use of new solar air heaters for drying agricultural products. In E3S Web of Conferences (Vol. 264, p. 01031). EDP Sciences.
36. Mkhonta S. K., Elder K. R., Huang Z. F. Exploring the complex world of two-dimensional ordering with three modes // Physical Review Letters.– 2013.– Vol. 111.– No. 3.–035501. https://doi.org/10.1103/PhysRevLett.111.035501 ↑70
37. Galenko P. K., Elder K. R. Marginal stability analysis of the phase field crystal model in one spatial dimension // Physical Review B.– 2011.– Vol. 83.– No. 6.– pp. 064113. https://doi.org/10.1103/PhysRevB.83.064113 ↑71, 73
38. Madaliev, M., Usmanov, A., & Usmonova, N. (2023). Numerical study of the efficiency of single and group centrifugal cyclones. In E3S Web of Conferences (Vol. 452, p. 06016). EDP Sciences.