Comparison of turbulent flow simulation results in 2d backward facing step channel using Ansys Fluent and Comsol Multiphysics programs

Murodil Madaliev[1,2*], *Shairakhon* Abdujalilova[2], *Zokhidjon* Abdulkhaev[2], *Mavlonbek* Usmonov[2], *Zuhriddin* Umirzakov[2], *Abdusalom* Sattorov[2], and *Kuvonchbek* Pulatov[3]

[1]Institute of Mechanics and Seismic Resistance of Structures of the Academy of Sciences of the Republic of Uzbekistan, 100125, Tashkent, Uzbekistan
[2]Fergana Polytechnic Institute, 86, Fergana Street, 150107, Fergana, Uzbekistan
[3]Fergana State University, 19, Murabbiylar Street, 150100, Fergana, Uzbekistan

Abstract. This paper conducts a comprehensive comparison of numerical simulations of turbulent flow in a sharply expanding channel, utilizing the NASA TMR database as a benchmark. Using advanced computational tools, Ansys Fluent and Comsol Multiphysics 6.2, the study focuses on the hydrodynamic behavior of turbulent flows that are influenced by variations in channel geometry. The analysis emphasizes key flow parameters, including velocity profiles, pressure distribution, and turbulence characteristics, to assess the performance of each software package. By benchmarking the computational results against experimental data from the NASA TMR database, the study highlights the accuracy and efficiency of the methodologies employed. Furthermore, this comparison elucidates the advantages and limitations of Ansys Fluent and Comsol Multiphysics in addressing complex turbulent flow problems. Ultimately, this research seeks to enhance our understanding of turbulent flow dynamics and contribute valuable insights for future studies and applications in fluid mechanics.

1 Introduction

Modeling turbulent flows in channels of complex geometry is one of the key tasks in the field of fluid dynamics, since such processes are widely encountered in various engineering and scientific applications. These flows are of great importance in such areas as aerospace and automotive engineering, energy, chemical industry, as well as in ventilation, air conditioning and heating systems. In these applications, turbulent flows have a direct impact on the performance and reliability of equipment, the energy efficiency of systems and the safety of operation. For example, in the aerospace industry, turbulence plays a decisive role in engine cooling processes, the design of aerodynamic profiles and the optimization of fuel consumption. Accurate calculations of turbulent flow parameters, such

* Corresponding author: Madaliev.me2019@mail.ru

as velocity, pressure and turbulent characteristics, are necessary for the optimization of designs and the improvement of the efficiency of various devices and systems. In addition, turbulence has a significant impact on heat and mass transfer, which is especially important when designing heat exchangers, turbines and other devices associated with high temperatures and pressures. Understanding and predicting the behavior of such flows requires the use of advanced computational methods and software tools that take into account the complex physics of processes and the interaction of various factors, such as changes in channel geometry, properties of the working medium, and the presence of external influences. Despite significant progress in the field of numerical modeling, problems associated with turbulent flows in channels of complex geometry remain very labor-intensive for calculations. This is due to the nonlinear nature of turbulent processes, the multi-scale nature of the phenomena occurring, and the need for an accurate description of transient processes and flow recirculation. Modern software packages such as Ansys Fluent and Comsol Multiphysics provide powerful and flexible tools for solving problems of this kind, but the use of different numerical methods and approaches leads to variations in the results. This makes the problem of comparing and analyzing the obtained data especially relevant [1–4].

A backward facing step channel is a classic problem that requires a detailed analysis both in terms of numerical methods and experimental data. Modeling such flows is associated with a number of difficulties, including the correct description of turbulence, transient processes, and the recirculation zone [5–7]. Modern software packages such as Ansys Fluent and Comsol Multiphysics provide powerful tools for solving problems of this kind. Each of these packages uses different approaches to the numerical solution of the Navier-Stokes equations, which leads to possible differences in the modeling results. For an objective assessment of the accuracy and efficiency of the methods, it is important to compare the obtained numerical results with experimental data [8–11]. The aim of this paper is to compare the results of numerical modeling of turbulent flow in a rapidly expanding channel using Ansys Fluent and Comsol Multiphysics 6.2. Experimental results from the NASA TMR database [12] are used as reference data, which allows validating the numerical models and assessing their applicability for solving problems of this type.

2 Physical statement of the problem

The data presented here are taken from the work of Driver and Seegmiller [13]. This case is also a test case included in the ERCOFTAC database and is often used in turbulent flow modeling workshops. In this problem, the turbulent boundary layer encounters a sharp rollback, causing flow separation. The flow then reattaches and recovers after the step. The Reynolds number calculated from the boundary layer momentum thickness ahead of the step is 5000, which is equivalent to a Reynolds number of about 36,000 based on the step height H. The boundary layer thickness ahead of the step is approximately 1.5 H (see Figure 1). The boundary conditions suitable for CFD modeling are shown in the following figure.

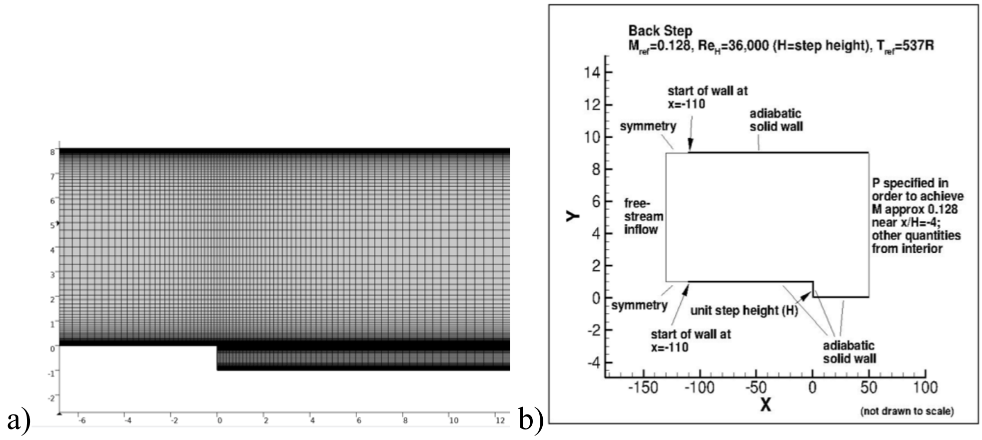


Fig. 1. 2D Backward Facing Step. a) computational mesh and b) boundary conditions.

A 100x100 mesh was chosen for the inlet part of the channel, while a 100x150 mesh was used for the expanding part.

3 Mathematical statement of the problem

The Navier-Stokes equations are a system of nonlinear partial differential equations that describe the motion of a viscous fluid or gas. They are the basis for modeling and analyzing flows in hydrodynamics and aerodynamics. In general, these equations express the conservation of mass (the continuity equation), momentum (vector equations of motion), and energy in a medium:

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

where: v is the fluid velocity vector, t is the time, p is the pressure, ρ is the density, ν is the kinematic viscosity, F is the external force acting on the fluid, ∇ is the nabla operator defining the gradient and divergence of the vector field.

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

The Navier-Stokes equations for an incompressible fluid, which describe the motion of a fluid under the action of viscous forces, are nonlinear. This means that the equations contain nonlinear components that depend on the derivatives of the unknown quantities. Because of this, the analytical solution of these equations is extremely difficult, which makes it necessary to use numerical methods to find approximate solutions.

4 SST turbulence model

The Menter shear stress transfer (SST) model [14–17] is a hybrid model that combines the advantages of the k−ε and k−ω models. The k−ω model is used in the near-wall region, while the k−ε model is used to describe turbulence in the outer zones. Due to its accuracy and versatility, this model has become widely used and is included in most modern computational fluid dynamics software packages.

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

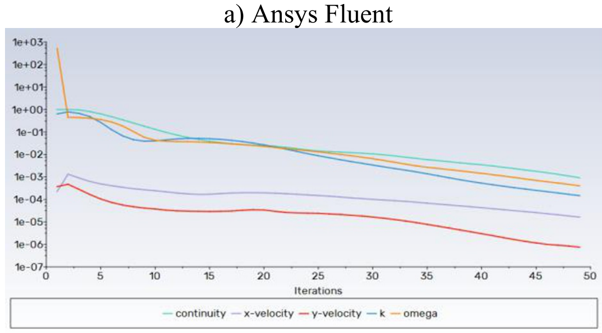
Here k is the specific turbulent kinetic energy ($m^2\ s^{-2}$), ω is the specific rate of turbulent dissipation (s^{-1}). The remaining values are presented in [14-15].

5 Method of solution

ANSYS Fluent provides a powerful set of tools for numerical modeling of various processes in fluid dynamics, including turbulent flows, heat transfer and chemical reactions. To solve the turbulence equations, a fully coupled method (Coupled Solver) was used, the coupling between the pressure and velocity equations is done using algorithms such as SIMPLE. For the problem, the iteration was completed after 48 steps (Figure 2a).

COMSOL Multiphysics provides a wide range of solvers for solving various physical problems, and the choice of an appropriate solver depends on the nature of the physics being modeled, the complexity of the problem, the required accuracy and the available computing resources. To solve the turbulence equations, a fully coupled method (Fully Coupled) based on the PARDISO direct solver algorithm was used. The iterative Newton method with damping was used, with the damping factor being 0.1. For the problem, the iteration was completed after 115 steps. The tolerance factor was set to 1, and the residual factor was 1000 (Figure 2b).

a) Ansys Fluent



b) Comsol Multiphysics

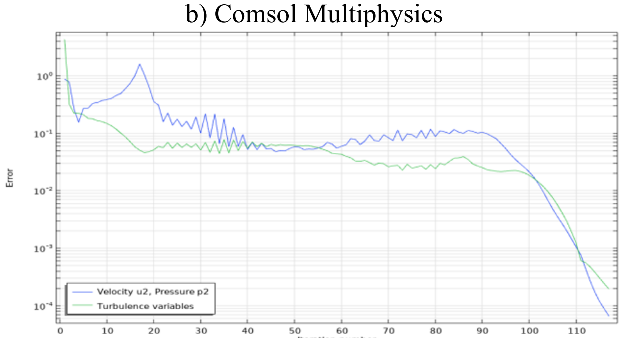


Fig. 2. Iterative processes.

https://doi.org/10.1051/bioconf/202414503041

As shown in Figure 2, ANSYS solved this problem in 110 seconds after 48 iterations, while COMSOL Multiphysics took 198 seconds and 115 iterations. This data shows the difference in performance and computational efficiency between the two software packages when solving the same problem.

6 Calculation results and their discussion

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 profile surface, P_∞ is the free flow pressure, ρ is the free flow density, U_0 is the free flow velocity. The coefficient of surface friction C_f is defined as the ratio of the friction force acting on the profile surface to the dynamic free flow pressure.

$$C_f = \frac{F}{0.5\rho U_0^2 S}$$

where F is the friction force acting on the profile surface, S is the profile surface area oriented along the flow.

Below are comparisons of the obtained numerical results with known experimental data. Figure 3 shows: a) the dependence of the friction coefficient on the pressure of the lower part of the channel; b) the dependence of the friction coefficient on the pressure of the lower part of the channel; as well as the experimental results.

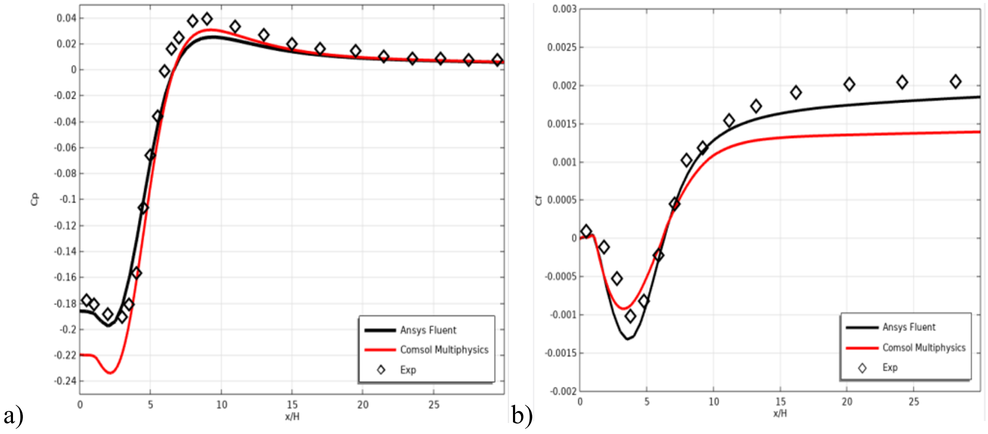
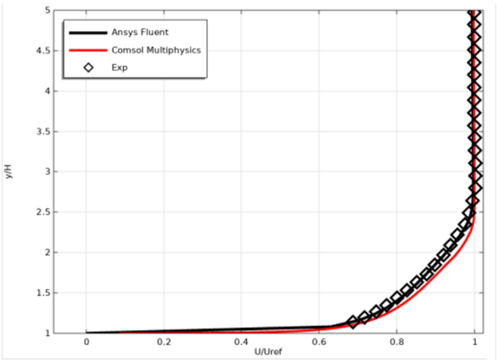


Fig. 3. a) dependence of the friction coefficient on the pressure of the lower part of the channel; b) dependence of the friction coefficient on the pressure of the lower part of the channel; as well as the results of the experiment.

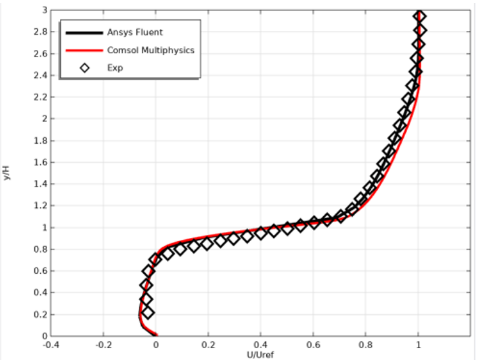
As shown in Figure 3, despite the same initial conditions, both programs show different values of the pressure and resistance coefficients. However, the ANSYS program shows more accurate and consistent results with theoretical data than COMSOL Multiphysics. This may indicate a better adaptation of the ANSYS numerical methods to this problem or a more accurate treatment of the boundary conditions and turbulent characteristics.

https://doi.org/10.1051/bioconf/202414503041

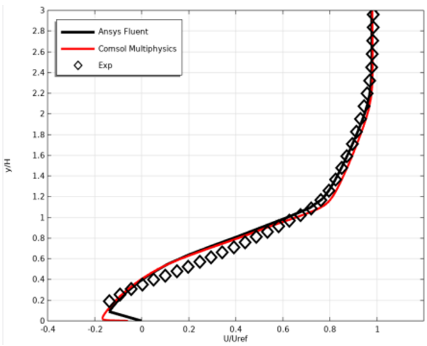
Figures 4-6 show the profiles of longitudinal velocity U, transverse velocity V and turbulent stress profiles $\overline{u'\vartheta'}$ along the lower surface of the channel at different sections downstream, respectively.



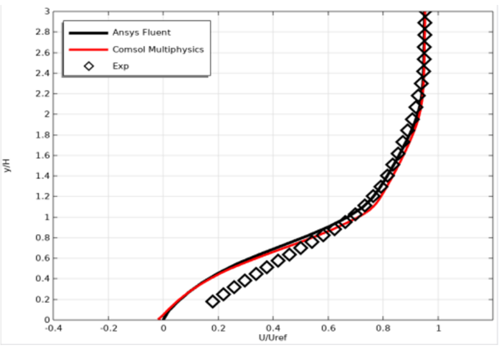
x/H=-4



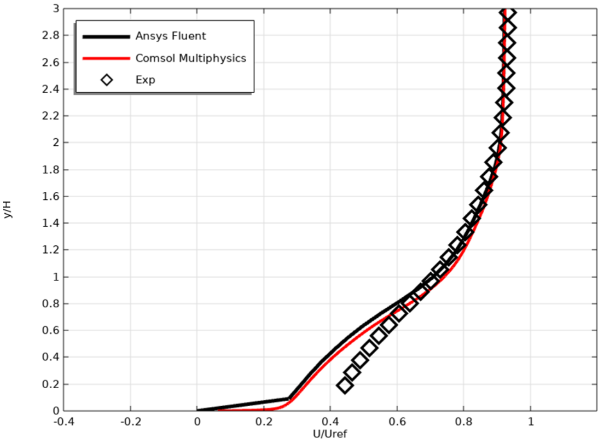
x/H=1



x/H=4



x/H=6



x/H=10

Fig. 4. Longitudinal velocity profiles on the lower surface.

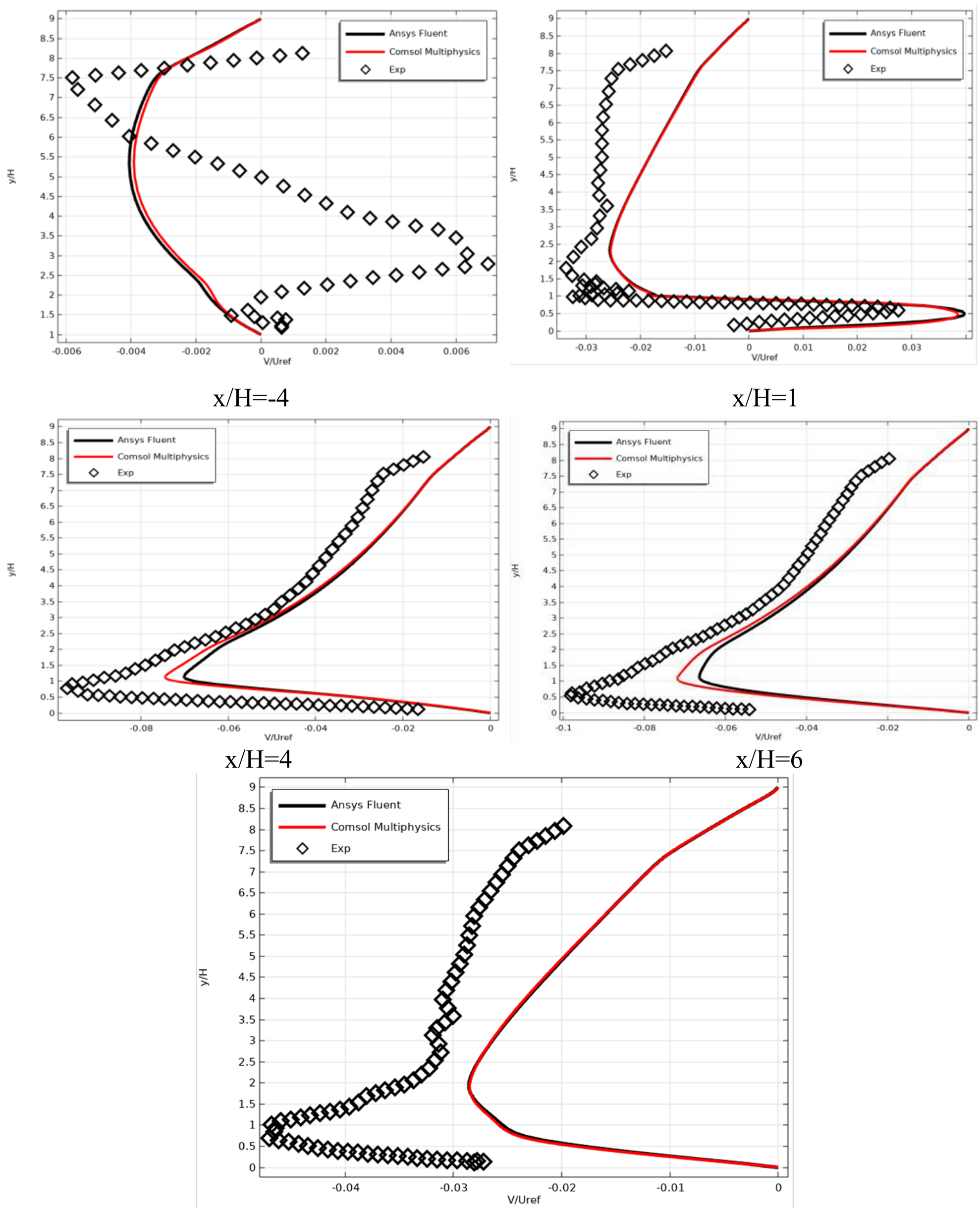


Fig. 5. Transverse velocity profiles on the lower surface.

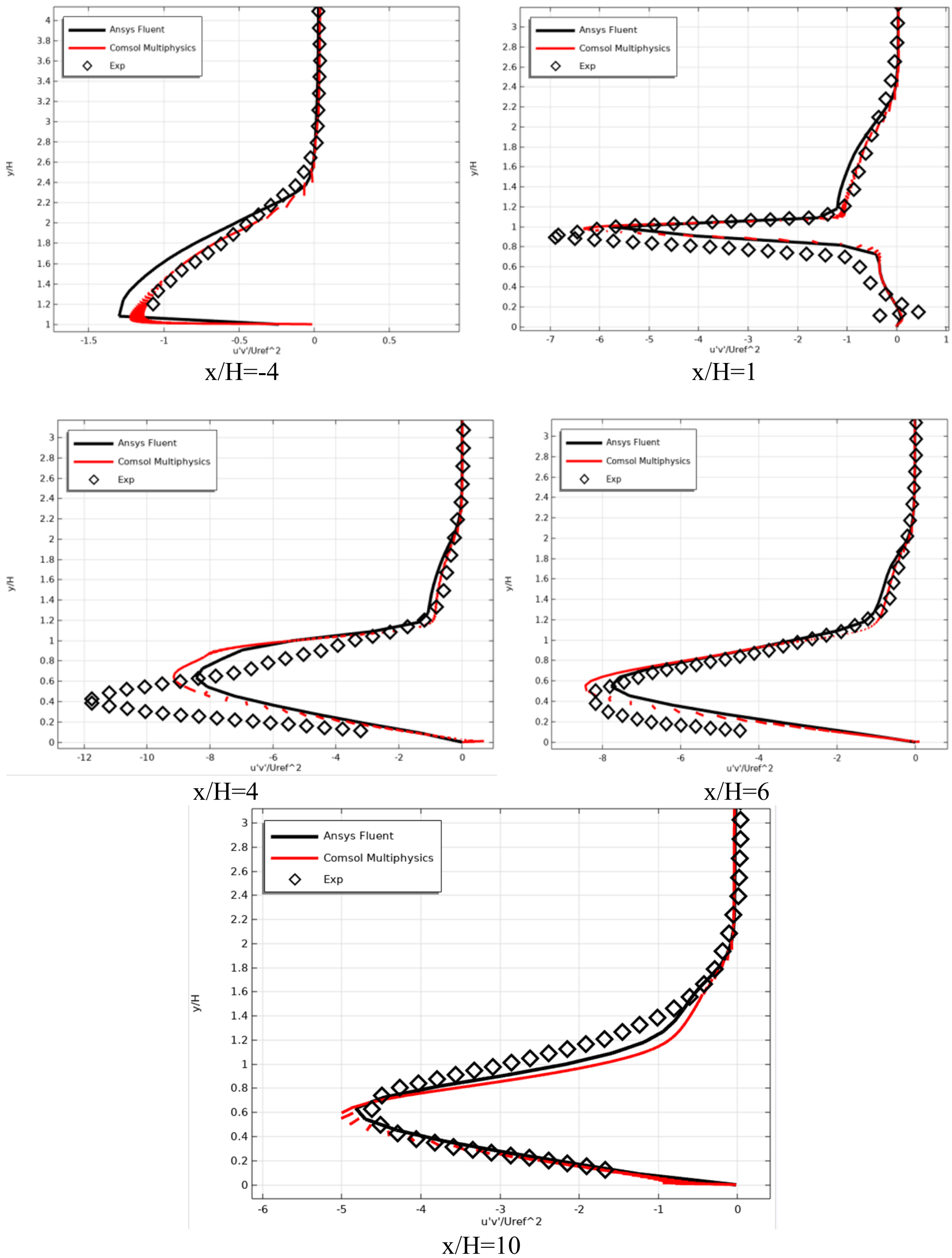


Fig. 6. Reynolds stress-velocity profiles on the lower surface.

Figures 4–6 show that the flow velocity and Reynolds stress values in both ANSYS and COMSOL Multiphysics are similar, indicating a high degree of consistency in the modeling of these parameters despite the differences in the approaches and numerical methods used in each program [18-20].

7 Conclusion

In this paper, we compare the results of numerical simulation of turbulent flow in a sharply expanding channel using ANSYS Fluent and COMSOL Multiphysics software. Both software packages showed satisfactory results in solving hydrodynamic problems, but the performance and accuracy of calculations vary. The ANSYS program demonstrated higher efficiency in terms of computation speed and fewer iterations to achieve convergence. It also showed better results for the pressure and drag coefficients, which confirms its advantage in solving problems with turbulent flows. At the same time, the results for flow velocities and Reynolds stresses in ANSYS and COMSOL Multiphysics were similar, which indicates high consistency of solutions for these parameters. Thus, the choice of software for modeling depends on the specifics of the problem and accuracy requirements, but ANSYS Fluent in this case proved to be a more preferable tool for numerical modeling of turbulent flows in complex geometry.

References

1. W. Orozco Murillo, J. A. Palacio-Fernande, I. D. Patiño Arcila, et al., Journal of Applied and Computational Mechanics **6(Special Issue)**, 1228-1244 (2020)
2. K. Hadad, H. R. Eidi, J. Mokhtari, 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. Z. M. Malikov, A. A. Mirzoev, M. Madaliev, Journal of Computational Applied Mechanics **53(2)**, 282-296 (2022)
6. Z. M. Malikov, M. E. Madaliev, D. P. Navruzov, K. Adilov, Numerical study of an axisymmetric jet based on a new two-fluid turbulence model. AIP Conference Proceedings **2637(1)** (2022)
7. M. Madaliev, E. Yunusaliev, A. Usmanov, et a., E3S Web of Conferences **365**, 01011 (2023)
8. E. Madaliev, M. Madaliev, S. Raxmankulov, S. Raxmonkulova, E3S Web of Conferences **452**, 02012 (2023)
9. M. Madaliev, J. Orzimatov, Z. Abdulkhaev, O. Esonov, M. Mirzaraximov, BIO Web of Conferences **84**, 02032 (2024)
10. E. Madaliev, M. Madaliev, I. Mullaev, et al., Numerical simulation of the layer mixing problem based on a new two-fluid turbulence model. AIP Conference Proceedings **2612(1)** (2023)
11. Z. M. Malikov, M. E. Madaliev, Vestnik Tomskogo Gosudarstvennogo Universiteta. Matematika i Mekhanika **71**, 121-138 (2011)
12. Turbulence modeling Resource. NASA Langley Research Center. http://turbmodels.larc.nasa.gov.
13. D. M. Driver, H. L. Seegmiller, AIAA Journal **23(2)**, 163-171 (1985) DOI: https://doi.org/10.2514/3.8890.
14. F. R. Menter, Zonal two-equation k-ω turbulence models for aerodynamic flows. AIAAPaper 1993-2906 (2012)

15. F. R. Menter, M. Kuntz, and 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., 2003)
16. A. A. Pasha, Journal of Applied and Computational Mechanics **4(2)**, 95-104 (2018)
17. P. R. Spalart, W. H. Jou, M. Strelets, S. R. Allmaras, *Comments on the Feasibility of LES for Wings and on a Hybrid, RANS/LES Approach*, in Proceedings of the 1st AFOSR International Conference on DNS/LES. Advances in DNS/LES **1**, 137–147 (1997)
18. P. Khujaev, J. Ismatullozoda, Z. Umirzakov, et al., E3S Web of Conferences **538**, 01004 (2024)
19. M. Madaliev, M. Usmonov, Z. Umirzakov, et al., E3S Web of Conferences **538**, 01013 (2024)
20. Z. Abdulkhaev, M. Madraximov, S. Abdujalilova, et al., E3S Web of Conferences **452**, 02010 (2023)