Turbulent flow out of a convex curve in a channel using the SST turbulence model

Shuhratjon O'tbosarov[1]*, *Savet* Xudaykulov[2], *Murodil* Madaliev[1,3], and *Oybek* Muminov[1]

[1]Fergana polytechnic institute, st. Fergana 86, Republic of Uzbekistan
[2]Research Institute of Irrigation and Water Problems, Tashkent, st. Korasuv 4/11, Republic of Uzbekistan
[3]Institute of Mechanics and Earthquake Engineering Academy of Sciences of the Republic of Uzbekistan, Mirzo-ulugbek district, street Durmon Yuli, 33, 100125 Tashkent, Uzbekistan

Abstract. The paper presents the results of the SST turbulence model in the Comsol Multiphysics software package for problems of convex boundary layer curvature, which are presented in the NASA database. In this work, the finite element method is used for the numerical implementation of the turbulence equations. To stabilize the discretized equations, Galerkin least squares (GLS) stabilization and crosswind propagation stabilization were used. The results obtained are compared with the results of experimental data.

1 Introduction

Computational fluid dynamics (CFD) is a method for numerical modeling and analysis of fluid dynamics processes. It uses mathematical models and computer algorithms to solve the Navier-Stokes equations that describe the movement of a liquid or gas. CFD allows you to study various aspects of fluid dynamics, such as fluid flow around bodies, heat transfer, mixing of substances and other physical phenomena. It can be used to optimize the design and design of various engineering systems, including automobiles, aircraft, ships, turbines, pumps and other devices. The CFD modeling process involves breaking up space into finite elements, or cells, and then solving the Navier-Stokes equations for each cell. Simulation results can be presented in the form of visualizations, graphs, or numerical data. CFD is widely used in various industries including aviation, automotive, energy, medicine and many others. It allows engineers and scientific researchers to better understand and predict the behavior of liquids and gases, which leads to more efficient and safe design and operation of various systems and devices [1–2].

One of the big challenges encountered when using CFD is turbulence modeling. Turbulent flow is characterized by a complex three-dimensional structure in which random and unpredictable fluctuations in speed and pressure occur. There are several approaches to modeling turbulence in CFD:

1. Closed turbulence equation models: This is the most common approach and is based on approximate equations derived from the Navier-Stokes equations. They contain additional

* Corresponding author: shuhratjonrustamovich1995@gmail.com

equations to model turbulent effects such as viscosity and mixing. Examples of such models include the k-ε model and the k-ω model.
2. Direct Numerical Simulation (DNS): This approach is based on the exact solution of the Navier-Stokes equations without the use of turbulence models. DNS requires high computational power and is used primarily to study fundamental aspects of turbulence.
3. Eddy simulations (LES): This approach combines direct numerical simulations for large-scale turbulence structures and simulations for small-scale structures. LES is used to model turbulence more accurately, but requires more computational power than closed equation models.

Each of these approaches has its own advantages and limitations, and the choice depends on the specific task and available resources. It is important to consider that turbulence modeling in CFD is still an active area of research, and new methods and models are constantly being developed to improve accuracy and efficiency [3–4].

The most common is the Reynolds approach. Based on this approach, a system of Reynolds-averaged Navier-Stokes (RANS) equations is obtained. In the Reynolds-averaged system of equations, the flow variables (velocity, pressure, etc.) are divided into averages and fluctuations. Average values take into account the main flow, while fluctuations reflect turbulent fluctuations. The averaged Navier-Stokes equations then model the fluctuations using closed turbulence models. The basic idea of RANS is that the main flow changes slowly compared to turbulent oscillations, so the Navier-Stokes equations can be averaged to obtain a system of equations describing the average flow. Closed turbulence models, such as the k-ε model or the k-ω model, are used to describe turbulent effects in averaged equations. Reynolds' approach has its limitations, since it is based on the assumption that turbulence is stationary and isotropic. It does not take into account dynamic effects and interactions between different scales of turbulence. However, due to its relative simplicity and computational efficiency, RANS is widely used in engineering calculations and design of various systems and devices.

The Menter k-ω SST (Shear Stress Transport) turbulence model [5–6] is one of the most common models used in the Reynolds-averaged Navier-Stokes (RANS) approach. It is a combination of two models - the k-ω model and the k-ε model. The Menter k-ω SST model provides more accurate results in areas of highly variable velocity gradients, such as walls and flow separation zones. It takes into account the effects of viscosity and mixing of turbulence in the flow, which allows for more realistic predictions.

This model has several features:
1. It uses two equations to describe the turbulent kinetic energy (k) and the specific dissipation rate (ω). The equation for k accounts for production, diffusion, and dissipation, and the equation for ω accounts for production, diffusion, and transport.
2. The model takes into account local velocity and viscosity gradients, which allows for more accurate modeling of different flow regions.
3. It also adopts a switching function, which can automatically switch between k-ω and k-ε models depending on the flow characteristics.

The Menter k-ω SST model is widely used in engineering calculations, especially in aerodynamics and fluid dynamics, where the modeling of complex turbulent flows is required. It provides a good balance between prediction accuracy and computational efficiency. To date, these models have been used to obtain numerical solutions to many important practical problems [7–11].

The purpose of this article is to study the SST turbulence model for problems of turbulent flow in a curved flat channel. 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 transfer (SST) model [5-6] is a combination of the k-ε and k-ω models. For the wall layer, k-ω is used, for the outer region - k-ε. This model is currently very popular and is included in many CFD packages.

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

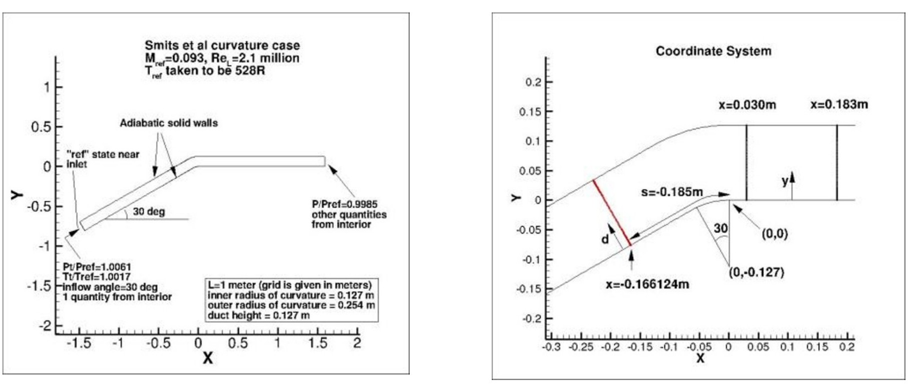
Here k is the specific turbulent kinetic energy ($m^2\ s^{-2}$), ω is the specific rate of turbulent dissipation (s^{-1}). Other values are presented in works [5-6].

3 Solution method

COMSOL Multiphysics is powerful software for modeling and simulating a variety of physical phenomena, including turbulent fluid dynamics. COMSOL Multiphysics provides standard solvers that can be used to solve the Navier-Stokes equations using the SST turbulence model. Standard COMSOL Multiphysics solvers were used for the standard SST turbulence model.

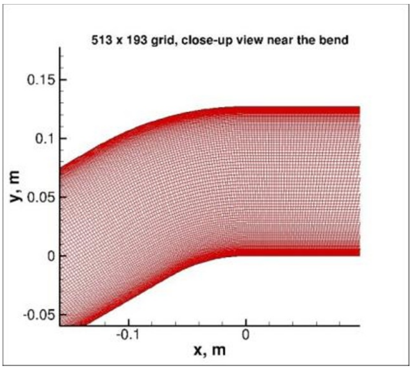
4 Turbulent flow in a curved 2D channel

A boundary layer is a thin layer of fluid that forms on the surface of an object moving relative to the fluid. The study of boundary layers is important in many engineering applications, including aerodynamics and fluid mechanics. The main purpose of this test is to test the implementation of the turbulence model SST in Comsol Multiphysics and compare the results obtained with experimental data presented on the NASA website [12] for turbulent flow in a curved flat channel. Experimental data were obtained in [13]. The experiment uses a rectangular duct of constant area with a height of 0.127 m with a rapid bend of 30 degrees (inner radius of curvature 0.127 m) Fig. 1. In the experiment, the aspect ratio of the air duct was 6:1. This case represents a flow in a channel with a Mach number M = 0.093 and a Reynolds number Re = 2,100,000.



a) b)

 https://doi.org/10.1051/e3sconf/202345202011



c)

Fig. 1. Curved channel. a) boundary conditions c) coordinate system and b) computational grid.

In Fig. 1 (Uref), the average speed near the entrance is 31.9 m/s. Pt is the total pressure, P is the static pressure, and Tt is the total temperature. The distance to the upstream inlet is chosen to allow natural development of a fully turbulent boundary layer and to provide approximately the correct thickness of the boundary layer before bending. The upper and lower boundaries are modeled by adiabatic solid walls. A computational grid of 513 x 193 in size was used, which is presented on the NASA website [12]. 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. [14]

$$C_p = \frac{p - p_\infty}{0.5\rho U_0^2} \tag{2}$$

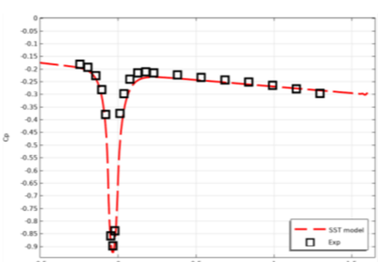
where p is the pressure at a point on the surface of the profile, p_∞ is the pressure of the free flow, ρ is the density of the free flow, U_0 is the speed of the free flow. [15-16]

The coefficient of surface friction C_f is defined as the ratio of the friction force acting on the surface of the profile to the dynamic pressure of the free flow.

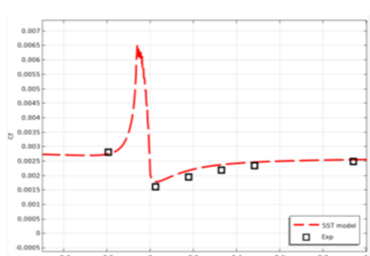
$$C_f = \frac{F}{0.5\rho U_0^2 S} \tag{3}$$

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

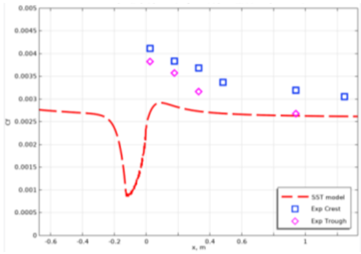
Below are comparisons of the obtained numerical results with known experimental data. Figure 2 shows: a) dependence of the friction coefficient pressure; b) coefficient of friction of the lower part of the channel and c) friction of the upper part of the channel, as well as the experimental results.



a)



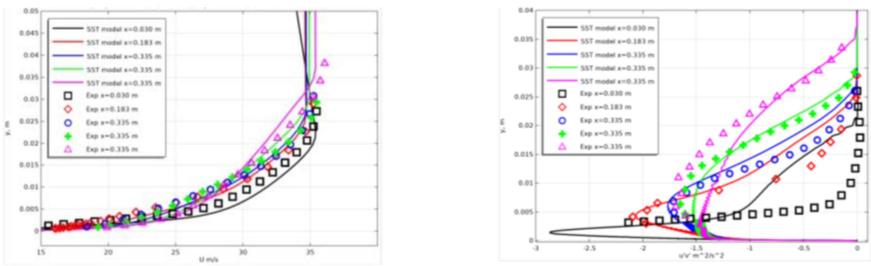
b)



c)

Fig. 2. Shown: a) dependence of the friction coefficient; pressure; b) the coefficient of friction of the lower part of the channel; and c) the friction of the upper part of the channel.

In Fig. Figure 3 shows the longitudinal velocity profiles U (m/s) and turbulent stress $\overline{u'\vartheta'}$ profiles along the lower surface of the channel at different sections along the flow.

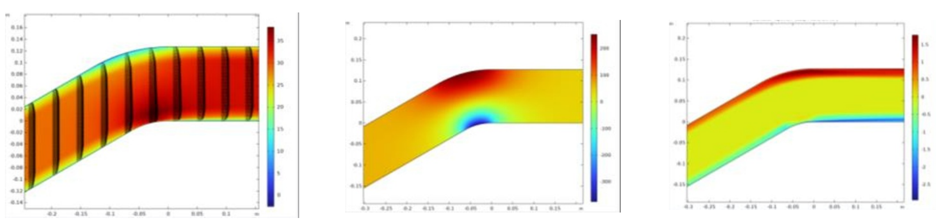


a) b)

Fig. 3. Profiles of longitudinal velocity and turbulent stresses on the lower surface of the channel.

As can be seen from Fig. 3, the results of the SST model are close to the experimental results. The turbulent stress results deviate from the experimental data at large distances from the bend.

In Fig. Figure 4 shows isolines of velocity, pressure, and turbulent stress according to the SST model in the concave part of the channel.



a) b) c)

Fig. 4. Isolines of a) velocity m/s, b) pressure Pa, c) turbulent stress m^2/s^2 of the concave part of the channel.

It is known that turbulence levels decrease near convex walls compared to flow near straight walls. Here the turbulence model can be evaluated for its ability to capture this effect. We can say that the SST model shows this situation very well [20-25].

5 Conclusion

This article shows the results of a standard SST turbulence model in the Comsol Multiphysics software package, which uses the finite element method. To validate the SST model, 2D

convex boundary layer curvature problems are considered. From the obtained results it is clear that the SST model has high accuracy. The study shows that the SST model has several advantages:
– it takes into account both viscosity and turbulence mixing, allowing for more accurate results in areas of widely varying velocity gradients.
– the model takes into account local flow characteristics such as velocity and viscosity gradients, allowing for more accurate modeling of different flow regions.
– the model has a good balance between prediction accuracy and computational efficiency, which allows calculations to be carried out with a reasonable execution time.

Therefore, the SST model can be recommended for calculating engineering problems of turbulent hydrodynamics.

References

1. W. Orozco Murillo, J. A. Palacio-Fernande, I. D. Patiño Arcila, J. S. Zapata, J. A. Hincapié Isaza, Analysis of a Jet Pump Performance under Different Primary Nozzle Positions and Inlet Pressures using two Approaches: One Dimensional Analytical Model and Three Dimensional CFD Simulations. Journal of Applied and Computational Mechanics, **6**, 1228-1244 (2020)
2. K. Hadad, H. R. Eidi, J. Mokhtari, VOC level control by ventilation improvement of Flexography printing room using CFD modeling. Journal of Applied and Computational Mechanics, **3(3)**, 171-177 (2017)
3. E. G. Tsega, V. K. Katiyar, A Numerical Simulation of Inspiratory Airflow in Human Airways during Exercise at Sea Level and at High Altitude. Journal of Applied and Computational Mechanics, **5(1)**, 70-76 (2019)
4. A. V. Sentyabov, A. A. Gavrilov, A. A. Dekterev, Investigation of turbulence models for computation of swirling flows. Thermophysics and aeromechanics, **18(1)**, 73-85 (2011)
5. F. R. Menter, “Zonal two-equation k-ω turbulence models for aerodynamic flows”. AIAAPaper, 2906 (1993)
6. F. R. Menter, M. Kuntz, R. Langtry, “TenYears of Industrial Experience with the SST Turbulence Model”. Turbulence, Heat and Mass Transfer 4, Begell House, Inc., 625 – 632 (2003)
7. A. A. Pasha, 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 (2018)
8. Z. M. Malikov, M. E. Madaliev, Numerical study of a swirling turbulent flow through a channel with an abrubt expansion. Vestnik Tomskogo Gosudarstvennogo Universiteta. Matematika i Mekhanika, **72**, 93-101 (2021)
9. Z. M. Malikov, M. E. Madaliev, Mathematical modeling of a turbulent flow in a centrifugal separator. Vestnik Tomskogo Gosudarstvennogo Universiteta. Matematika i Mekhanika, **71**, 121-138 (2021)
10. M. M. Erkinjon son, Numerical Calculation of an Air Centrifugal Separator Based on the SARC Turbulence Model, J. Appl. Comput. Mech., **7(2)**, x–xx. (2021) https://doi.org/10.22055/JACM.2020.31423.1871
11. 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, Advances in DNS/LES,

Proceedings of 1st AFOSR International Conference on DNS/LES, Greyden Press, Columbus, **1**, 137–147 (1997)
12. Turbulence modeling Resource. NASA Langley Research Center http://turbmodels.larc.nasa.gov (Last accessed 15.07.2023)
13. A. J. Smits, S. T. B. Young, P. Bradshaw, The Effect of Short Regions of High Surface Curvature on Turbulent Boundary Layers, J. Fluid Mech., **94(2)**, 209-242 (1979) https://doi.org/10.1017/S0022112079001002.
14. B. A. Abdukarimov, A. A. Kuchkarov, Research of Hydrodynamic Processes Occurring in Solar Air Heater Collectors with a Concave Air Duct Absorber, Applied Solar Energy (English translation of Geliotekhnika), **58(6)**, 847–853 (2022)
15. B. A. Abdukarimov, A. Kuchkarov, A Numerical Solution of the Mathematical Model of Air Flow Movement in a Solar Air Heater with a Concave Tube Applied Solar Energy, **58(1)**, 109–115 (2022)
16. B. Abdukarimov, S. O'tbosarov, A. Abdurazakov, *Investigation of the use of new solar air heaters for drying agricultural products.* E3S Web of Conferences, EDP Sciences **264** (2021)
17. A. Arifjanov, S. Jurayev, T. Qosimov, S. Xoshimov, Z. Abdulkhaev, *Investigation of the interaction of hydraulic parameters of the channel in the filtration process*. In E3S Web of Conferences, EDP Sciences, **401**, 03074 (2023)
18. Z. Abdulkhaev, M. Madraximov, A. Akramov, A. Arifjanov, *Groundwater flow modeling in urban areas*. In AIP Conference Proceedings, AIP Publishing, 2789(1) (2023)
19. Z. Abdulkhaev, M. Madraximov, A. Arifjanov, N. Tashpulotov, *Optimal methods of controlling centrifugal pumps*, In AIP Conference Proceedings, AIP Publishing, 2612(1) (2023)
20. M. E. U. Madaliev, Z. E. Abdulkhaev, N. E. Toshpulatov, A. A. Sattorov, Comparison of finite-difference schemes for the first order wave equation problem. In AIP Conference Proceedings, 2637, 1 (2022)
21. A. Arifjanov, L. Samiev, S. Yusupov, D. Khusanova, Z. Abdulkhaev, S. Tadjiboyev, Groundwater Level Analyse In Urgench City With Using Modflow Modeling And Forecasting System. In E3S Web of Conferences, EDP Sciences, **263**, 03010 (2021)