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

Shuhratjon O'tbosarov*, Savet Xudaykulov1, Murodil Madaliev1,3, and Oybek Muminov1

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, 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:

- 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

© The Authors, published by EDP Sciences. This is an open access article distributed under the terms of the Creative Commons Attribution License 4.0 (https://creativecommons.org/licenses/by/4.0/).
equations to model turbulent effects such as viscosity and mixing. Examples of such models include the $k-\varepsilon$ model and the $k-\omega$ 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 $k-\varepsilon$ model or the $k-\omega$ 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 $k-\omega$ 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 $k-\omega$ model and the $k-\varepsilon$ model. The Menter $k-\omega$ 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 ($\omega$). The equation for $k$ accounts for production, diffusion, and dissipation, and the equation for $\omega$ 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 $k-\omega$ and $k-\varepsilon$ models depending on the flow characteristics.

The Menter $k-\omega$ 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 NA SA Turbulence Modeling Resource (TMR) website [12].
2 SST turbulence model.

\[
\begin{align*}
(U \cdot \nabla)k &= \nabla \left[ \bar{v} + \sigma_k \nabla k \right] + P - \beta \omega k \\
(U \cdot \nabla)\omega &= \nabla \left[ \bar{v} + \sigma_\omega \nabla \omega \right] + \frac{\gamma}{\nu_t} P - \beta \omega \omega + \frac{\sigma_\omega}{\nu_t} \nabla \omega \nabla k
\end{align*}
\]

3 Solution method

4 Turbulent flow in a curved 2D channel
In Fig. 1 (Uref), the average speed near the entrance is 31.9 m/s. $P_t$ is the total pressure, $P$ is the static pressure, and $T_t$ 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]

\[ C_p = \frac{P - P_{\infty}}{\frac{1}{2} \rho U_0^2} \]

where $P$ is the pressure at a point on the surface of the profile, $P_{\infty}$ is the pressure of the free flow, $\rho$ 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.

\[ C_f = \frac{F}{\frac{1}{2} \rho U_0^2 S} \]

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.
3 Conclusion

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].
 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


https://doi.org/10.22055/JACM.2020.31423.1871

12. Turbulence modeling Resource. NASA Langley Research Center
http://turbmodels.larc.nasa.gov (Last accessed 15.07.2023)

https://doi.org/10.1017/S0022112079001002


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)


E3S Web of Conferences 452, 02011 (2023)
https://doi.org/10.1051/e3sconf/202345202011

IPFA 2023