Comparison of turbulence models for solving problems of swirling jet flows

Murodil Madaliev\textsuperscript{1}*, Dilshod Navro'zov\textsuperscript{2}, Yunus Hamrayev\textsuperscript{2}, Nasillo Hamroyev\textsuperscript{2}, Dilnora Sharipova\textsuperscript{2}, and Salima Do'stova\textsuperscript{2}

\textsuperscript{1}Institute of Mechanics and Seismic Stability of Structures named after M.T. Urazbaev of the Academy of Sciences of the Republic of Uzbekistan, 100000 Tashkent, Uzbekistan.

\textsuperscript{2}Bukhara Institute of Natural Resources Management of the National Research University “TIIAME”, 200100 Bukhara, Uzbekistan

Abstract. There are several turbulence models that can be used to solve the swirling jet problems, which are typical turbulence problems. A comparison of these models allows us to determine which one is better suited for a given task. When comparing turbulence models for the swirling jet problem, the most important criteria are the accuracy and stability of the solution. Accuracy is assessed by comparison with experimental data or other numerical methods, and stability is assessed by the absence of oscillations and convergence of the solution. In addition, it is also important to consider the computational complexity of each model and its applicability to a specific problem. The swirling flow in a nozzle is studied in the article using various turbulence models (SA, k-\varepsilon, k-\omega, L-VEL, v\textsuperscript{2}-f, yPlus, SST).

1 Introduction

Swirling flows, due to their wide practical application, represent an important branch of modern hydrodynamics. The complexity of these flows leads to the use of the most modern research methods in their study. The universally adopted methods of numerical modeling, and various fine measurement techniques, including non-contact ones, which are no longer the property of individual research laboratories and are becoming increasingly widespread, allow us to hope for significant progress in the study and prediction of the properties and structure of complex hydrodynamic processes, including swirling flows. Swirling flows are widespread in many natural phenomena and are often used in technology [1]. As an example (in the absence of combustion), we can mention cyclones and tornadoes in the atmosphere, whirlpools in rivers and reservoirs, vortex wake left by an airplane wing, separators and heat exchangers of cyclone type, sprinklers and agricultural machines spraying fertilizers, jet pumps, and chemical reactors with vortex flow [2]. Elements of the theory of swirling flows are used even when analyzing the flight of a boomerang and a bee. In technical combustion devices, the significant and beneficial effect of the swirling of supplied air and fuel flows on...
the stabilization and intensification of the combustion process is intensively used to ensure efficient and environmentally friendly combustion of fuel in a wide variety of practical areas: in gasoline and diesel engines, in gas turbines and gas turbine engines, in industrial furnaces, boilers, and a number of other heating systems. As experimental studies show, swirl radically affects the flow field; its important properties and characteristics, such as the jet propagation and attenuation and the ejection of matter by a jet in inert flows, the size and shape of the flame and the intensity of combustion in reacting flows, significantly depend on the degree of swirl, imparted to the flow [3].

Experiments are a great help for designers, however, their setting is quite expensive; preliminary calculation of the flow field using mathematical models and numerical methods would significantly contribute to reducing the cost of development and operating expenses. The duration and cost of development can be significantly reduced by combining experimental and theoretical data on combustion aerodynamics with detailed numerical gas dynamic calculations, and applying and improving appropriate software [4].

The influence of the initial flow swirl on the flow field increases sharply with an increase in the degree of swirl. The effect of weak swirl (S<0.4) is reduced to an increase in the width of the free or confined jet: the increase in the width of the jet, mixing of the flow, and the decrease in jet velocity occur more intensely with an increase in the degree of swirl. Although there may be significant pressure gradients in the transverse (or radial) direction at any section of the jet, the non-zero gradient is a consequence of the effect of the swirl on the pressure distribution. The most significant and useful phenomena in swirling jet flows for applied problems with combustion can be considered the existence of a recirculation zone formed in the central part at supercritical values of the swirl parameter (S=0.6 for swirling devices with straight outlet). Numerical studies of swirling flows with standard two-equation k-ε turbulence models have shown that these models can significantly underestimate mixing rates. Modifications of two-equation models for improvement [5-9].

Predictions of mixing in swirling flows varied with changes in eddy viscosity. Such methods as Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) are labor-intensive and require high-speed computers to solve complex aerodynamic problems. Therefore, their wide practical use is associated with the development of computer technology and, according to experts, can begin only at the end of this century. Therefore, in the near future, semi-empirical methods will remain the main working tool for solving applied problems of aerodynamics [10-14].

Therefore, the purpose of this study is to numerically analyze swirling flows using different turbulence models and compare their results. The results of this study can be useful for understanding turbulence, separation, and reattachment, and for selecting appropriate turbulence models, which are important for the study of practical engineering applications.

2 Physical and mathematical formulation of the problem
3 Mathematical formulations and computational methods

\[
\begin{align*}
\rho \frac{\partial \bar{U}_i}{\partial t} + \rho \bar{U}_j \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial p}{\partial x_j} &= \mu \frac{\partial}{\partial x_j} \left( \frac{\partial \bar{U}_i}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( - \rho \nu_j' u_i' \right) \\
\frac{\partial \bar{U}_i}{\partial x_j} &= 0
\end{align*}
\]

Here t is the turbulent viscosity to be determined. In this study, seven turbulence models included in the COMSOL Multiphysics software package were used to determine turbulent viscosity.

Turbulence models

The Spalart-Allmaras model is one of the turbulence models used to solve the Navier-Stokes equations in computer models. This model was developed by François Spalart and Philippe Allmaras in 1994. The Spalart-Allmaras model is based on

\[
E = 3S - W - E_{fl} - E_{co} - E_{IPFA}
\]
Turbulent eddy viscosity is calculated from the following equation:

\[ \nu_t = C_{\mu} \frac{k}{\varepsilon} \]

Equations for the Spalart-Allmaras (SST) model are given by:

\[ \begin{align*}
(U \cdot \nabla)k &= \nabla \left[ \frac{\nu_t}{\sigma_k} \nabla k \right] + P - \varepsilon \\
(U \cdot \nabla)\varepsilon &= \nabla \left[ \frac{\nu_t}{\sigma_{\varepsilon}} \nabla \varepsilon \right] + C_{\varepsilon} \frac{\varepsilon}{k} \frac{P - G_{\varepsilon}}{\varepsilon}
\end{align*} \]

\[ \nu_t = C_{\mu} \frac{k}{\varepsilon} \]

where \( \nu_t \) is the turbulent viscosity, \( k \) is the turbulent kinetic energy, \( \varepsilon \) is the turbulence dissipation rate, \( P \) is the rate of production of turbulent kinetic energy, \( G_{\varepsilon} \) is the dissipation rate of turbulent kinetic energy, and \( \sigma_k \) and \( \sigma_{\varepsilon} \) are the turbulent length scales.

The SST model is one of the most commonly used turbulence models. It uses the \( k \)-\( \varepsilon \) model and the \( \omega \) model that describe the transport of effective viscosity by two equations. Two vortices are also used to solve the Navier-Stokes equations in computer models. It was developed in the 1970s and is used to describe turbulence in regions where flow velocity is low.
The $v^2-f$ turbulence model is one of the turbulence models used to solve the Navier-Stokes equations in computer models. This model was developed in the 1990s and is designed to simulate flows with low to moderate turbulence. The $v^2-f$ model is based on two variables: the square of vorticity ($v^2$) and function $f$, which is related to the rate of dissipation of turbulent energy. The equations for $v^2$ and $f$ are determined based on the Navier-Stokes equations and the equations for the transport of turbulent energy and its dissipation. One of the advantages of the $v^2-f$ model is its ability to describe flows with low to moderate turbulence, which presents difficulties for other turbulence models [19, 20].

$$
(U \cdot \nabla)k = \nabla \left[ v + \frac{v}{\sigma_k} \nabla k \right] + P - \varepsilon \\
(U \cdot \nabla)\varepsilon = \nabla \left[ v + \frac{v}{\sigma_\varepsilon} \nabla \varepsilon \right] + \frac{1}{\tau} \left( C_{\varepsilon} (\zeta \alpha) P_k - C_{\varepsilon} (k \alpha \varepsilon) \varepsilon \right) \\
(U \cdot \nabla)\zeta = \nabla \left[ v + \frac{v}{\sigma_\zeta} \nabla \zeta \right] + \frac{1}{k} \left( \alpha v + \frac{v}{\sigma_\varepsilon} \right) \nabla k \nabla \zeta + \left( f_w - \alpha f \right) f_k - \frac{\zeta}{k} P_k
$$

$$
\nu_t = C_\mu k \zeta \tau
$$

4 Calculation grids

Fig. 2.

$$
Y_x = U \\
V_y = P =
$$
5 Solution method

The finite element method was used [17-18] to numerically solve the system of initial non-stationary equations (1-5). For the solution, standard COMSOL Multiphysics 6.1 solvers were used [19].

6 Calculation results and discussion

Figure 3 shows isolines of longitudinal, transverse, and tangential flow velocity and pressure for various turbulence models.

Fig. 3. Isolines of longitudinal velocity and flow pressure for various turbulence models.
7 Conclusion

The main goal of this study is to analyze various turbulence models of the COMSOL software for the numerical study of swirling turbulent flows. The numerical study was conducted with SA, k-ε, SST, and $v^2$-f turbulence models. The results obtained were compared with experimental results. The following conclusions can be drawn from the comparison:

The SA, SST and $v^2$-f turbulence models show approximately the same results. The differences between numerical results and experimental data stem from the fact that errors in numerical results can come from many different sources, including turbulence models. Of course, this informal ranking of turbulence models highly depends on the user and the data of interest for the user. However, these results have provided significant and invaluable insight into the capabilities of turbulence models, and revealed which CFD turbulence model can be used for the problems of industrial design. We can state that the $v^2$-f turbulence model very well describes swirling flows.

References


20. Z. М. Malikov, D. Р. Navruzov, Kh. Djumayev, E3S Web of Conferences 264, 01008 (2021) https://doi.org/10.1051/e3sconf/202126401008