Calculation of an Axisymmetric Hot Subsonic Turbulent Jet

Farrukh Nazarov [1,2 a)], Nematjon Mustafayev[3, b)] and Sardor Abdukhamidov[1,c)]

[1]*Institute of Mechanics and Seismic Stability of Structures named after M.T.Urazbaev, Uzbekistan Academy of Sciences, 100125, Tashkent, Dormon yuli, 40, Uzbekistan.*
[2]*Fergana state technical university, Fergana, Uzbekistan*
[3]*Bukhara State Technical University*

[a)] Corresponding author: farruxnazar@mail.ru
[b)] nematjonmustafayev70@mail.ru
[c)] sardor.abdukhamidov@mail.ru

Abstract. This article examines a hot jet flowing into a submerged space from a nozzle with a radius of 25.4 mm. The Mach number at the jet exit for this particular case is approximately M=0.376. In this paper, the results were obtained for various turbulence models using the COMSOL MULTIPHYSICS® software package. The solutions of these models are compared with NASA experimental results for turbulent stresses, longitudinal and transverse velocities in various cross-sections. Additionally, a comparison of axial velocities and turbulent kinetic energy with experimental results is conducted.

INTRODUCTION

An axisymmetric hot subsonic jet is a flow of hot gas propagating around an axis of symmetry. The study of such jets is important in aerodynamics and engineering, particularly in the development of rocket engines and aircraft systems.

An axisymmetric hot subsonic jet has a number of features that determine its behavior and properties [1-7]:

1. Axisymmetric: The jet is axisymmetric, meaning that its shape and properties are the same along the axis of symmetry, making it easier to model and analyze.

2. Subsonic velocity: The gas velocity in this jet is below the speed of sound. This is due to the gas expanding along the jet, resulting in a decrease in velocity and an increase in pressure.

3. Temperature: The hot jet contains gases at elevated temperatures, which may require special materials for the nozzle construction and other components to withstand the high temperatures.

4. Cone shape: When exiting the nozzle, the hot jet may have a cone shape as the expansion of the gas causes its shape to change.

These features make axisymmetric hot subsonic jets important for developing technologies in aviation, rocketry, and other engineering fields. Comparing computational fluid dynamics (CFD) results with experimental data is an important step in studying axisymmetric hot subsonic jets. This allows one to evaluate the accuracy of the simulations and ensure the validity of the applied hydrodynamic models. Comparing CFD results with experimental data reveals key differences and similarities [8-11]. If the results are consistent, the confidence in the model increases significantly. It is important to keep in mind that the comparison may require additional calibration or adjustment of the model parameters in case of discrepancies. Therefore, comparing CFD results with experimental data plays a key role in substantiating the accuracy and reliability of the modeling of an axisymmetric hot subsonic jet. To generate an axisymmetric hot subsonic jet, certain pressure and temperature conditions are used on the inlet surface of the housing. These parameters determine the initial conditions for the modeling or experimental creation of the jet. Typically, to generate a subsonic jet, a certain level of total pressure and temperature at the inlet is required, which allows achieving the desired conditions in the jet, including the velocity and temperature along its axis. When setting the initial conditions, it is important to consider not only the pressure and temperature values, but also their distribution and gradients in accordance with a specific problem and requirements. Thus, setting the total pressure and temperature on

the inlet surface of the jet is an important part of the process of creating and studying an axisymmetric hot subsonic jet.

Turbulence modeling is an important aspect in the study of subsonic jets and other aerodynamic phenomena. Several approaches to turbulence modeling exist in computer studies of liquid and gas flows, including the following techniques [12,14-18]:

1. Model k-ε: This model is one of the most widely used in engineering calculations. It is based on equations for the amount of turbulent kinetic energy (k) and its dissipation (ε), and allows one to take into account the effect of turbulence on the flow [19-25].

2. Model k-ω: This model is also based on two equations for the turbulent kinetic energy (k) and the twist frequency (ω). It is widely used to model the turbulence boundary in high pressure gradient flows.

3. LES (Large Eddy Large-eddy simulation (LES): This approach solves the Navier-Stokes equations for large eddies and is used to study turbulent flows with high eddy frequencies [26-30].

The choice of model depends on the flow type, turbulence characteristics, available computational resources, and the desired accuracy of reproducing the physical processes. It is important to note that turbulence modeling in aerodynamics is a complex field and requires appropriate knowledge and experience for proper application.

PHYSICAL FORMULATION OF THE PROBLEM

The experiment used a jet emanating from a nozzle with a radius of 1 inch (25.4 mm). The Mach number at the jet exit for this particular case is approximately $M_{jet} = \dfrac{a_{jet}}{u_{jet}} = 0.376$ while the "acoustic Mach number" $\dfrac{a_{jet}}{u_{jet}}$ is approximately 0.5. In the experiment, an axisymmetric jet emerges into still (stationary) air. However, since flow into still air is difficult to achieve for some CFD codes, the CFD here is calculated with very low background environmental conditions ($M_{ref} = 0.01$ moving from left to right in the same direction as the jet). This difference in boundary conditions has some effect, but tests have shown that the impact is relatively small and $M_{ref} = 0.01$ represents a reasonable compromise. The appropriate jet conditions are achieved by setting the total pressure and temperature at the jet inlet surface as shown in Figure 1 [13].
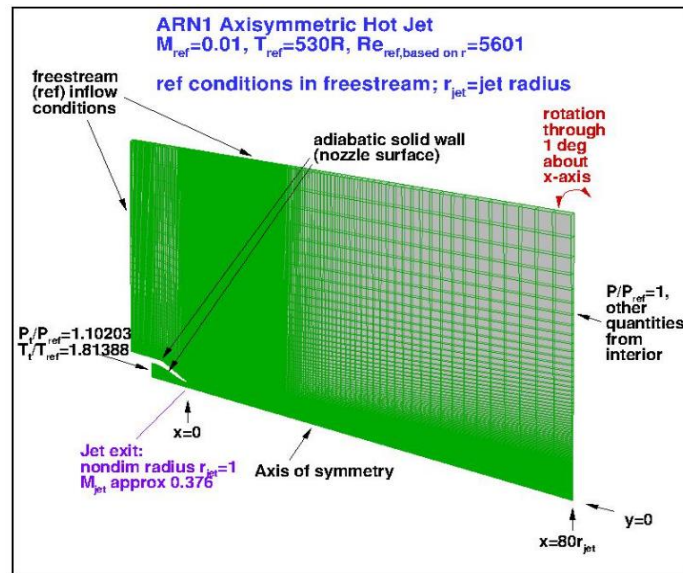

FIGURE 1. Computational grids.

MATHEMATICAL FORMULATION OF THE PROBLEM

For the numerical study of the problem, the equation for heat propagation in liquids and the Navier-Stokes equations for the motion of liquids or gases are calculated jointly [31-35].

$$\begin{cases} \dfrac{\partial \overline{U}_j}{\partial x_j} = 0, \\[3mm] \dfrac{\partial \overline{U}_i}{\partial t} + \overline{U}_j \dfrac{\partial \overline{U}_i}{\partial x_j} = -\dfrac{1}{\rho}\dfrac{\partial p}{\partial x_i} + \dfrac{\partial}{\partial x_j}\left[(v+v_t)\left(\dfrac{\partial \overline{U}_i}{\partial x_j}+\dfrac{\partial \overline{U}_j}{\partial x_i}\right)\right], \\[3mm] \rho C_p \dfrac{\partial T}{\partial t} + \rho C_p \overline{u}\cdot\nabla T + \nabla \overline{q} = Q + Q_{ted}, \\[3mm] q = -k\nabla T. \end{cases} \qquad (11)$$

here is k-the thermal conductivity, $\overline{U}_i$ - respectively, the axial, radial and tangential components of the air flow velocity; p – hydrostatic pressure; ρ – gas density; v, v_t - molecular and turbulent viscosities, C_p -heat capacity at constant pressure, q is the internal heat flux, Q is the amount of heat.

SOLUTION METHOD

The finite element method was used to numerically solve the system of initial non-stationary equations (1). Standard solvers from COMSOL Multiphysics 6.1 were used for the solution.

COMSOL Multiphysics offers a range of solvers for various types of physics problems. The choice of solver depends on the type of physics being modeled, the complexity of the problem, the desired accuracy, and the available computational resources. For standard SST turbulence models, k-ε, k-ω standard COMSOL MULTIPHYSICS solvers were used.

Figure 2 shows the change in the number of iterations and the error.
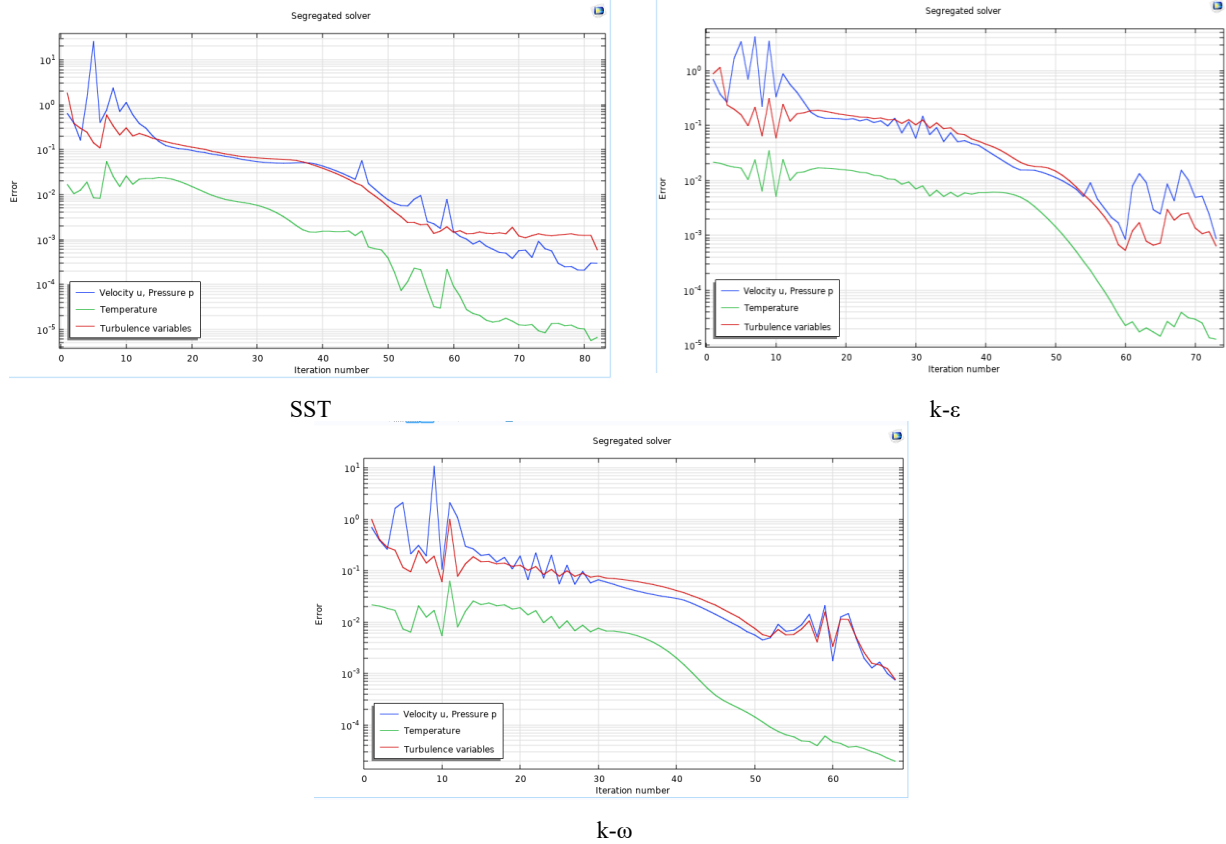

SST

k-ε

k-ω

FIGURE 2. Change in the number of iterations and errors

The time required to perform calculations on different grids is presented in Table 1. All calculations were performed on a computer with a quad-core Intel i5-7300 HQ processor, 2.5 GHz, 16 GB of DDR3 memory, a 1024 GB hard drive, and a Windows 7 (64-bit) operating system.

TABLE 1. Time spent on performing calculations for different models

models	SST	k-ε	k-ω
time spent in seconds	492	275	293

CALCULATION RESULTS AND THEIR DISCUSSION

Figure 3 shows a comparison of the results of an axisymmetric hot subsonic jet using the *SST model* , *k-ε* and *k-ω* with experimental data for dimensionless axial velocity and turbulent kinetic energy from the distance to the nozzle.

a   b
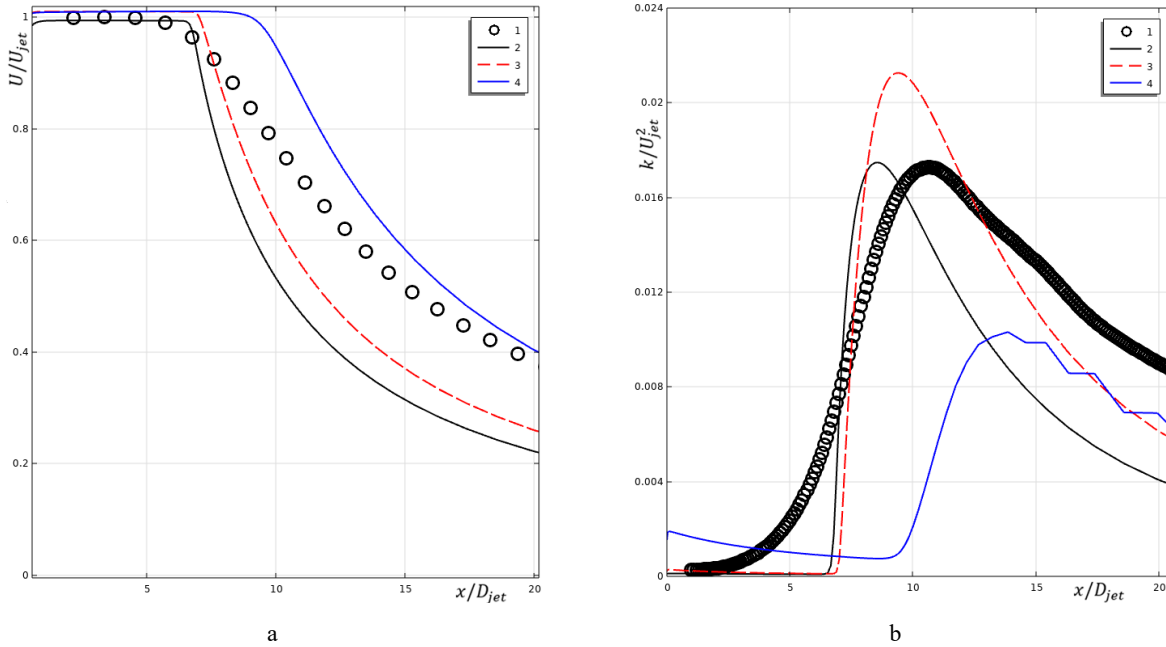
FIGURE 3. Comparison of numerical results of turbulence models with experimental data for dimensionless axial velocity (a) and turbulent kinetic energy (b) from the distance to the nozzle. 1-experimental data, 2- SST, 3- *k-ε*, 4-*k-ω*.

Figure 4 shows a comparison of the results for an axisymmetric hot subsonic jet using turbulence models with experimental data for turbulent stress profiles in different sections.

As can be seen from Figure 4, the model k-ε the turbulent stress model agrees well with the experimental results compared to other models.

Figure 5 shows a comparison of the results of an axisymmetric hot subsonic jet using a turbulence model with experimental data for dimensionless longitudinal velocity profiles at different distances from the nozzle.

Figure 5 shows the graphs of longitudinal velocities in the model *k-ε* agrees well with experimental results compared to other models.

Figure 6 shows a comparison of the results of an axisymmetric hot subsonic jet using a turbulence model with experimental data for dimensionless transverse velocity profiles at different distances from the nozzle.
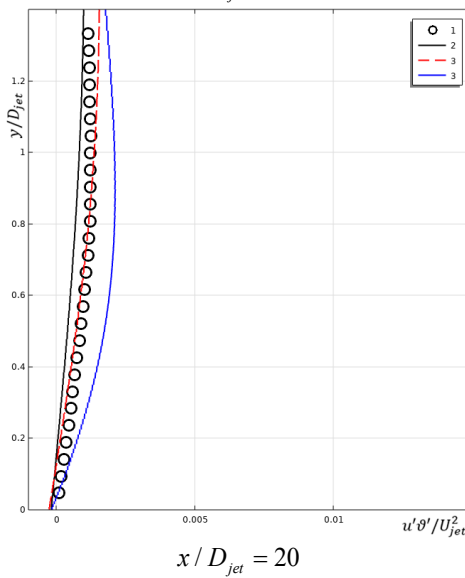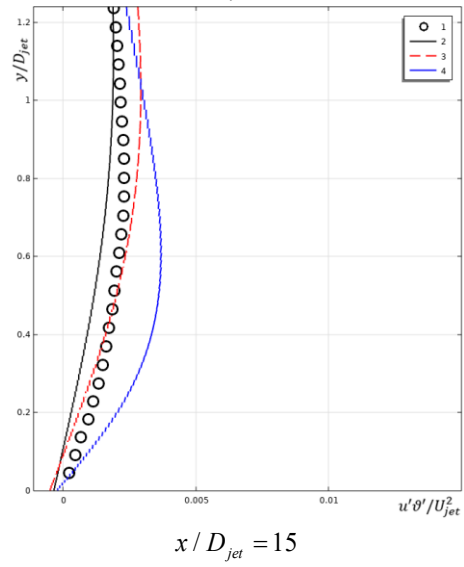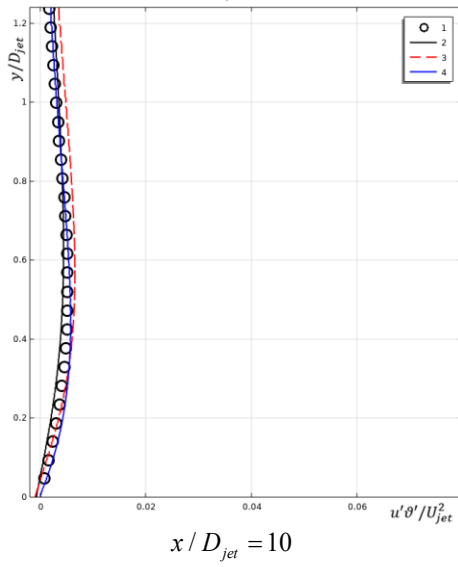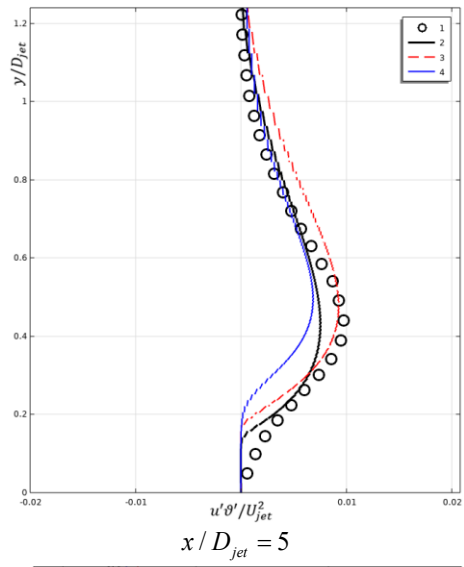
$x / D_{jet} = 2$

$x / D_{jet} = 5$

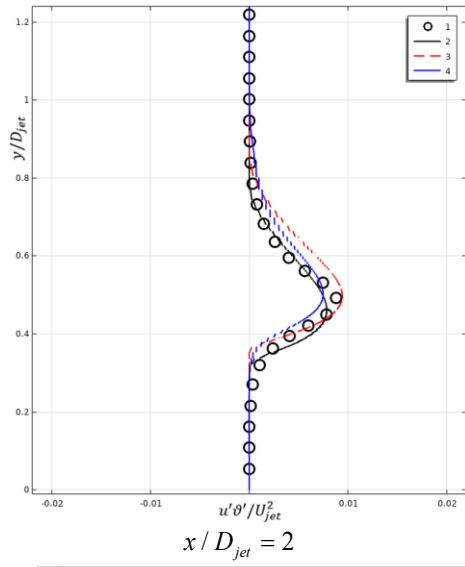$x / D_{jet} = 10$

$x / D_{jet} = 15$

$x / D_{jet} = 20$

FIGURE 4. Comparison of results for an axisymmetric hot subsonic jet using turbulence models with experimental data for the turbulent stress profile in different sections.

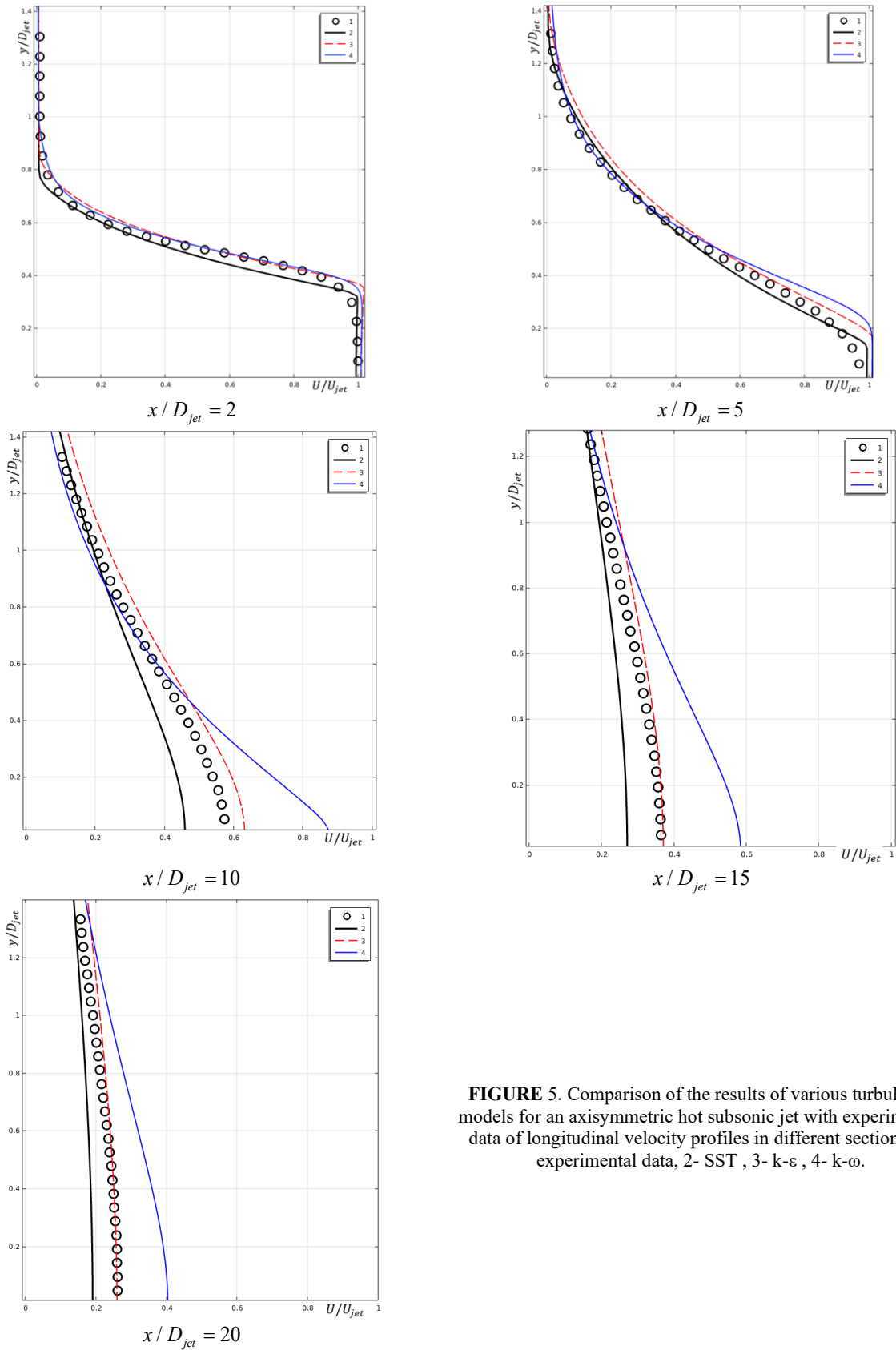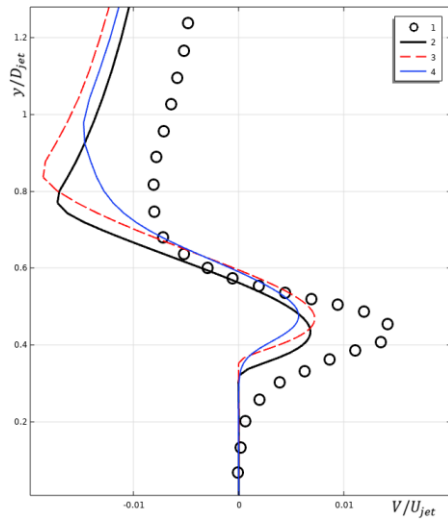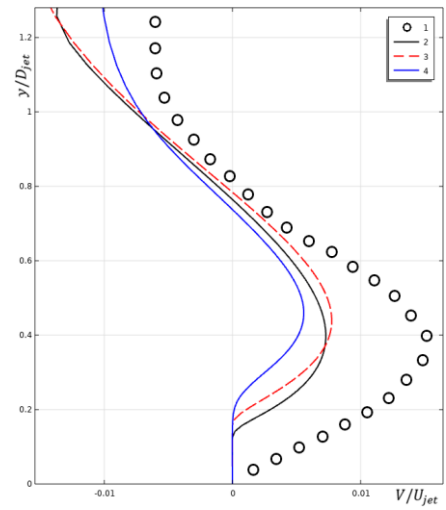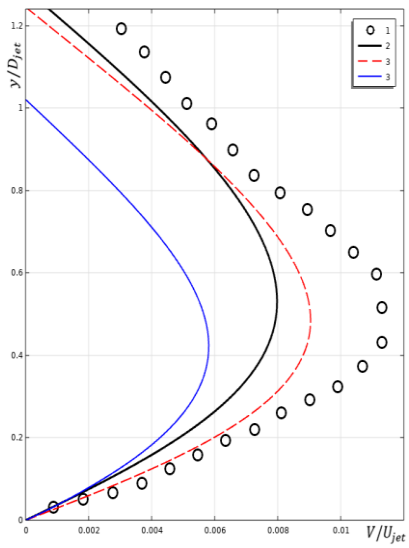1-experimental data, 2- SST, 3- k-ε, 4- k-ω.

$x/D_{jet} = 2$

$x/D_{jet} = 5$

$x/D_{jet} = 10$

$x/D_{jet} = 15$

$x/D_{jet} = 20$

FIGURE 5. Comparison of the results of various turbulence models for an axisymmetric hot subsonic jet with experimental data of longitudinal velocity profiles in different sections. 1- experimental data, 2- SST , 3- k-ε , 4- k-ω.
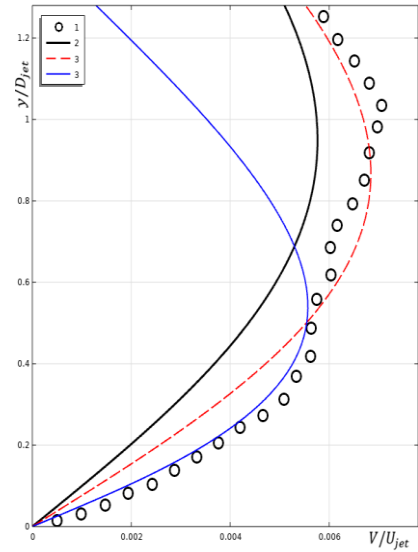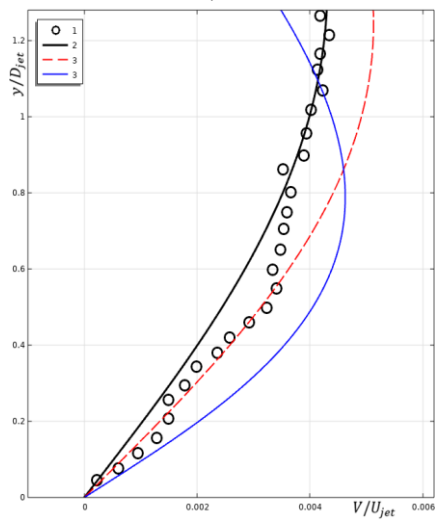
$x / D_{jet} = 2$

$x / D_{jet} = 5$

$x / D_{jet} = 10$

$x / D_{jet} = 15$

$x / D_{jet} = 20$

FIGURE 6. Comparison of the results of an axisymmetric hot subsonic jet using the model. Turbulence with experimental data, transverse velocity profiles for different cross-sections under experimental conditions.

1-exp, 2-SST, 3- k-ε, 4- $k - \omega$.

As can be seen from Fig. 6, the transverse velocity graphs in sections $\dfrac{x}{D_{jet}} = 2$ and $\dfrac{x}{D_{jet}} = 5$, the results in all

models do not numerically match the experiment. In the remaining sections, the numerical results of the model k-ε are closer to the experimental results.

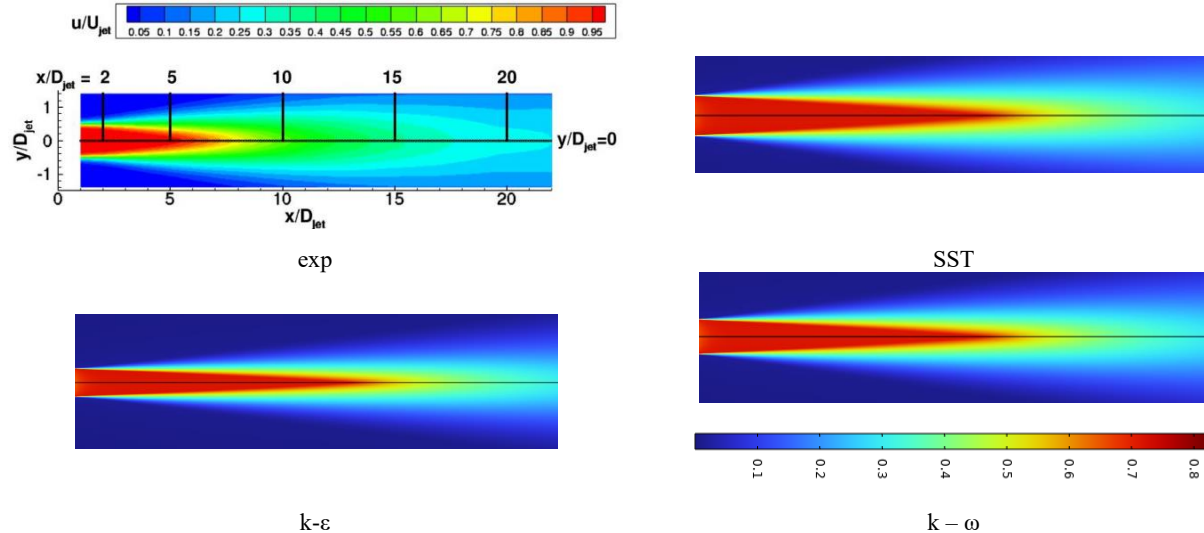Figure 7 shows the flow velocity contour line.

FIGURE 7. Comparison of the results of various turbulence models for an axisymmetric hot subsonic jet with experimental data for flow velocity isolines.

CONCLUSION

In this article, numerical solutions of the SST models k-ε were k-ω compared with experimental results. As can be seen from the obtained turbulent stress and longitudinal velocities, all models agree with the experimental results in the and sections. In the remaining sections, the model k-ε is closer to the experimental data than the other models. The calculation results show that the model has the minimum iteration time k-ε. These results suggest that the model is appropriate for such problems k-ε.

ACKNOWLEDGMENTS

REFERENCES

1. M. Alam, Journal of the Taiwan Institute of Chemical Engineers **144 (104742)**, 10–42 (2023).
2. J. Katz, Annu. Rev. Fluid Mech **38 (1)**, 27–63 (2006).
3. G. N. Abramovich, *Theory of Turbulent Jets* (Nauka, Moscow, 1984), 716 p.
4. D. C. Wilcox, *Turbulence Modeling for CFD* (DCW Industries Inc., 1994).
5. C. B. Jang, S. W. Choi and J. B. Baek, Int. J. Hydrogen Energy **40** (45), 15700–15710 (2015).
6. P. R. Spalart and S. R. Allmaras, AIAA Paper **92**–0439, 1–18 (1992).
7. M. L. Shur, M. K. Strelets, A. K. Travin and P. R. Spalart, AIAA J. **38** (5), 784–792 (2000).
8. F. R. Mentor, AIAA J. **32** (8), 1598–1605 (1994).
9. M. Tsubokura et al., SAE Int. J. Passeng. Cars – Mech. Syst **5 (2012-01-0448)**, 340–357 (2012).
10. Y. Okada et al., "Unsteady vehicle aerodynamics during a dynamic steering action: 1st report, on-road analysis," *SAE Technical Paper* №. 2012-01-0446 (2012).
11. A. D'Hooge et al., SAE Int. J. Passeng. Cars – Mech. Syst **7 (2014-01-0599),** 617–625 (2014).
12. J. Keogh et al., "Techniques for aerodynamic analysis of cornering vehicles," in *18th Asia Pacific Automotive Engineering* Conference Proceedings (Melbourne, Australia, 2015).

13. *9th ERCOFTAC/IAHR Workshop on Refined Turbulence Modelling* (Darmstadt University of Technology, Germany, 2001).
14. Nazarov F. K. et al. "Calculation of axisymmetric transonic jet with various turbulence models using Comsol multiphysics package" in *XV International Online Conference "Improving Farming Productivity and Agroecology – Ecosystem Restoration" (IPFA 2023)*, edited by Bozarov (AIP Publishing LLC, 2025) 040040.
15. Malikov Z., Nazarov F. "Numerical study of flow on a flat plate with zero pressure gradient based on a two-fluid model" in the *international conference: "Ensuring seismic safety and seismic stability of buildings and structures, applied problems of mechanics"* AIP Conf. Proc., edited by Abirov (AIP Publishing LLC, 2025) 060003.
16. Nazarov F. K. et al. Comp. Res. Modeling, **16(5)**, 1125-1142 (2024).
17. F. Nazarov et al., "Numerical calculation of turbulent flow in a two-dimensional flat diffuser based on the Comsol Multiphysics software," in *Proceedings of the 7th International Conference on Future Networks and Distributed Systems,* edited by Mohammad Hammoudeh (Association for Computing Machinery, New York NY United States, 2023), pp. 372–376.
18. Nazarov et al., "Numerical study of laminar flow using various difference schemes," in *Proceedings of the 7th International Conference on Future Networks and Distributed Systems,* edited by Mohammad Hammoudeh (Association for Computing Machinery, New York NY United States, 2023), pp. 377–383.
19. Malikov Z. M. et al., Comp. Res. Modeling. **T. 16, №. 2**, 395-408 (2024).
20. Malikov Z. M., Kh N. F., Herald Bauman MSTU, Ser. Nat. Sci., **2 (101)**, 22–35 (2022)
21. Malikov Z. M., Nazarov F. K. Comp. Res. Model. **13 (4)**, 793-805 (2021).
22. F. K. Nazarov, S. M. Khasanov and A. A. Yakubov, IJRTE **8 (4)**, 2140–2144 (2019).
23. Turbulence modeling Resource. NASA Langley Research Center. http://turbmodels.larc.nasa.gov
24. D. Anderson, J. Tannehill, and R. Pletcher, Computational Fluid Mechanics and Heat Transfer, Third Edition (2016).
25. Z. M. Malikov and F. X. Nazarov, Comp. Res. Modeling **13 (4)**, 793–805 (2021).
26. Z. M. Malikov and F. Kh. Nazarov, Herald Bauman MSTU, Ser. Nat. Sci., **2 (101)**, 22–35 (2022)
27. Z. M. Malikov, F. X. Nazarov and M. E. Madaliev, Vestn. Tomsk. Gos. Univ. Mat. Mechanika **78**, 125–142 (2022).
28. F. K. Nazarov, S. M. Khasanov and A. A. Yakubov, IJRTE **8 (4)**, 2140–2144 (2019).
29. Z. M. Malikov, F. K. Nazarov and S. Khaydarov, JACM **11 (4)**, 871–886 (2025).
30. W. P. Jones and B. E. Launder, Int. J. Heat Mass Transfer **15 (2)**, 301–314 (1972).
31. D. P. Navruzov et al., "Direct numerical simulation of flow in a flat channel with a rectangular step based on the nonstationary Navier–Stokes equations," in *the third international scientific conference construction mechanics, hydraulics and water resources engineering* AIP Conference Proceedings, edited by Bozorov (AIP Publishing LLC, 2023) 2612 (1), 030007.
32. A.N.Kolmogorov, "Equations of turbulent motion in an incompressible fluid," Dokl. Akad. Nauk SSSR **30**, 299–303 (1941).
33. D. C. Wilcox et al. Turbulence modeling for CFD. – La Canada, CA: DCW industries, 1998. – T. 2. – pp. 103-217.
34. P. Spalart, "Trends in turbulence treatments," in *Fluids 2000 Conference and Exhibit,* edited by Richard Wood (Denver, CO, U.S.A.2000), Paper No. 2306.
35. Z. M. Malikov et al., "Simulation of swirling flows based on modified two-fluid turbulence model," in *Proceedings of the 6th International Conference on Future Networks and Distributed Systems,* edited by Mohammad Hammoudeh (Association for Computing Machinery, New York NY United States, 2022), pp. 27–32.