Journal of Construction and Engineering Technology
Journal homepage: www.journal.ferpi.uz

MATHEMATICAL MODELING OF SWIRLING JET STREAMS WITH VARYING DEGREES OF SWIRLING

Dilshod Navruzov

Bukhara Institute of Natural Resources Management, National Research University "TIIAME", Bukhara, Uzbekistan

*Correspondence: navruzov.d@mail.ru

Abstract

In this article, a swirling jet is studied on the basis of a two-fluid turbulence model. This task, despite its simplicity, is quite a difficult problem for many turbulence models. This model is derived based on the dynamics of two-liquids. These papers also show that the developed two-fluid model is able to adequately describe complex anisotropic turbulence. It is shown that the results of the two-fluid model, despite the use of a rather coarse computational grid, are in satisfactory.

Keywords: k swirled jet, two-fluid model, counter-flow scheme, degree of twist, uniform mesh.

Introduction.

Swirling flows, due to their numerous practical applications, represent an important branch of modern hydrodynamics. The complexity of these currents stimulates the use of the most modern research methods in their study. Numerical modeling methods are being widely mastered, various fine measurement methods, including non-contact ones, which are no longer the property of individual research laboratories, are becoming increasingly widespread and allow us to hope for significant progress in the study and prediction of the properties of complex hydrodynamic processes, including flows with twist [1,2]. Swirling currents are widespread. They are observed in many natural phenomena and are often used in technology. As an example, we can point to cyclones and tornadoes in the atmosphere, to whirlpools in rivers and reservoirs, to the vortex wake left by an airplane wing, to separators and cyclone-type heat exchangers, to sprinklers and agricultural machines spraying fertilizers, to jet pumps and chemical reactors with vortex flow. Even when analyzing the flight of a boomerang and a bee, elements of the theory of swirling flows are used. In technical combustion devices, the significant and beneficial effect exerted by the swirling of the 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, industrial furnaces, boilers, as well as a number of other heating systems. As experimental studies show, swirl radically affects the flow field - its important properties and characteristics, such as the propagation and attenuation of jets 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, reported to thread[3,4].

The influence of the initial flow swirl on the flow field increases sharply with increasing degree of swirl (expressed by the swirl parameter S or the blade installation angle). The effect of weak swirl (S<0.4) is reduced to an increase in the width of the free or restricted jet. An increase in the width of the jet and a decrease in the speed in the jet occur more intensely with an increase in the degree of swirl [5-8]. For applied problems with combustion, one of the most significant and useful phenomena in swirling jet flows 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 direct output). Numerical studies of swirling flows with

Published online: 5 August 2024
https://doi.org/10.5281/zenodo.13893318

© FarPI Hitech Engineer

standard two-equation k-ε turbulence models have shown that these models can significantly underestimate mixing rates. Therefore, to improve the results, modifications of existing models are intensively carried out [9-12]. Methods such as Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) are labor-intensive and use them to solve complex aerodynamic engineering problems require high-speed computers. Therefore, their widespread 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 still remain the main working tool for solving applied problems of aerodynamics.

Another approach to describing turbulence is the so-called two-fluid approach. This approach was proposed by Spaulding [13-14]. Using this approach, the Rayleigh–Taylor instability was studied in [15], the combustion process in [16], turbulent stratified flows in [17], bounded and free turbulent shear flows in [18], etc. Further development of the two-fluid approach is demonstrated in [19-20]. These works demonstrate the ability of the two-fluid approach to describe with great accuracy the turbulent processes of momentum and heat transfer in anisotropic turbulent flows. Therefore, the goal of this work is to use a two-fluid model to study such a complex problem as a swirling jet.

Formulation of the problem.

The main equations for studying the problem posed are the hydrodynamic equations of the two-fluid model for an incompressible medium [19]

$$\begin{cases} \dfrac{\partial V_i}{\partial x_i}=0, \\ \dfrac{\partial V_i}{\partial t}+V_j\dfrac{\partial V_i}{\partial x_j}+\dfrac{1}{\rho}\dfrac{\partial p}{\partial x_i}=\dfrac{\partial}{\partial x_j}\left[\nu\left(\dfrac{\partial V_i}{\partial x_j}+\dfrac{\partial V_j}{\partial x_i}\right)-\vartheta_j\vartheta_i\right], \\ \dfrac{\partial \vartheta_i}{\partial t}+V_j\dfrac{\partial \vartheta_i}{\partial x_j}=-\vartheta_j\dfrac{\partial V_i}{\partial x_j}+\dfrac{\partial \sigma_{ji}}{\partial x_j}+F_{fi}+F_{\perp i}, \\ \sigma_{ji}=\nu_{ij}\left(\dfrac{\partial \vartheta_i}{\partial x_j}+\dfrac{\partial \vartheta_j}{\partial x_i}\right), \quad \nu_{ij}=3\nu+2\dfrac{\left|\vartheta_i\vartheta_j\right|}{def\left(\vec{V}\right)} \quad for\ i\neq j,\ \nu_{ii}=3\nu+2\dfrac{\left\langle\vartheta^2\right\rangle}{def\left(\vec{V}\right)}, \\ \left\langle\vartheta^2\right\rangle=\sum_{k=1}^{3}\left|\dfrac{\partial \vartheta_k}{\partial x_k}\right|\vartheta_k^2\Big/\sum_{k=1}^{3}\left|\dfrac{\partial \vartheta_k}{\partial x_k}\right|, \quad \vec{F}_f=-K_f\vec{\vartheta}, \ \vec{F}_\perp=C_s rot\vec{V}\times\vec{\vartheta}. \end{cases} \tag{1}$$

In the given system of equations, V_i is the component of the average flow velocity, ϑ_i is the component of relative velocity, ν_{ji} are the effective molar viscosities, p is the pressure, ρ is the density of the medium, ν is the molecular viscosity, K_f is the coefficient of friction, C_s is the coefficient in the Saffman force [20], $def\left(\vec{V}\right)$ is the rate of complete strain, determined as:

$$def\left(\vec{V}\right)=\sqrt{2S_{ij}S_{ij}}, \quad S_{ij}=\frac{1}{2}\left(\frac{\partial V_i}{\partial x_j}+\frac{\partial V_j}{\partial x_i}\right). \tag{2}$$

The coefficient of friction is determined from the following relation

$$K_f=C_1\lambda_{real}+C_2\frac{|d\cdot\vartheta|}{d^2}. \tag{3}$$

In this expression, d is the nearest distance to the solid wall, λ_{real} is the real part of the largest root of the characteristic equation

$$\det(A-\lambda E)=0 \tag{4}$$

where E is the identity matrix and A is the matrix equal to

$$A=\begin{vmatrix} -\dfrac{\partial V_1}{\partial x_1} & -\dfrac{\partial V_1}{\partial x_2}-C_s\zeta_3 & -\dfrac{\partial V_1}{\partial x_2}+C_s\zeta_2 \\ -\dfrac{\partial V_2}{\partial x_1}+C_s\zeta_3 & -\dfrac{\partial V_2}{\partial x_2} & -\dfrac{\partial V_2}{\partial x_3}-C_s\zeta_1 \\ -\dfrac{\partial V_3}{\partial x_1}+C_s\zeta_2 & -\dfrac{\partial V_3}{\partial x_2}+C_s\zeta_1 & -\dfrac{\partial V_3}{\partial x_3} \end{vmatrix}, \tag{5}$$

$$\zeta=rot\vec{V}.$$

Constants of the model are $C_s=0.2,\ C_1=0.7825, C_2=0.3.$

A two-fluid model will be used to study a swirling jet flowing into a flooded space. It is known that a swirling jet is characterized by such parameters as the degree of swirl S and the Reynolds number Re. In the problem under consideration, a swirling jet flows into a flooded space from a round nozzle and these parameters are equal

$$\mathrm{Re}=\frac{DU_0}{\nu}=56000, \quad S=\frac{\int_0^R V_{0\varphi}V_{0z}r^2dr}{R\int_0^R V_{0z}^2rdr} \tag{6}$$

Here are D - the diameter and R - radius of the nozzle, U_0- the maximum longitudinal flow velocity at the outlet of the pipe, ν - kinematic viscosity, V_{0z}, $V_{0\varphi}$,- the longitudinal and tangential flow velocities at the outlet of the nozzle.

To study an axisymmetric swirling jet, we write the system of equations (1) in cylindrical coordinates

$$\begin{cases}
\frac{\partial V_z}{\partial z}+\frac{\partial rV_r}{r\partial r}=0\\
\frac{\partial V_z}{\partial t}+V_z\frac{\partial V_z}{\partial z}+V_r\frac{\partial V_z}{\partial r}+\frac{1}{\rho}\frac{\partial p}{\partial z}=\frac{1}{\mathrm{Re}}(\frac{\partial^2V_z}{\partial r^2}+\frac{\partial V_z}{r\partial r})-\frac{\partial r\vartheta_r\vartheta_z}{r\partial r}\\
\frac{\partial V_r}{\partial t}+V_z\frac{\partial V_r}{\partial z}+V_r\frac{\partial V_r}{\partial r}+\frac{1}{\rho}\frac{\partial p}{\partial r}=\frac{V_\phi^2}{r}+\frac{1}{\mathrm{Re}}(\frac{\partial^2V_r}{\partial r^2}+\frac{\partial V_r}{r\partial r}-\frac{V_r}{r^2})-\frac{\partial r\vartheta_r\vartheta_r}{r\partial r}\\
\frac{\partial V_\phi}{\partial t}+V_z\frac{\partial V_\phi}{\partial z}+V_r\frac{\partial V_\phi}{\partial r}+\frac{V_rV_\phi}{r}=\frac{1}{\mathrm{Re}}(\frac{\partial^2V_\phi}{\partial r^2}+\frac{\partial V_\phi}{r\partial r}-\frac{V_\phi}{r^2})-\frac{\partial r^2\vartheta_r\vartheta_\phi}{r^2\partial r}\\
\frac{\partial \vartheta_z}{\partial t}+V_z\frac{\partial \vartheta_z}{\partial z}+V_r\frac{\partial \vartheta_z}{\partial r}=-(1-C_s)\frac{\partial V_z}{\partial r}\vartheta_r+\frac{1}{r}\frac{\partial}{\partial r}(r\nu_{zr}\frac{\partial \vartheta_z}{\partial r})-C_r\vartheta_z\\
\frac{\partial \vartheta_r}{\partial t}+V_z\frac{\partial \vartheta_r}{\partial z}+V_r\frac{\partial \vartheta_r}{\partial r}=-C_s\frac{\partial V_z}{\partial r}\vartheta_z-(C_s\frac{\partial rV_\phi}{r\partial r}-\frac{2V_\phi}{r})\vartheta_\phi+\\
+\frac{2}{r}\frac{\partial}{\partial r}(r\nu_{rr}\frac{\partial \vartheta_r}{\partial r})-2\nu_{\phi\phi}\frac{\vartheta_r}{r^2}-C_r\vartheta_r\\
\frac{\partial \vartheta_\phi}{\partial t}+V_z\frac{\partial \vartheta_\phi}{\partial z}+V_r\frac{\partial r\vartheta_\phi}{r\partial r}=-(1-C_s)\frac{\partial rV_\phi}{r\partial r}\vartheta_r+\frac{1}{r^2}\frac{\partial}{\partial r}[r^2\nu_{r\phi}(\frac{\partial \vartheta_\phi}{\partial r}-\frac{\vartheta_\phi}{r})]-C_r\vartheta_\phi
\end{cases} \tag{7}$$

The experimental values of the flow parameters at the nozzle exit were approximated by the following expressions

$$V_{0\phi}=2Sr^2(4+n_w)(1-r)^{n_w}$$
$$V_{0z}=(1+x_k)(1-r)^{x_k}$$
$$n_w=4ux_k=0.06 \tag{8}$$

The relative velocities at the nozzle exit were equal

$$v_z=0.025, \quad v_r=0.1r^{0.75}, \quad v_\varphi=0.1r^{0.75}.$$

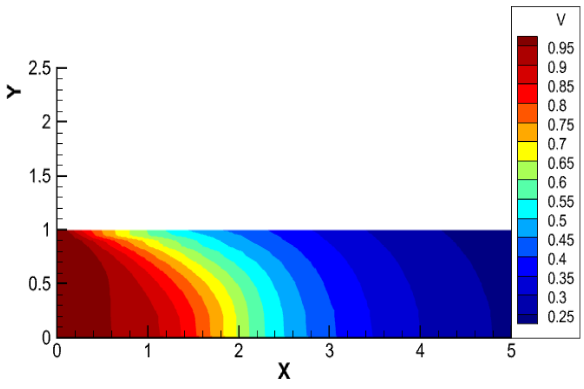
For calculations, a grid size of 250x250 with steps $\Delta z=0.2$, $\Delta r=0.02$ was used. The equations were brought into dimensionless form by relating all velocities to U_0, and the spatial scales to the radius of the pipe. Integration over time was carried out with a dimensionless step $\Delta t=0.02$.

To ensure stability when solving the equations of system (3), the explicit difference scheme against the Leonardo flux QUICK [22] was used in the convective terms, and the central difference in an implicit form was used for the diffusion terms. It is known that Leonardo's QUICK scheme has third-order accuracy for structured computational grids. Consequently, the calculation scheme had an accuracy of the first order in time and second in space. To implement the semi-implicit scheme, the sweep method was used. The relationship between pressure and velocities was carried out using the SIMPLE procedure [23], where the iterative lower relaxation method was used [24].

Numerical results and discussion

Figure 1. **a** and **b** show the results of calculating the longitudinal velocity profile with weak turbulence, where the degree of twist is S=0.1 and S=0.2.

a)



b)

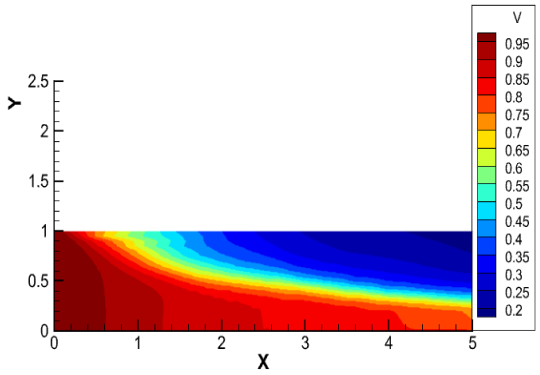
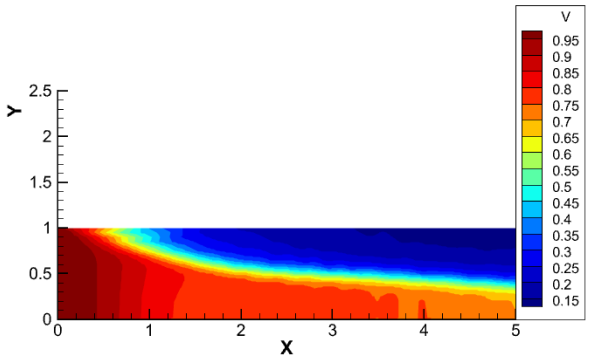


Fig.1. Longitudinal velocity profile at low twist rate.

Figure 2. **a** and **b** show the results of calculating the longitudinal velocity profile with weak turbulence, where the degree of twist is S=0.3 and S=0.4.

a)



b)

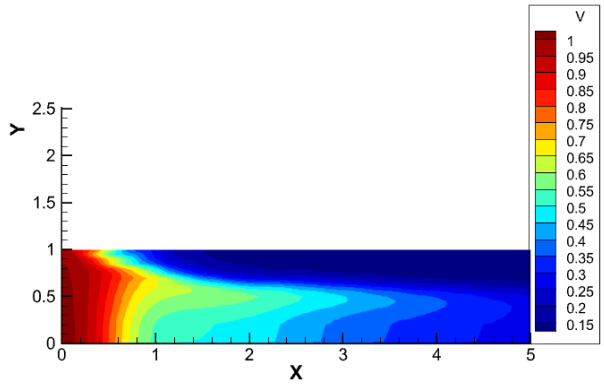
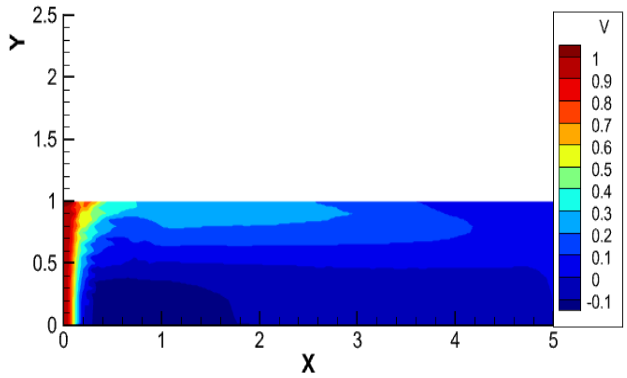


Fig.2. Longitudinal velocity profile at moderate twist rate

Similar results for the tangential profile at a high degree of swirl are presented in Figure 3. **a** and **b** show the results of calculating the longitudinal velocity profile with weak turbulence, where the degree of twist is S=0.5 and S=0.6.

a)



b)

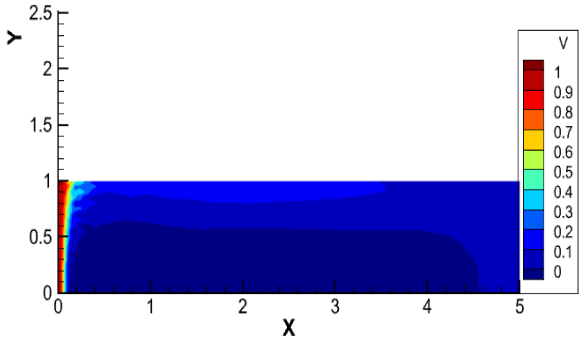


Fig.3. Longitudinal velocity profile at a high degree of twist.

Conclusion

In this work, a two-fluid turbulence model is used to study a swirling jet. The results obtained show that the turbulence model is able to adequately describe the dynamics of a swirling jet, despite the use of a coarse uniform computational grid. This circumstance is one of the advantages of the proposed model. Because many RANS models require high resolution computational grids and fairly small-time integration steps. As for the two-fluid model, integration can be carried out in significantly large time steps. The next point, as shown in [19], is that the two-fluid model does not suffer from the "round jet" anomaly. The meaning of this anomaly is that almost all RANS models for a round jet give a flow expansion almost one and a half times greater than the experimental data. Therefore, in all RANS models, special corrections are introduced to eliminate this anomaly. The two-fluid model can give better results if a finer computational mesh is used or if it is refined at the nozzle exit and in the boundary layer region

References

[1]. James D. Chenometh., Chandrasekhar Kannepalli.,Srinivasan Arunajatesan., and Ashvin Hosangadi "Modeling swirling jet flows using a hubrid RANS/LES methodology" ResearchGate, (2008) https://doi 10.2514/6.2008-4746.

[2]. Malikov Z.M., Navruzov D.P., Adilov K., Jurayev S,R. Numerical study of a circular jet based on a modern turbulence model // «Computer Applications for Management and Sustainable Development of Production and Industru» (CMSD2021). doi:10.1117/12.2631607.

[3]. Arunajatesan, S. and Sinha, N., "Hybrid RANS-LES Modeling for Cavity Aeroacoustics Predictions," *International Journal of Aeroacoustics*, Vol. 2, No. 1, pp 65-91, 2003.

[4]. Arunajatesan, S.A., and Dash, S.M., "Progress Towards Hybrid RANS-LES Modeling For High-Speed Jet Flows," Paper No. AIAA-2002-0428, 40th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, January 14-17, 2002.

[5] Sinha, N., Hosangadi, A., and Dash, S.M., "The CRAFT NS Code and Preliminary Applications to Steady/Unsteady Reacting, Multi-phase Jet/Plume Flowfield Problems," CPIA Pub. 568 May 1991.

[6]. Rai, M.M., and Moin, P.," Direct Numerical Simulation of Transition and Turbulence in a Spatially Evolving Boundary Layer," *Journal of Computational Physics*, Vol. 109, pp.1 62-192, 1993.

[7]. Forsythe, J.R., "Numerical Computation of Turbulent Separated Supersonic Flow Fields", *Ph.D Thesis*, Wichita State University, 2000.

[8]. B.M. Kholboev, D.P. Navruzov, D.S. Asrakulova, N.R. Engalicheva and A.A. Turemuratova. Comparison of the results for calculation of vortex currents after sudden expansion of the pipe with different diameters // *Int. J. of Applied Mechanics and Engineering, 2022, vol.27, No.2, pp.115-123.* DOI: 10.2478/ijame-2022-0023.

[9]. M. G. Hall, Vortex breakdown, Ann. Rev. Fluid Mech. 4 (1972), 195–218.

[10]. A. K. Gupta, D. G. Lilley., N. Syred, Swirl Flows, Abacus Press, Cambridge, MA, 1984.

[11] O. Lucca-Negro, T. O'Doherty, Vortex breakdown: A review, Prog. Energy Combust. Sci. 27 (4) (2001), 431–481.

[12]. K. Oberleithner, C. O. Paschereit, R. Seele, I. Wygnanski, Formation of turbulent vortex breakdown: Intermittency, criticality, and global instability, AIAA J. 50 (7) (2012), 1437–1452.

[13]. Spalding D.B. A turbulence model for buoyant and combusting flows. 4-th Int. Conf. on Numerical methods in Thermal Problems, Swansea, 15-18 July 1984. Also, as Imperial College report CFD/86/4 (1984).

[14]. Spalding D.B. Chemical reaction in turbulent fluids. J. Physico-chemical Hydrodyn., 4 (1983), pp. 323 – 336.

[15]. G. A. Hoshoudi. Rayleigh-Taylor instability in a magnetized quantum plasma taking into account viscosity. Plasma Physics, 2011, Volume 37, No. 9, pp. 831-840

[16]. Markatos N.C. Kotsifaki C.A. One-dimensional, two-fluid modelling of turbulent premixed flames. Appl. Math. Modell, 18 (1994), pp. 646 – 657.

[17]. Y.M. Shen, C.-O. Ng, A.T. Chwang. A two-fluid model of turbulent two-phase flow for simulating turbulent stratified flows. Ocean Eng., 30 (2003), pp. 153 – 161

[18]. Z.M. Malikov., D.P. Navruzov., X. Djumayev. Models results Comparison of different approaches to turbulence for flow past a heated flat plate" E3S Web of Conferences 264, 01008 (2021) https://doi.org/10.1051/e3sconf/202126401008

[19]. Malikov Z.M. Mathematical Model of Turbulence Based on the Dynamics of Two Fluids. Applied Mathematical Modelling 82 (2020) pp.409–436.

[21]. M. Madaliyev., D. Navruzov., Y. Hamrayev, T. Pulatov., S. Juraev. Study of the Spalart-Allmares Turbulence Model for the Calculation of a Centrifugal Separator// Problems in the Textile and Light Industry in the Context of Integration of Science and Industry and Ways to Solve Them AIP Conf. Proc. 3045, 020005-1–020005-7; https://doi.org/10.1063/5.0197352 Published by AIP Publishing. 978-0-7354-4889-6/$30.00.

[22]. Leonard B P 1979 A stable and accurate convective modeling procedure based on quadratic upstream interpolation. Comp Meth Appl Mech Eng 19, pp 59–98.

[23]. Patankar S.V. Numerical Heat Transfer and Fluid Flow. Taylor&Francis. ISBN 978-0-89116-522-4, 1980.

[24]. Андерсон *Д*, Таннехилл *Дж*, Плетчер Р 1990 Вычислительная гидромеханика и теплообмен. -М.: Мир, Т.1

[25]. Malikov Z.M., Madaliyev M.E., Navruzov D.P., Adilov K. Numerical study of an axisymmetric jet based on a new turbulence model // International Scientific Conference on Actual Problems of Applied Mechanics, AIP Conference Proceedings. DOI.org/10.1063/5.0118473.