BIS ENERGY AND ENGINEERING

Volume 3 2026 | DOI: 10.31603/biseeng.658
7th Borobudur International Symposium on
Science and Technology (BIS-STE) 2025

UNIMMA PRESS
UNIVERSITAS MUHAMMADIYAH MAGELANG

Rans-based turbulence modeling of taylor–couette flow: Simulation of vortex formation using the geko model

Murodil Erkinjon Madaliev[1*], Abdirasul Xolmurzayev Islomjon Tohirov[1], and Mukhlisa Rustamova[1]

[1] Fergana State Technical University, Fergana, Uzbekistan
*Corresponding author email: madaliev.me2019@mail.ru

Abstract

This study presents a numerical investigation of Taylor–Couette flow using the ANSYS Fluent computational fluid dynamics (CFD) platform. The analysis focuses on the formation, stability, and evolution of vortex structures generated in the annular region between coaxial cylinders due to the rotation of the inner cylinder. Turbulence is modeled using the Reynolds-Averaged Navier–Stokes (RANS) approach coupled with the Generalized k–ω (GEKO) turbulence model, which provides enhanced flexibility for simulating complex swirling and shear-driven flows. The simulation results successfully capture the onset and development of Taylor vortices as the rotational speed increases. Characteristic vortex cells, secondary flow patterns, and transitions toward more complex flow structures are observed, consistent with classical Taylor–Couette behavior. The results demonstrate that the GEKO model effectively represents flows influenced by curvature effects, centrifugal instabilities, and enhanced mixing. These findings contribute to a better understanding of hydrodynamic stability and have practical implications for engineering and environmental applications, including mixing enhancement, heat and mass transfer optimization, energy-efficient reactor design, and water management systems in agroecological environments.

Keywords

Mathematical modeling, GEKO turbulence models, Reynolds-averaged Navier-Stokes equations, Ansys fluent

Published:
May 04, 2026

Selection and Peer-review under the responsibility of the 7th BIS-STE 2025 Committee

Introduction

In the context of growing water shortages associated with climate change, population growth and expansion of agricultural production, issues of efficient water management are becoming an integral part of sustainable development. This problem is especially acute in regions with arid climates, where water supply directly affects crop yields, soil fertility and the preservation of ecosystems. In this regard, in recent decades, interest has increased significantly in the development of scientifically based methods for regulating the water balance in agroecological systems, including both natural and engineering components. One of the key tasks in this area is to ensure rational distribution of moisture in the soil profile, as well as increasing the efficiency of filtration and infiltration processes. In addition, uniform water supply to the root system of plants is of great importance, which reduces moisture loss, minimizes water erosion and increases the resilience of agroecosystems to climatic stress. Achieving these goals is impossible without a deep understanding of the fundamental processes occurring in liquids as they move through complex geometric and physical environments. Hydrodynamics and mass transport in such systems are often accompanied by the formation of turbulent and vortex structures, which play an important role in mixing, transport of dissolved substances and energy. In this context, modeling flows with stable and controlled vortices, such as the Taylor-Couette flow, opens up new possibilities for the analysis and optimization of water flows in agricultural and natural settings [1–5].

One of such complex flows, of interest from both theoretical and applied points of view, is the Taylor-Couette flow. This flow occurs between two coaxial cylinders when one of them (usually the inner one) rotates relative to the other. When a certain critical Reynolds number is reached, characteristic annular vortices, known as Taylor vortices, are formed in the gap between the cylinders. Further, with increasing rotation speed, the flow becomes increasingly unstable and passes into a turbulent regime with a spatially structured shape [6–8]. Such vortex structures that arise in the Taylor-Couette flow find wide practical application in various branches of science and technology due to their ability to provide intensive mixing, increased mass and heat transfer, and resistance to flow fluctuations. Their versatility is due to the fact that such structures can be generated in compact geometries under controlled conditions, which makes them especially valuable in engineering systems where space and energy costs are strictly limited.

In the chemical industry, Taylor vortices are used to intensify the processes of mixing reagents, accelerate the dissolution of substances and increase the efficiency of heat exchangers. Due to the high velocity and pressure gradients that arise in these vortex zones, the kinetic characteristics of the reactions are significantly improved. This allows optimizing production cycles and reducing energy consumption.

In mechanical engineering, vortex flows are used in various types of mixing and centrifugal devices, where high efficiency with minimal dimensions is important. Due to the compactness of the Taylor-Couette configuration, such devices can be integrated into closed systems, providing reliable mixing without the need for large moving parts.

In biotechnology and medicine, especially in the design of bioreactors, stable vortex structures are used to create a uniform distribution of nutrients and oxygen throughout the reactor volume. This is critical for the vital activity of cell cultures, accelerated growth of microorganisms and increased productivity of biological processes.

In wastewater treatment systems, vortices are used to enhance flow turbulence, which contributes to more efficient separation of pollutants, sedimentation of solid particles and uniform distribution of reagents throughout the reactor volume. This is especially valuable in installations with limited volume, where it is necessary to combine clarification and aeration processes in one working area [9–11].

Finally, in agroecological practice, vortex flows are considered a promising direction in the creation of closed irrigation systems, filtration columns, hydroponic installations and other engineering solutions for sustainable agriculture. The ability to control the structure and intensity of the flow allows adapting water supply systems to specific conditions - soil, climatic or technological. Thus, the use of Taylor-type vortex structures opens up new horizons in agricultural engineering and eco-technologies.

In agroecology, it is especially important to control the flow structure in artificial or natural water systems: soil filtration layers, irrigation pipelines, storage tanks. The use of models based on the analysis of vortex flows can significantly improve the efficiency of these systems due to a better understanding of the distribution of energy and momentum in the flow [12–14].

In this paper, we consider the modeling of the Taylor-Couette flow using the ANSYS Fluent software package. To model turbulence, the Reynolds averaging method (RANS) is used, in particular, the universal GEKO model, which allows you to adapt the turbulence model to specific problem conditions. The main goal of the study is to study the conditions for the emergence of Taylor vortices and their characteristics, as well as to identify possible ways to apply the obtained results in water flow control problems in agroecological settings.

Mathematical and physical formulation of the problem

The mathematical and physical formulation of the problem begins with Couette flow, which was initially examined by Maurice Couette in 1890 in relation to viscosity studies. Later, in 1923, Geoffrey E. Taylor identified and explained the cellular flow pattern that emerges when a cylinder rotates. Taylor-Couette flow refers to the motion of fluid confined between two concentric cylinders, where the inner cylinder, and in some cases the outer cylinder, rotates around the same central axis. This type of flow is a fundamental phenomenon in fluid mechanics and has been investigated for more than a century. When the outer cylinder remains stationary, the flow gradually changes as the rotational speed of the inner cylinder increases from rest, progressing from a purely laminar flow to vortex flow and eventually to turbulent vortex flow. The behavior of a Newtonian fluid within an annular gap can be described using the rotational Reynolds number.

$$\mathrm{Re} = \frac{\omega r_i \left(r_i - r_0 \right)}{\nu} \tag{1}$$

In this equation, ω denotes the angular velocity of the inner cylinder, while r_i and r_o represent the radii of the inner and outer cylinders, respectively. Meanwhile, ν refers to the kinematic viscosity of the fluid. Taylor-Couette flow describes the fluid motion that occurs between two coaxial cylinders rotating independently. The configuration of this system is illustrated schematically in Figure 1. The difference in rotational motion between the two cylinders induces shear in the fluid, which subsequently drives the flow within the system [1].

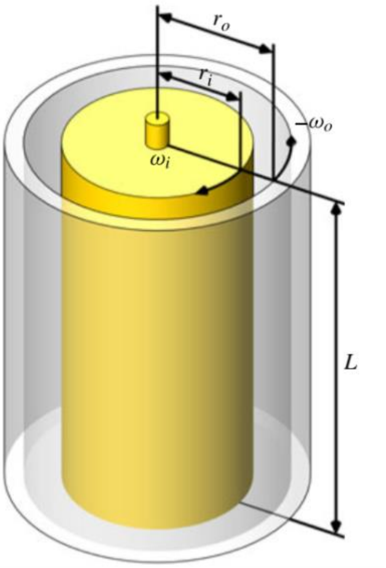


Figure 1. Schematic diagram of the Taylor-Couette system. The system consists of two coaxial cylinders with an inner cylinder radius r_i and an outer cylinder radius r_o. Both cylinders have a length L. The inner cylinder rotates with an angular velocity ω_i, and the outer cylinder rotates with an angular velocity ω_o; in our case, it is stationary.

The inner cylinder has a diameter of 6.985 cm, and the outer cylinder has a diameter of 9.525 cm, resulting in an annular gap width of 1.27 cm between the cylinders. Both cylinders were made of plexiglass, and the outer cylinder was enclosed in a square plexiglass box. The length of the cylinders was 10 cm. The inner cylinder rotates with an angular velocity of ωi = 23.01 rad/s. The density of the working fluid was 1.85 g/cm3, and the viscosity of the working fluid was 1.71×10−6 m2/s.

For the numerical study of the problem, the Reynolds-averaged Navier-Stokes equation system is used [15–17].

$$\begin{cases} \dfrac{\partial \rho}{\partial t} + \dfrac{\partial\left(\bar{U}_i\right)}{\partial x_j} = 0, \\ \dfrac{\partial\left(\rho \bar{U}_i\right)}{\partial t} + \dfrac{\partial\left(\rho \bar{U}_i \bar{U}_j\right)}{\partial x_j} = -\dfrac{\partial p}{\partial x_i} + \rho \bar{\mathrm{G}} + \dfrac{\partial}{\partial x_j}\left[\left(\mu + \mu_t\right)\left(\dfrac{\partial \bar{U}_i}{\partial x_j} + \dfrac{\partial \bar{U}_j}{\partial x_i}\right)\right]. \end{cases} \tag{2}$$

The GEKO (Generalized k-ω) turbulence model, developed by Ansys [18–23], is an enhanced version of the standard SST (Shear Stress Transport) model. It was designed to provide greater flexibility and accuracy in simulating turbulent flows. This model is especially suitable for complex flow problems involving swirling motion, mixed flow regimes, and regions with strong velocity and pressure gradients. Therefore, it is commonly applied in systems such as hydro cyclones, centrifuges, and other equipment where the complex characteristics of turbulence must be carefully considered.

$$\begin{cases} \dfrac{\partial(\rho k)}{\partial t}+\dfrac{\partial\left(\rho\bar{U}_j k\right)}{\partial x_j}=P_k-C_\mu\rho k\omega+\dfrac{\partial}{\partial x_j}\left[\left(\mu+\dfrac{\mu_t}{\sigma_k}\right)\dfrac{\partial k}{\partial x_j}\right], \\ \dfrac{\partial(\rho\omega)}{\partial t}+\dfrac{\partial\left(\rho\bar{U}_j\omega\right)}{\partial x_j}=C_{\omega1}F_1\dfrac{\omega}{k}P_k-C_{\omega2}F_2\rho\omega^2+\rho F_3CD+\dfrac{\partial}{\partial x_j}\left[\left(\mu+\dfrac{\mu_t}{\sigma_\omega}\right)\dfrac{\partial\omega}{\partial x_j}\right]. \end{cases} \tag{3}$$

Turbulent eddy viscosity is calculated by: $\mu_t=\rho v_t=\rho\dfrac{k}{\max\left(\omega,S/C_{Realize}\right)}$.

$$P_k=-\tau_{ij}\frac{\partial U_i}{\partial x_j}, \tag{4}$$

$$\tau_{ij}^{EV}=-\rho\overline{u_i^{'}u_j^{'}}=\mu_t 2S_{ij}-\frac{2}{3}\rho k\delta_{ij}, \tag{5}$$

$$CD=\frac{2}{\sigma_\omega}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j}, \tag{6}$$

$$\tau_{ij}=\tau_{ij}^{EV}-C_{CORNER}\frac{1.2\mu_t}{\max\left(0.3\omega,\sqrt{0.5\left(S^2+\Omega^2\right)}\right)}\left(S_{ik}\Omega_{kj}-\Omega_{ik}S_{kj}\right), \tag{7}$$

$$S_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right),\ \Omega_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}-\frac{\partial U_j}{\partial x_i}\right), \\ S=\sqrt{2S_{ij}S_{ij}},\ \Omega=\sqrt{2\Omega_{ij}\Omega_{ij}}. \tag{8}$$

The remaining coefficients and functions were presented in the article. [20].

Method of solution

For numerical modeling of the Taylor-Couette flow in the Ansys Fluent software package, the implicit solution method was used, which ensures the stability of calculations when modeling high-speed flows. Roe-FDS was selected as the flow calculation scheme, which allows for accurate modeling of shock waves and supersonic effects. For spatial discretization, the second-order scheme (Second Order Upwind) was used, which minimizes numerical errors and ensures the accuracy of calculating turbulent effects; for the gradient, the Least Squares Cell Based method was used, and for the main flow variables (velocity, turbulence kinetic energy and dissipation rate), the Second Order Upwind method was used. Temporal integration was performed using the Global Time Step to stabilize the numerical solution. To control the solution, the relaxation coefficients were set: for the turbulent kinetic energy 0.75, for the specific dissipation rate 0.75, and for the turbulent viscosity 1.0, which ensures a balance between convergence and accuracy of the simulation. The residuals were monitored using the continuity equations, velocities in the x and y directions, energy, turbulent kinetic energy (k), and specific dissipation rate (ω), with an absolute convergence criterion of 0.001, which guarantees the achievement of a stable solution.

Results and discussion

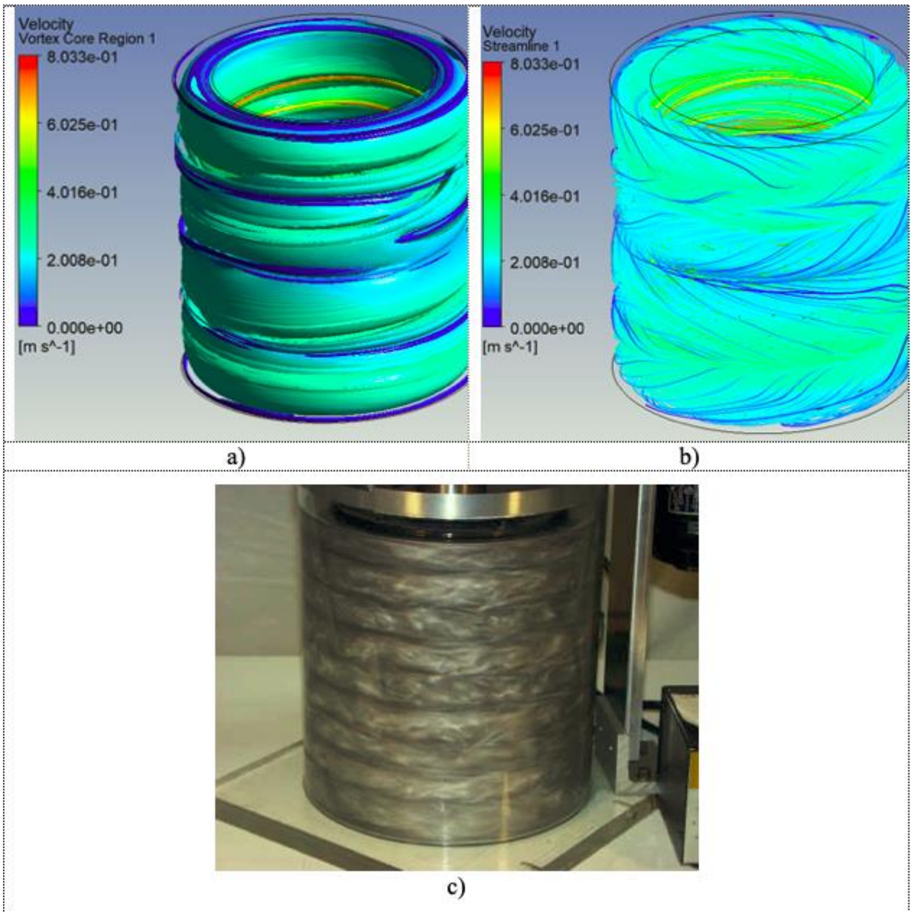


Figure 2. Isolines obtained as a result of numerical modeling.

Figure 2 shows the isolines obtained as a result of numerical modeling. Here: a) isolines of vortex structures; b) flow streamlines; c) visualization of vortices observed in the experiment.

The presented images show the results of the study of vortices arising in the intercylinder gap during rotation of the inner cylinder. Subfigure (a) shows the distribution of isolines of the intensity of vortex formations obtained as a result of numerical simulation using the GEKO turbulence model in the ANSYS Fluent software environment. These isolines clearly reflect the zones of formation of stable vortices characteristic of the Taylor flow at certain values of the Reynolds number. Subfigure (b) shows the streamlines demonstrating a complex pattern of circulation and secondary flows arising between the rotating inner and stationary outer cylinders. These lines allow visualizing the direction and structure of flows, including vortex rings and the zone of their interaction. Subfigure (c) illustrates an experimentally obtained image of real Taylor vortices recorded using optical visualization methods. Comparison of the simulation and experiment confirms the reliability of the numerical approach and demonstrates a high degree of coincidence of the vortex structure, which indicates the correctness of the selected turbulence model [24-25].

Figure 3 shows the isolines in the cross section, where a) is the velocity vector, b) is the axial velocity, c) is the radial velocity, d) is the kinetic energy, e) is the change in pressure.

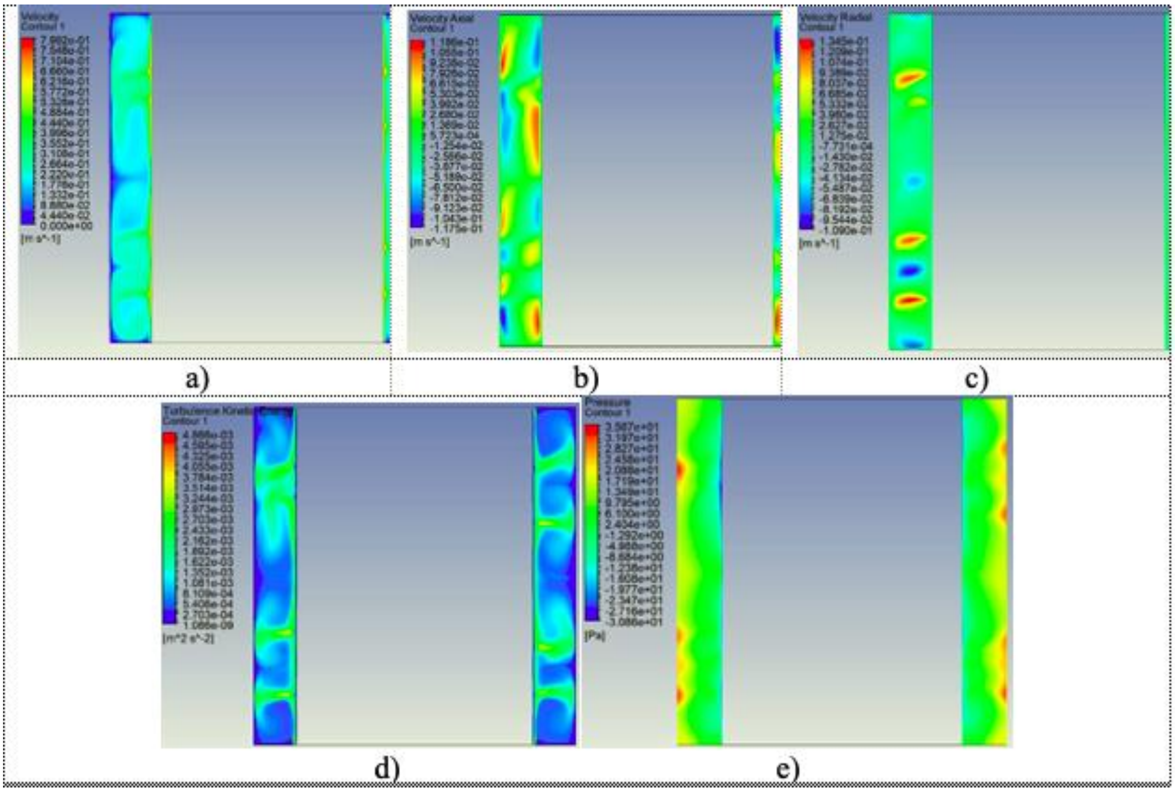
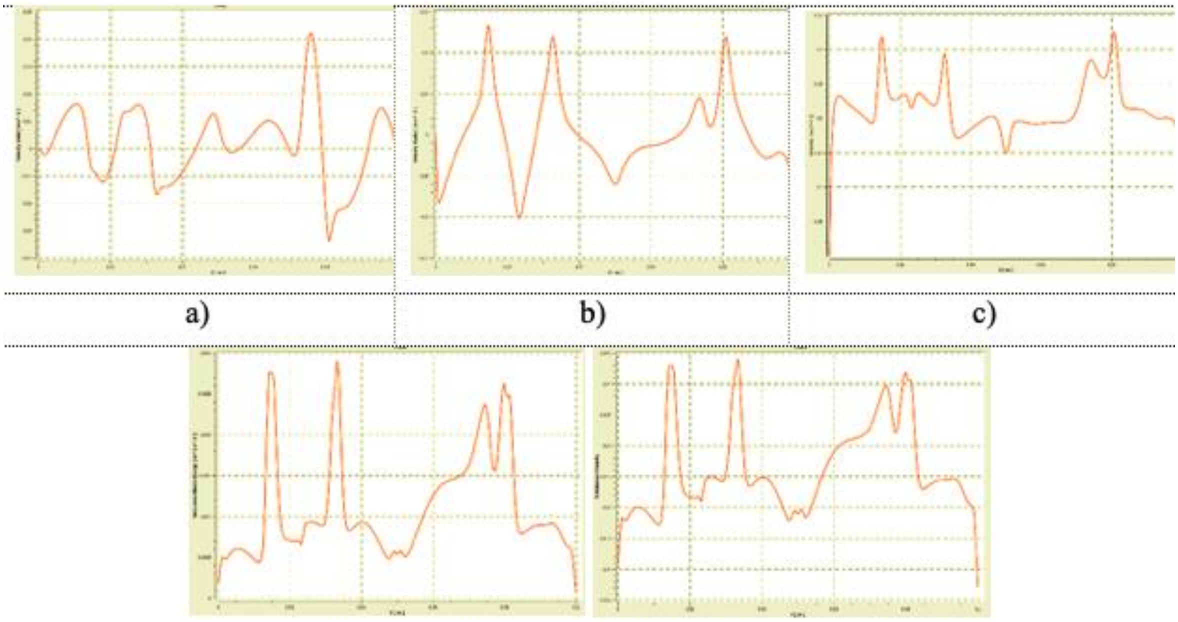


Figure 3. shows the isolines in cross section

In Figure 4, the changes along the centerline of the two cylinders are: a) velocity vector, b) axial velocity, c) radial velocity, d) kinetic energy, e) turbulence intensity.

Figure 4 shows the results of numerical modeling of the Taylor-Couette flow, reflecting the longitudinal distribution of the main hydrodynamic parameters along the centerline between two coaxial cylinders. It shows changes in the velocity vector, demonstrating the general direction and intensity of the flow; axial velocity, characterizing the movement of fluid along the longitudinal axis of the system; radial velocity, responsible for the movement between the inner and outer cylinders; kinetic energy distribution, allowing us to estimate the dynamics and intensity of local oscillations; as well as the turbulence intensity, reflecting the degree of instability and vortex formation in the flow. Analysis of these parameters allows us to identify key features of the flow structure, determine the zones with the greatest activity and confirm the adequacy of the GEKO turbulent model used in reproducing complex vortex structures in a closed inter-cylinder space.



a) b) c)

d) e)

Figure 4. The changes along the center line of two cylinders are as follows: a) velocity vector, b) axial velocity, c) radial velocity, d) kinetic energy, e) turbulence intensity

Conclusions

In this study, the Taylor–Couette flow was numerically investigated using the GEKO turbulence model implemented in the ANSYS Fluent software package. The computational results provided a clear visualization of the characteristic vortex structures that arise in this class of axisymmetric rotational flows. The simulations demonstrated that the GEKO model is capable of accurately capturing both the global behavior of the flow and the finer details of its turbulent dynamics, including the distributions of axial and radial velocity components, the spatial variation of kinetic energy, and the intensity of turbulence throughout the annular gap. The ability of the model to reproduce these features confirms its suitability for analyzing flows in which centrifugal instabilities and coherent vortex cells play a dominant role. The findings obtained from this work may serve as a valuable reference for future studies of similar rotating flows in a broad range of practical applications. These include the design of energy-efficient irrigation and water-distribution systems in agroecology, as well as the optimization of mixing, separation, and filtration processes in engineering and environmental technologies. Overall, the results underscore the potential of controlled vortex flows as an effective tool for sustainable water-management strategies and highlight the importance of modern turbulence models—such as GEKO—for simulating complex hydrodynamic phenomena in agroecological and engineering systems.

Reference

[1] Ostilla-Mónico, R., Huisman, S. G., Jannink, T. J., Van Gils, D. P., Verzicco, R., Grossmann, S., ... & Lohse, D. (2014). Optimal Taylor–Couette flow: radius ratio dependence. Journal of fluid mechanics, 747, 1-29.

[2] Taylor, G. I. 1936 Fluid friction between rotating cylinders. Proc. R. Soc. Lond. A 157, 546–564.

[3] Couette, M. 1890 Études sur le Frottement des Liquides. Gauthier-Villars et Fils.

[4] Brauckmann, H. & Eckhardt, B. 2013a Direct numerical simulations of local and global torque in Taylor–Couette flow up to Re D 30 000. J. Fluid Mech. 718, 398–427.

[5] Brauckmann, H. & Eckhardt, B. 2013b Intermittent boundary layers and torque maxima in Taylor–Couette flow. Phys. Rev. E 87, 033004.

[6] Dominguez-Lerma, M. A., Cannell, D. S. & Ahlers, G. 1986 Eckhaus boundary and wavenumber selection in rotating Couette–Taylor flow. Phys. Rev. A 34, 4956–4970.

[7] Eckhardt, B., Grossmann, S. & Lohse, D. 2007 Torque scaling in turbulent Taylor–Couette flow between independently rotating cylinders. J. Fluid Mech. 581, 221–250

[8] Gebhardt, TH. & Grossmann, S. 1993 The Taylor–Couette eigenvalue problem with independently rotating cylinders. Z. Phys. B 90 (4), 475–490.

[9] Van Gils, D. P. M., Huisman, S. G., Bruggert, G. W., Sun, C. & Lohse, D. 2011b Torque scaling in turbulent Taylor–Couette flow with co- and counter-rotating cylinders. Phys. Rev. Lett. 106, 024502.

[10] Huisman, S. G., Scharnowski, S., Cierpka, C., Kahler, C. J., Lohse, D., & Sun, C. (2013). Logarithmic boundary layers in highly turbulent Taylor-Couette flow. arXiv preprint arXiv:1302.6081.

[11] Lathrop, D. P., Fineberg, J., & Swinney, H. L. (1992). Transition to shear-driven turbulence in Couette-Taylor flow. Physical Review A, 46(10), 6390.

[12] Lewis, G. S., & Swinney, H. L. (1999). Velocity structure functions, scaling, and transitions in high-Reynolds-number Couette-Taylor flow. Physical Review E, 59(5), 5457.

[13] Smith, G. P., & Townsend, A. A. (1982). Turbulent Couette flow between concentric cylinders at large Taylor numbers. Journal of Fluid Mechanics, 123, 187-217.

[14] Pirro, D., & Quadrio, M. (2008). Direct numerical simulation of turbulent Taylor–Couette flow. European Journal of Mechanics-B/Fluids, 27(5), 552-566.

[15] Madaliev, M., Yunusaliev, E., Abdulkhaev, Z., Otakulov, B., Yusupov, S., Ergashev, I., & Tohirov, I. (2024). Comparison of numerical results of linear and nonlinear turbulence models based on the rans approach. In E3S Web of Conferences (Vol. 587, p. 01003). EDP Sciences.

[16] Madaliev, M., Abdujalilova, S., Tojiyev, R., Sadulayev, X., Sulaymanov, A., Otakulov, B., & Tohirov, I. (2024). Numerical study of the transition zone between laminar and turbulent flows for the flow plate problem. In BIO Web of Conferences (Vol. 145, p. 03035). EDP Sciences.

[17] Madaminov, J., Ergashev, I., Rustamova, M., Tohirov, I., Kodirov, M., Valixonov, D., & Mirzayev, M. (2024). Efficiency of using a solar air heating collector in a continental climate. In E3S Web of Conferences (Vol. 592, p. 01014). EDP Sciences.

[18] Li, Z., Liu, Y., Zhou, W., Wen, X., & Liu, Y. (2022). Thermal pollution level reduction by sweeping jet-based enhanced heat dissipation: A numerical study with calibrated Generalized k-ω (GEKO) model. Applied Thermal Engineering, 204, 117990.

[19] Durango-Cogollo, M., Garcia-Bravo, J., Newell, B., & Gonzalez-Mancera, A. (2020). CFD modeling of hydrocyclones—A study of efficiency of hydrodynamic reservoirs. Fluids, 5(3), 118.

[20] Menter, F. R., Kuntz, M., & Langtry, R. (2003). Ten years of industrial experience with the SST turbulence model. Turbulence, heat and mass transfer, 4(1), 625-632.

[21] Wijesooriya, K., Mohotti, D., Lee, C. K., & Mendis, P. (2023). A technical review of computational fluid dynamics (CFD) applications on wind design of tall buildings and structures: Past, present and future. Journal of Building Engineering, 74, 106828.

[22] Juraboyev, A., Juraev, U., Akhmedov, J., Kurganov, U., Matkarimov, N., & Madirimov, K. (2024). Natural-climatic and geographical features of the territory in the organization of recreation areas. In E3S Web of Conferences (Vol. 587, p. 05015). EDP Sciences.

[23] Kuldashov, N., Boltaev, Z., Xudoyberdiyev, M., Juraev, U., & Ruziyev, T. (2023). Natural waves in a spatial viscoelastic cylinder with a radial crack. In E3S Web of Conferences (Vol. 417, p. 06014). EDP Sciences.

[24] Makhmudova, D., Rustamova, M., Xujakulov, R., & Begmatova, N. (2025, June). Mathematical modeling of reducing water consumption in drinking water treatment facilities. In AIP Conference Proceedings (Vol. 3286, No. 1, p. 040035). AIP Publishing LLC.

[25] Makhmudov, I. E., Aliev, M. K., Makhmudova, D. E., Musayev, S. M., Rustamova, M. M., & Nematov, D. B. (2023). Static Mixer Apparatus for blending Ozone with water in the Process Pipeline. In E3S Web of Conferences (Vol. 449, p. 06014). EDP Sciences.