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

COMPARISON OF SMAGORINSKY, DYNAMICKEQN AND WALE LES MODELS FOR TURBULENT FLOW IN A RAPIDLY EXPANDING CHANNEL USING OPENFOAM

Namunabonu Nosirova[1], Murodil Madaliev[2,*], Shaxnoza Abdukhalilova[2], Xusanboy Muhammadyokubov[2]

[1] Institute of Mechanics and Seismic Stability of Structures named after. M.T. Urazbaeva
[2] Fergana State Technical University Fergana, Uzbekistan
*Correspondence: Madaliev.me2019@mail.ru

Abstract

This paper considers the problem of modeling turbulent flow in a sharply expanding channel using the large-scale simulation (LES) approach. The main focus is on comparing three different turbulence models: the Smagorinsky model, the dynamic kinetic energy model (DynamicKEqn) and the WALE model. The calculations are performed using the OpenFOAM software package, which provides capabilities for high-precision numerical modeling. The simulation results are presented, including flow characteristics such as velocity profiles, stresses and vortex structures, as well as their comparison between the models. The models are also compared in terms of changes in longitudinal, transverse velocities and pressure coefficients at the bottom of the channel in different sections. Based on the analysis, the advantages and limitations of each model in the context of the problem under consideration are revealed.

Keywords: Navier-Stokes equations, LES approach, OpenFOAM.

1. Introduction

Modeling turbulent flows in various geometric configurations is one of the urgent problems of modern fluid mechanics and numerical modeling. Such problems are widely used in engineering and scientific research, including aerodynamics, hydrodynamics and various heat and mass transfer processes. Of particular interest are flows in sharply expanding channels, which are often encountered in technical devices such as diffusers, jet engines, heat exchangers and hydraulic structures. A sharp change in channel geometry leads to the emergence of complex phenomena, such as recirculation zones, high velocity and pressure gradients, as well as intensive development of turbulence. To accurately predict such characteristics, it is necessary to use approaches that are able to take into account the complex physics of the process. One of the most promising methods for modeling such problems is the large-scale simulation (LES) method. This approach allows resolving large vortex structures that have a key effect on the flow dynamics, while simulating small-scale turbulent phenomena. Considerable attention in the study of turbulent flows in channels with sharply expanding cross-sections is given to theoretical and experimental analysis. In 1910, Blasius [1] analytically investigated the laminar separation of stationary two-dimensional flows of incompressible fluid in straight channels. This problem was further considered by many scientists to study the mechanisms of separation flows during observation and experiments of differential schemes for solving the Navier-Stokes equations. The existing flows have been theoretically and experimentally investigated for both laminar [2-4] and turbulent [5-7] modes of motion of both incompressible and compressible fluids. Most of the works in this direction study flows in channels with bilateral symmetric sharp expansion [8-14]. Experimental data for this purpose in a flat channel were obtained in works [8, 9], in which the formation of a circulation zone behind the bar is observed. A number of workers have used the equations of motion to calculate currents with an abrupt expansion approaching the edge layer [14, 15]. Important work on abrupt expansion channels was done by Armali et al. [4], who performed a thorough analysis of the flow behavior inside a backward-step channel experimentally and numerically [16]. Lee and Smith

Received: 17 April 2025
Revised: 06 May 2025
Accepted: 21 June 2025
https://doi.org/10.24412/2181-4473-2025.3.102

[17] used potential flow theory to obtain their results, but this theory failed to predict the flow distribution or reattachment behind the step. Initial numerical predictions for backward-step flows were made by Roach [18], Taylor and Ndefoe [19], and Durst and Pereira [20]. The split region at the bottom corner of the step was predicted by Alleborn et al. [21] after careful study of the channel expansion. Brandt et al. [22] and Hack Bush [23] used a multigrid method, and Lange et al. [24] used a local block refinement method to more accurately predict the results. Also worth mentioning is the detailed experimental work of Armali et al. [16], who studied the step geometry with the ratio H/h = 1.94. Kim and Moin [25] used the second-order space-time method for calculations.

This paper focuses on the comparison of three different LES-based turbulence models [26–29]: the Smagorinsky model [26], the Dynamic KEqn model [30–32], and the WALE model [33–35]. Each of these models has its own characteristics and areas of application, which makes their analysis important for understanding the advantages and limitations in the context of modeling flows in sharply expanding channels.

The aim of the study is to evaluate the accuracy and efficiency of these models based on the comparison of velocity profiles, stresses, vortex structures, as well as changes in longitudinal and transverse velocities and pressure coefficients in different channel sections. The calculations were performed using the OpenFOAM software package, which provides powerful tools for numerical modeling of turbulent flows.

This work is aimed at identifying the optimal approach to modeling the flow in question, which can be used in the development and optimization of engineering systems and processes associated with turbulent flows.

2. Physical and mathematical formulation of the problem

We consider a turbulent flow in a flat two-dimensional channel with a sharp expansion. To analyze the behavior of the fluid in the channel with a reverse step, it is assumed that the step length is H = 25.4 mm. The mathematical and physical flow model, as well as the geometry of the computational domain, are shown in Fig. 1. The problem is solved using a Cartesian coordinate system, the origin of which is located in the lower left corner in front of the steps. The channel width at the inlet is H, while at the outlet, in the right section, the channel width is twice as large - 2H. The step height is equal to H. At the channel inlet, the longitudinal velocity is specified by the value U = U0 = 10 m/s, while the transverse velocity V and pressure p are zero. To simulate the fluid flow, the unsteady LES equations for an incompressible fluid are used without simplifications. When switching to dimensionless quantities, the channel width in its greater part is taken as the length scale — H, and the average velocity at the channel entrance — U0 is used for the velocity scale, and no-slip conditions are used in all walls. The Reynolds number for this flow is Re = 25400.

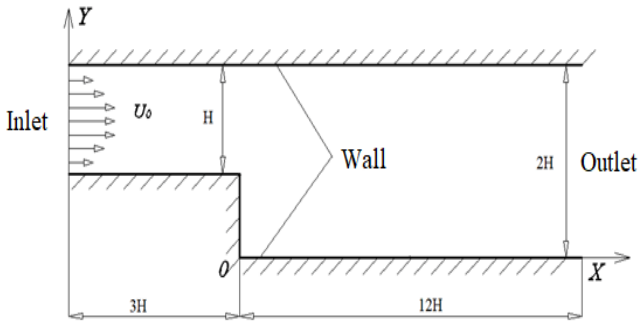


Fig. 1. Physical scheme of the flow of the computational domain

The following grid was chosen for the calculations: a grid of 70x30 was used for the inlet part of the channel, and 180x60 for the extended part of the channel. The grid was compressed near the wall, as shown in Figure 2. This was done in order to more accurately account for the laminar layer.

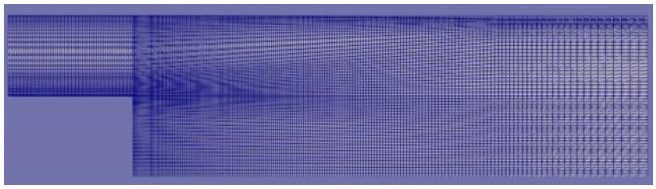


Fig. 2. Computational grid for the problem

2.1 Mathematical model

To simulate turbulent flow in a rapidly expanding channel using the OpenFOAM software environment, it is necessary to solve the Navier-Stokes equations for non-liquefiable liquids, taking into account turbulence. In this problem, the main equations that describe the flow dynamics are:

The mass conservation equation (the continuity equation), which describes the law of mass conservation within the computational domain:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

The equation of conservation of momentum, which describes the change in the velocity of a fluid under the influence of external and internal forces:

$$\frac{\partial u_i}{\partial t} + \overline{u}_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_i}. \quad (2)$$

here u_i - velocity vector describing the motion of a fluid, P - pressure, ν - kinematic viscosity, ϱ - density.

3. Turbulence approaches

LES approach

The LES (Large Eddy Simulation) model is a numerical simulation method for turbulent flows that is based on the idea of separating the turbulent field into large-scale and small-scale eddies. In LES, the dynamics of large eddies are modeled, while small eddies are filtered and their influence is taken into account through the effective turbulent viscosity [26–29].

Filtering equation: In LES, the calculations are based on filtering the Navier-Stokes equations, which allows separating large-scale components from small-scale ones:

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(-\tau_{ij}^{SGS} + 2\nu \overline{S}_{ij}\right) + F_i \quad (3)$$

here $\overline{u}_i$ - filtered velocity components, $\overline{p}$ - filtered pressure, F_i - external force, τ_{ij}^{SGS} subgrid-scale stress tensor, $\overline{S}_{ij}$ - filtered strain rate tensor.

Where

$$\tau_{ij}^{SGS} - \frac{1}{3}\tau_{kk}^{SGS}\delta_{ij} = 2\nu_t \overline{S}_{ij} \quad (4)$$

Turbulent viscosity model with Smogorinsky models: LES uses effective turbulent viscosity to account for the influence of small-scale eddies. This viscosity is typically modeled using Smogorinsky-type models [26]:

$$\nu_t = (\Delta)^2 \cdot C_s \cdot \left|\overline{S}\right| \quad (5)$$

here: ν_t — turbulent viscosity, Δ — grid cell size, Cs — Smogorinsky constant, $\left|\overline{S}\right|$ — the magnitude of the strain tensor. The LES model is a powerful tool for numerical modeling of turbulent flows, which allows one to effectively solve complex problems that require high accuracy in describing large eddies and flow dynamics.

Model DynamicKEqn (Dynamic Kinetic Energy Equation) is an advanced model for large-scale turbulent flow simulation (LES). It is designed to more accurately calculate the kinetic energy of turbulence through a dynamic approach to determining its parameters. Compared to classical models such as the Smogorinsky model, DynamicKEqn provides a more flexible and adaptive way to model turbulent structures. DynamicKEqn uses a dynamic procedure to determine the turbulent viscosity coefficient. Unlike the Smogorinsky model with a constant coefficient, in DynamicKEqn this coefficient is adapted depending on the local flow characteristics. This allows taking into account the variability of turbulence in space and time. The model is based on the equation for the transfer of turbulent kinetic energy (k), which is written as follows [30]:

$$\frac{\partial k}{\partial t} + \bar{u}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\nu_t \frac{\partial k}{\partial x_j}\right] + P_k - \varepsilon \quad (6)$$

where: k is the kinetic energy of turbulence, u is the velocity vector, Pk is the production of turbulent energy, ε is the energy dissipation, νt is the turbulent viscosity.

The WALE (Wall-Adapting Local Eddy Viscosity) model is a large-scale turbulence simulation (LES) model designed to more accurately simulate flows, especially near walls. It was developed as an improvement on the Smogorinskii model to better account for the behavior of small-scale eddies in regions with a high velocity gradient and to minimize errors near solid surfaces. The model is based on the use of the velocity deformation tensor and its squares to determine the turbulent viscosity (νt). The turbulent viscosity in the WALE model is calculated using the following formula [33]:

$$\nu_t = (C_\omega \Delta)^2 \frac{\left(s_{ij}^d s_{ij}^d\right)^{3/2}}{\left(s_{ij}^d s_{ij}^d\right)^{3/2} + \left(s_{ij} s_{ij}\right)^{3/2}} \quad (7)$$

where: S_{ij} is the strain rate tensor, $S_{ij}{}^d$ is the deviatoric part of the strain rate tensor, C_w is the model coefficient (usually C_w=0.325), Δ is the filter size (usually corresponds to the grid step).

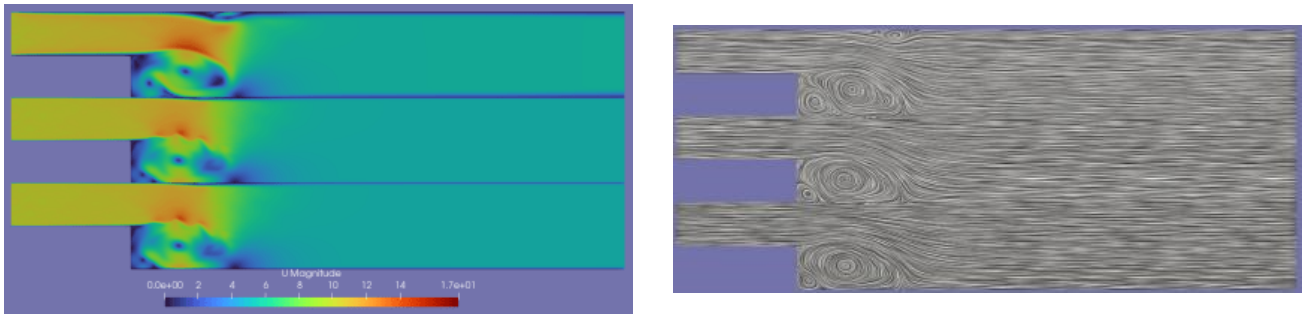
4. Method of solution

The PIMPLE algorithm, which combines the SIMPLE and PISO methods for pressure and velocity correction, was used for numerical simulation of turbulent flows. The GAMG solver was used for pressure (accuracy $1e^{-6}$, relaxation 0.1), and smoothSolver with the GaussSeidel method was used for velocity (U), turbulence kinetic energy (k), and

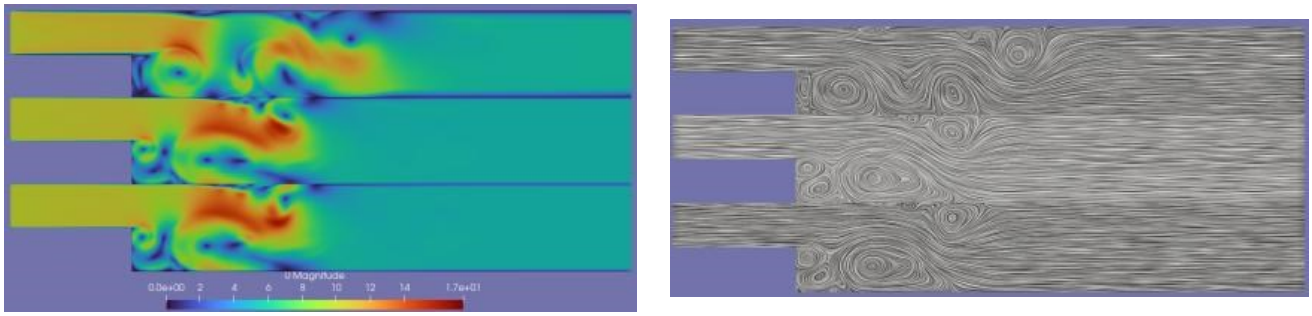
viscosity (v_t) with an accuracy of $1e^{-5}$ and a relaxation of 0. To improve the accuracy, a second-order scheme (Upwind) was used, and the relaxation coefficients were 0.9 for velocity and 0.8 for other variables. The residual control was set to $1e^{-4}$ for pressure and $1e^{-5}$ for other variables, ensuring the stability and accuracy of calculations.

4. Results and discussion

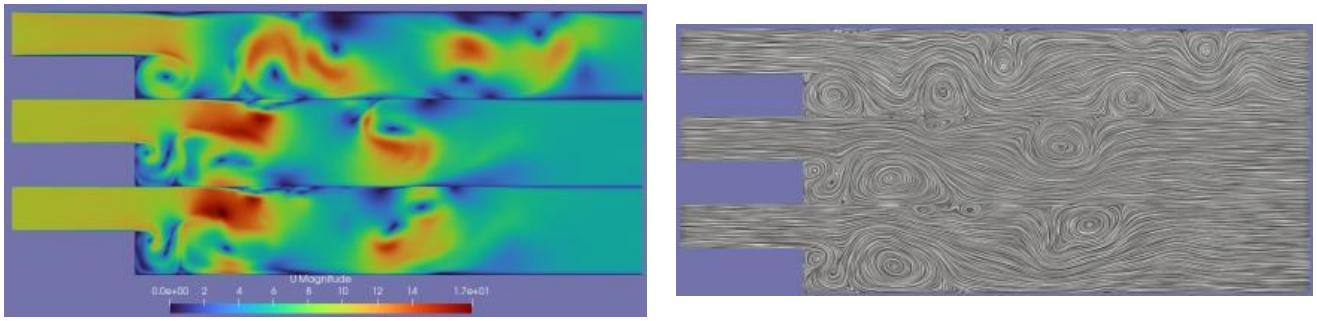
Figure 3 shows velocity contours and streamlines that allow the flow characteristics to be analyzed for different turbulence models



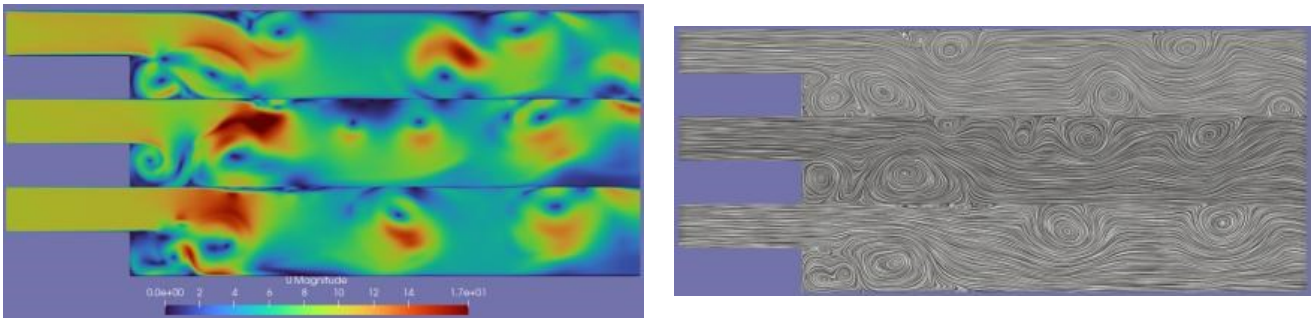
T=0.02



T=0.04



T=0.06



T=0.08

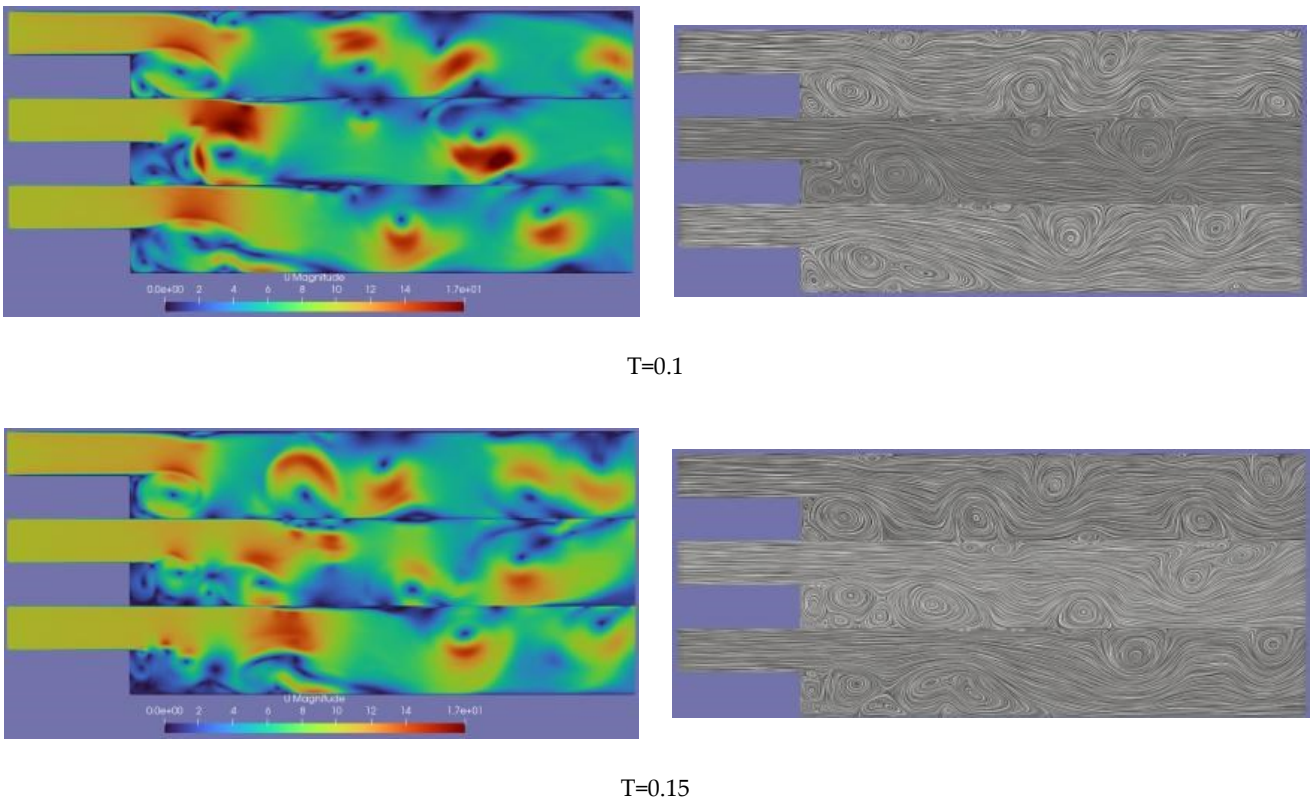


Fig. 3. Changes in velocity isolines and current lines at different time intervals

The upper part of the figure illustrates the results obtained using the Smogorinsky model, which is used to describe turbulent processes due to its simplicity and efficiency. The middle part of the figure shows the results calculated using the DynamicKEqn model, known for its adaptability and ability to take into account local changes in the flow. The lower part of the figure displays the results obtained using the WALE model, which provides high accuracy near the walls and takes into account small-scale vortex structures. Figure 4 shows the vortex structures that arise during turbulent flow in detail and clearly.

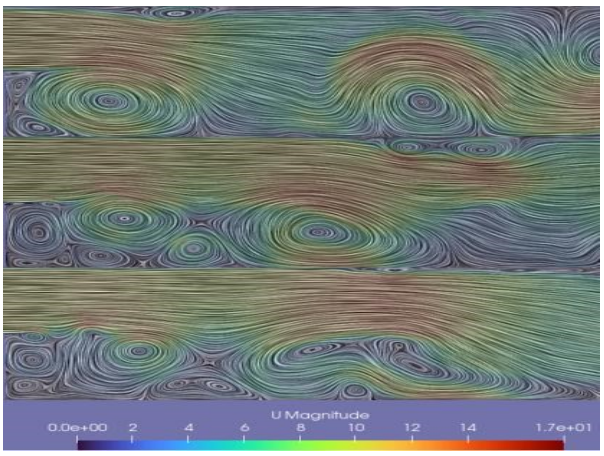


Fig. 4. Vortex structures arising in the process of turbulent flow

The image clearly shows vortices of different scales, from large formations that form the main structure of the flow to smaller vortices that detail the complexity and dynamics of the process. This visual representation allows not only to assess the distribution of vortices, but also to identify the characteristic features of turbulent motion, including their interaction and behavior in different areas of the flow. Analysis of the presented figure demonstrates the importance of taking into account vortex structures in modeling and provides a deeper understanding of the physics of the phenomenon. Fig. 5 shows the velocity isolines and streamlines averaged over time.

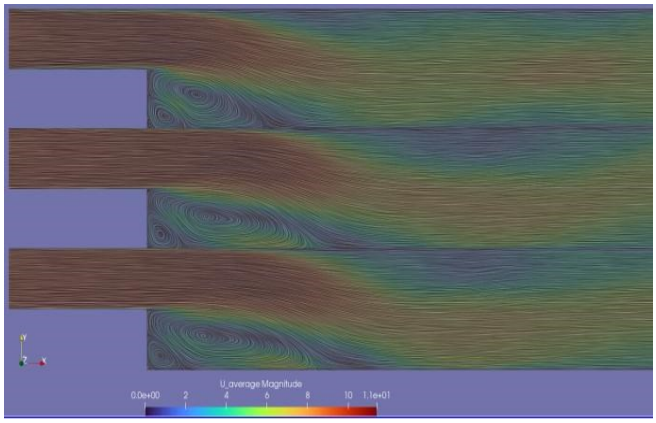
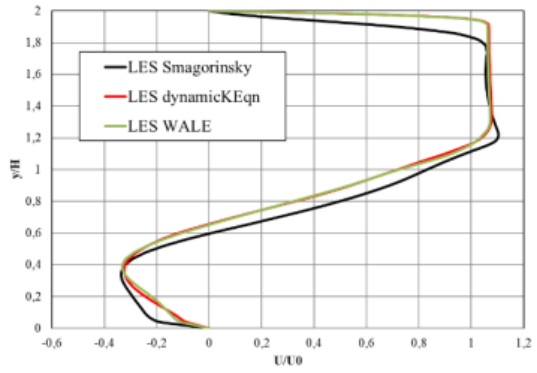
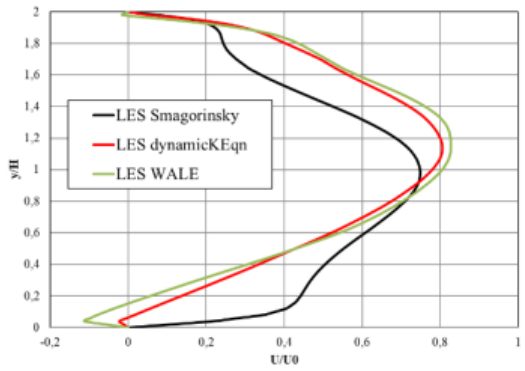


Fig. 5. Velocity contour lines and current lines, averaged over time

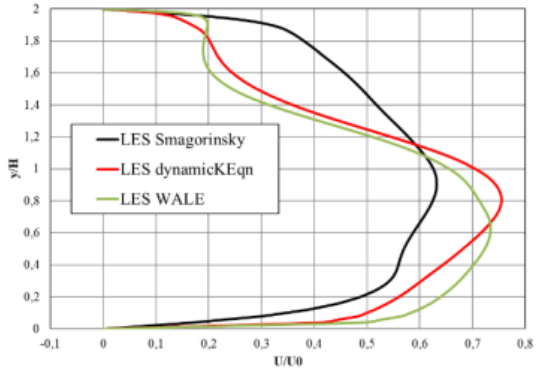
Fig. 5 shows the change in the dimensionless velocity U/U_0 on sections at x/H=1, 4 and 6, where $U0$ is the velocity at the channel entrance.



x/H=1



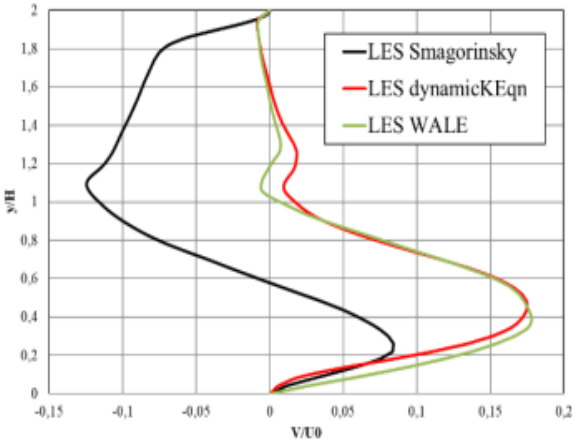
x/H=4



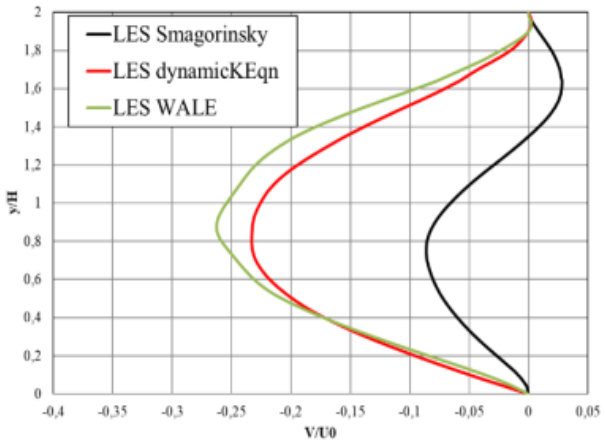
x/H=6

Fig. 6. Change in dimensionless velocity $U/U0$ on sections at x/H=1, 4 and 6

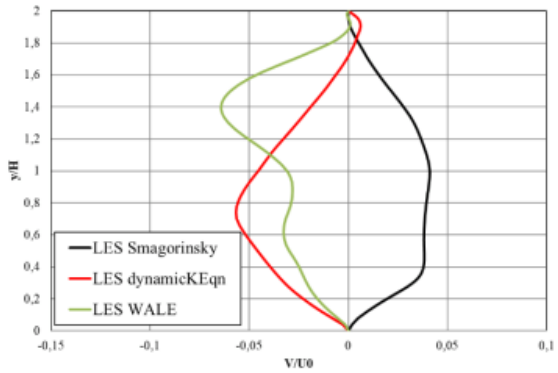
As can be seen from Figure 6, the results of the DynamicKEqn and WALE models are significantly closer to each other, demonstrating similar predictions of the velocity distribution. At the same time, the results of the Smogorinsky model differ significantly from the results of these models, which indicates its limited accuracy in comparison with more modern and adaptive approaches such as DynamicKEqn and WALE. Figure 7 presents the distribution of the transverse velocity $V/U0$ on the sections with x/H=1, 4 and 6.



x/H=1



x/H=4



x/H=6

Fig. 7. Change in dimensionless velocity V/$U0$ on sections at x/H=1, 4 and 6

A similar picture as in Fig. 6 can be seen in Fig. 7.

Fig. 8 shows changes in the pressure and resistance coefficients on the lower wall of the channel. The distribution of the surface pressure coefficient illustrates the change in pressure along the surface of the channel wall depending on the distance from a given point.

$$C_p = \frac{p-p_\infty}{0.5\rho U_0^2} \quad (8)$$

where p is the pressure at a point on the airfoil surface, P∞ is the free-stream pressure, ϱ is the free-stream density, and U0 is the free-stream velocity. The pressure coefficient Cp shows how the flow interacts with the object. If Cp>0, it means that the pressure is higher than in the free stream, and if Cp<0, the pressure is lower than in the free stream. Typically, in aerodynamic objects, areas with high Cp correspond to high-pressure regions such as flow separation zones or shock waves.

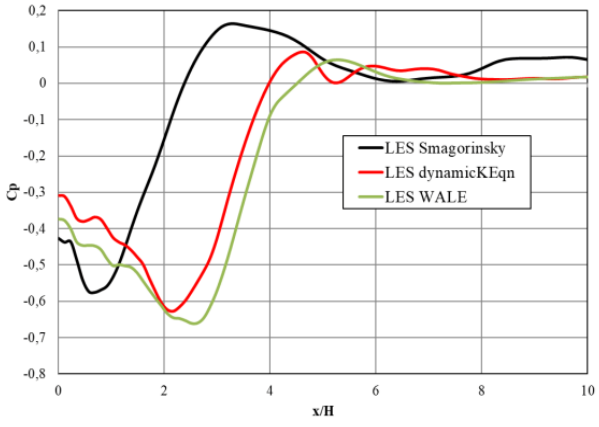


Fig. 8. Change in pressure coefficients on the lower wall of the channel

As can be seen from Figure 8, the results of the DynamicKEqn and WALE models demonstrate a high degree of similarity, confirming their accuracy and consistency in predicting the pressure distribution on the bottom wall of the channel. At the same time, the results of the Smogorinsky model differ significantly from these models, indicating its rougher predictions compared to more advanced approaches such as DynamicKEqn and WALE.

6. Conclusions

In this paper, the problem of modeling turbulent flow in a sharply expanding channel is considered using the large-scale simulation (LES) approach. Three different turbulence models are compared: the Smogorinsky model, the dynamic kinetic energy model (DynamicKEqn) and the WALE model. The calculations are performed using the OpenFOAM software package, which provides high accuracy of numerical modeling. The results are presented, including flow characteristics such as velocity profiles and vortex structures, with their comparison between the models. A comparison of the models is also made in terms of changes in longitudinal and transverse velocities, as well as pressure coefficients in different channel sections. Based on the analysis, the advantages and limitations of each model in the context of the problem under consideration are revealed, which helps to evaluate their effectiveness in solving this problem.

References

1. Blasius H. Laminare Stromung in Kanalen Wechselnder Breite. Zeitschrift fur Math. und Phys., 1910, vol. 58, no. 10, pp. 225–233.
2. Honji H. The starting flow down a step. J. Fluid Mech., 1975, vol. 69, iss. 2, pp. 229–240. DOI: https://doi.org/10.1017/S0022112075001413
3. Синха С.П., Гупта А.К., Оберай М.М. Ламинарное отрывное обтекание уступов и каверн. Ч. 1. Течение за уступом. Ракетная техника и космонавтика, 1981, т. 19, № 12, с. 33–37.

4. Armaly B.F., Durst F., Pereira J.C.F., et al. Experimental and theoretical investigation of backward-facing step flow. J. Fluid Mech., 1983, vol. 127, pp. 473–496. DOI: https://doi.org/10.1017/S0022112083002839
5. Чжен П. Отрывные течения, М., Мир, 1972.
6. Гогиш Л.В., Степанов Г.Ю. Турбулентные отрывные течения. М., Наука, 1979.
7. Le H., Moin P., Kim J. Direct numerical simulation of turbulent flow over a backward-facing step. J. Fluid Mech., 1997, vol. 330, pp. 349–374. DOI: https://doi.org/10.1017/S0022112096003941
8. Durst F., Melling A., Whitelow J.H. Low Reynolds number flow over a plane symmetric sudden expansion. J. Fluid Mech., 1974, vol. 64, iss. 1, pp. 111–118. DOI: https://doi.org/10.1017/S0022112074002035
9. Cherdron W., Durst F., Whitelow J.H. Asymmetric flows and instabilities in symmetric ducts with sudden expansions. J. Fluid Mech., 1978, vol. 84, iss. 1, pp. 13–31. DOI: https://doi.org/10.1017/S0022112078000026
10. Macadno E.O., Hung T.-K. Computational and experimental study of a captive annular eddy. J. Fluid Mech., 1967, vol. 28, iss. 1, pp. 43–64. DOI: https://doi.org/10.1017/S0022112067001892
11. Kumar A., Yajnik K.S. Internal separated flows at large Reynolds number. J. Fluid Mech., 1980, vol. 97, iss. 1, pp. 27–51. DOI: https://doi.org/10.1017/S0022112080002418
12. Плоткин А. Расчеты спектральным методом некоторых отрывных ламинарных течений в каналах. Аэрокосмическая техника, 1983, № 7, с. 75–85.
13. Acrivos A., Schrader M.L. Steady flow in a sudden expansion at high Reynolds numbers. Phys. Fluids, 1982, vol. 25, iss. 6, pp. 923–930. DOI: https://doi.org/10.1063/1.863844
14. Куон О., Плетчер Р., Льюис Дж. Расчет течений с внезапным расширением при помощи уравнений пограничного слоя. Теор. основы инж. расч., 1984, т. 106, № 3, с. 116–123.
15. Льюис, Плетчер. Пределы применимости уравнений пограничного слоя для расчета ламинарных течений с симметричным внезапным расширением. Теор. основы инж. расч.- 1986.- № 2.-с. 284-294.
16. Маликов З.М., Мадалиев М.Э. Численное моделирование течения в плоском внезапно расширяющемся канале на основе новой двужидкостной модели турбулентности. Вестник МГТУ им. Н.Э. Баумана. Сер. Естественные науки, 2021, № 4 (97), с. 24–39. DOI: https://doi.org/10.18698/1812-3368-2021-4-24-39
17. Lee, Y. S. and Smith, L. C., 1986, Analysis of power-law viscous materials using complex stream, potential and stress functions, in Encyclopedia of Fluid Mechanics, vol. 1, Flow Phenomena and Measurement, ed. N. P. Cheremisinoff, pp. 1105–1154.
18. Roache, P. J., 1972, Computational Fluid Dynamics, Hermosa, New Mexico, pp. 139–173.
19. Taylor, T. D., and Ndefo, E., 1971, Computation of viscous flow in a channel by the method of splitting, Proc. of the Second Int. Conf. on Num. Methods in Fluid Dynamics, Lecture Notes in Physics, vol. 8, pp. 356–364, Springer Verlag, New York.
20. Durst, F., and Peireira, J. C. F., 1988, Time-dependent laminar backwardfacing step flow in a two-dimensional duct, ASME J. Fluids Eng., 110, pp. 289–296.
21. Alleborn, N., Nandakumar, K., Raszillier, H., and Durst, F., 1997, Further contributions on the two-dimensional flow in a sudden expansion, J. Fluid Mech., 330, pp. 169–188.
22. Brandt, A., Dendy, J. E., and Ruppel, H., 1980, The multigrid method for semi-implicit hydrodynamic codes, J. Comput. Phys., 34, pp. 348–370.
23. Hackbusch, W., 1985, Multigrid Methods for Applications, Springer, Berlin.
24. Lange, C. F., SchaËfer, M., and Durst, F., 2002, Local block refinement with a multigrid flow solver, Int. J. Numer. Methods Fluids 38, pp. 21–41.
25. Kim, J., and Moin, P., 1985, Application of a fractional-step method to incompressible Navier-Stokes equations, J. Comput. Phys., 59, pp. 308– 323.
26. Smagorinsky, J. (1963). General circulation experiments with the primitive equations: I. The basic experiment. Monthly Weather Review, 91(3), 99–164.
27. Lilly, D. K. (1967). The representation of small-scale turbulence in numerical simulation experiments. Proceedings of the IBM Scientific Computing Symposium on Environmental Sciences, 195–210.
28. Germano, M., Piomelli, U., Moin, P., & Cabot, W. H. (1991). A dynamic subgrid-scale eddy viscosity model. Physics of Fluids A: Fluid Dynamics, 3(7), 1760–1765.
29. Ferziger, J. H., & Perić, M. (2002). Computational Methods for Fluid Dynamics. Springer.
30. Germano, M., Piomelli, U., Moin, P., & cabrera, M. (1991). "A Dynamic Subgrid-Scale Eddy Viscosity Model." Journal of Fluid Mechanics, 238, 325–336.
31. Moin, P., & Mahesh, K. (1998). "Direct Numerical Simulation: A Tool in Turbulence Research." Annual Review of Fluid Mechanics, 30, 539–578.
32. Speziale, C. G., & Rogallo, R. S. (1991). "A Dynamic Subgrid-Scale Model for Large Eddy Simulation." Physics of Fluids, 3(7), 1622–1630.
33. Piomelli, U., & Balaras, E. (2002). "Turbulence Modeling in Large Eddy Simulation: An Overview." Progress in Aerospace Sciences, 38, 345-368.
34. Nicoud, F., & Ducros, F. (1999). "Subgrid-Scale Stress Modelling Based on the Square of the Velocity Gradient." Flow, Turbulence and Combustion, 62(3), 183–200.
35. Georges, J. F., & Moin, P. (1994). "A Study of Large Eddy Simulation of Turbulent Flow in a Square Channel." Journal of Fluid Mechanics, 278, 275–309.