pISSN:2349-5707 Volume:13,Issue 5, May -2026 eISSN:2349-5715

NUMERICAL STUDY OF PITZ-DAILY TURBULENT FLOW USING OPENFOAM

Nazarov F.X.,
Fardaev J.N. ,
Adilov K.
Institute of Mechanics and Seismic Stability of
Structures named after M.T. Urazbayev ANRUz, Tashkent, Uzbekistan
Tashkent State Technical University named after Islam Karimov
Tashkent State Transport University
E-mail: farruxnazar@mail. ru.,
jonibek19940302@gmail.com., kamoladilov2021@gmail.com

https://doi.org/10.5281/zenodo.20072095

Abstract. This paper presents a numerical simulation of turbulent flow with step separation based on an experiment conducted by Pitz and Daily in 1983. The problem is solved using OpenFOAM software. The k-ε (k-epsilon) and LES (Large Eddy Simulation) models are used to calculate turbulence. The simulation results are compared with experimental data, and differences in flow profiles are analyzed.

Keywords: Pitz-Daily, OpenFOAM, LES, k-epsilon, turbulent flow, CFD

Introduction. Step-separated flows are widely encountered in engineering applications: in combustion chambers, heat exchange systems, and aerodynamic channels. These flows are characterized by complex structures, including recirculation zones, flow reattachment, and developed vortex formations. In such geometries, turbulence, mixing, and pressure distribution significantly affect the efficiency of flow and heat transfer.

Modeling such flows is of particular interest for the purpose of optimizing them and increasing the efficiency of technological processes. Both experimental and numerical methods are used for this purpose. In 1983, Pitz and Daily experimentally studied turbulent flow with step separation, including under combustion conditions, providing valuable benchmark data for subsequent numerical studies.

This paper presents a simulation of this experiment using OpenFOAM simulations and modern turbulence models.

Step-down flows have been the subject of numerous scientific studies. Armaly et al. (1983) analyzed the transition from laminar to turbulent flow at different Reynolds numbers, highlighting key characteristics of recirculation zones. Eaton and Johnston (1980) examined in detail the influence of step angle and inlet conditions on flow structure. Kim and Moin (1985) used Direct Numerical Simulation (DNS) to accurately visualize vortices and turbulence development [1–3].

The Pitz and Daily (1983) experiment is a classic and is devoted to the study of a turbulent mixing layer under combustion conditions [4]. During the experiment, the values of velocity, pressure, reattachment zone length, and mixing layer expansion velocity were recorded. The data obtained by them are used as a reference in many numerical studies. Numerical modeling of turbulent flows is rapidly developing due to the growth of computing power. OpenFOAM is an open CFD platform offering extensive modeling capabilities. Versteeg and Malalasekera [5] described in detail the numerical methods implemented in OpenFOAM, including discretization schemes and solution algorithms.

The k-ε model (Launder and Spalding, 1974) has proven itself to be effective for calculating steady-state turbulent flows. Its advantages include ease of implementation and high stability of calculations. The LES model (Spalart, 1992) is more accurate and allows for a detailed reproduction of the turbulent flow structure, especially in the recirculation and mixing zones, but

pISSN:2349-5707 Volume:13,Issue 5, May -2026 eISSN:2349-5715

requires significantly more computational resources [6, 7]. The present study aims to compare the simulation results obtained using both models in order to assess their applicability to problems similar to the Pitz-Daily experiment.

Physical and mathematical formulation of the problem. A turbulent flow with a backward-facing rib is considered in a two-dimensional coordinate system. The physical picture of the analyzed flow and the configuration of the computational domain are shown in Fig. 1. The calculation is performed using the Reynolds-mean approximation of the Navier-Stokes equations (RANS) and the large eddy simulation (LES).

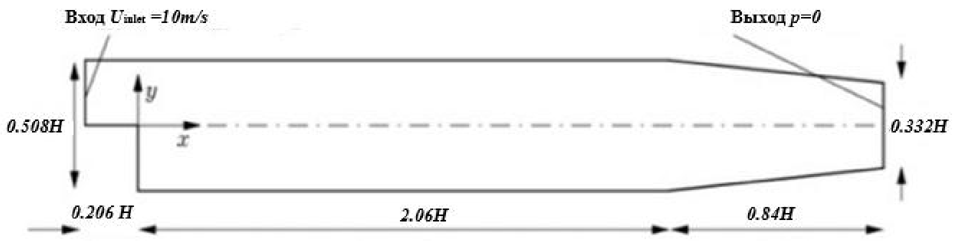


Fig. 1. Backward-facing step flow diagram

Here $H = 100\,mm$.

The Navier-Stokes equations are a system of differential equations that describe the motion of an incompressible fluid [8,9]:

$$\rho\frac{\partial \vec{U}}{\partial t}+\rho\left(\vec{U}\quad\right)\vec{U}=-\quad p+\quad \mu_{eff}\left(\quad \vec{U}+\left(\quad \vec{U}\right)^{T}\right), \qquad (1)$$

$$\vec{U}=0.$$

Where:

- $\vec{U}$ - fluid velocity vector,
- t - time,
- p - pressure,
- ρ - density,
- μ_{eff} - effective dynamic viscosity, which includes molecular and turbulent components: $\mu_{eff}=\mu+\mu_t$
- ∇ is the nabla operator that defines the gradient and divergence of a vector field.

The boundary conditions are given as follows:

Input (inlet): fixed speed value $U_{inlet}=10\ m/s$, "uniform" profile.

Outlet: zero pressure $p=0$, with zero gradient for velocity.

Upper and lower walls: no-slip conditions for velocity and zero pressure gradient.

The k-ε and LES models are used to model turbulence. The k-ε model is based on solving additional transport equations for turbulent kinetic energy and its dissipation rate. The LES model resolves large-scale eddies directly, while small scales are modeled using sublattice models (e.g., the Smogorinsky dynamic model).

Computational mesh. The mesh was generated using the blockMesh utility. The domain is divided into rectangular blocks, with localized cell density near the step and recirculation zone. The cell size was chosen to adequately resolve velocity gradients and turbulence. A finer mesh with a larger number of cells was used in the LES models. In computational fluid dynamics, it is crucial that the simulation adequately reflects real physical processes. One of the key advantages is flexibility: parameters can be quickly changed, and obtaining results requires less time and resources. For this study, the computational domain was discretized using rectangular cells (see Fig. 2).

pISSN:2349-5707 Volume:13,Issue 5, May -2026 eISSN:2349-5715

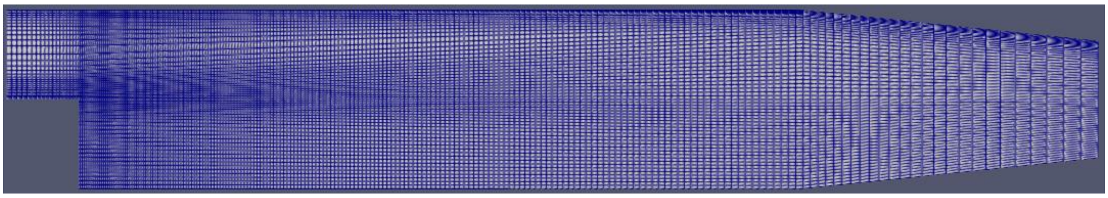


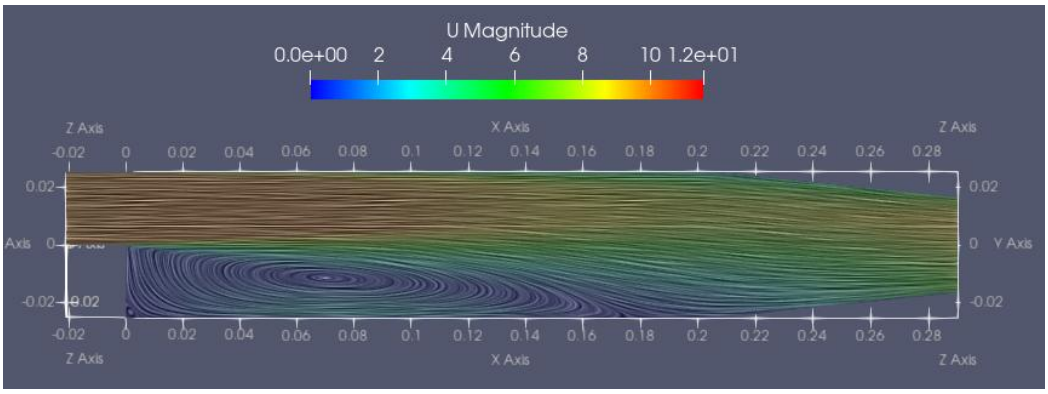
Fig. 2. Grinding of the mesh at the wall with a reverse step

Visualization and comparison of models. The figure below shows a comparison of the results obtained using the k-ε and LES models at a point-by-step basis:

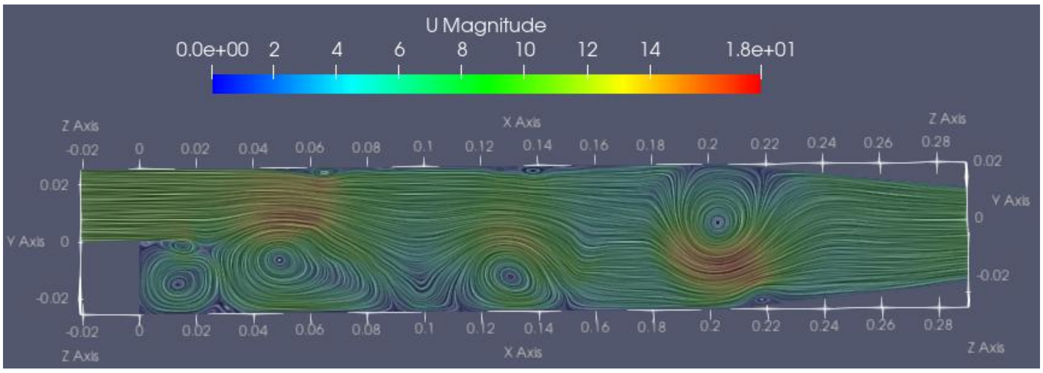
As can be seen from the graph, the LES model gives a more detailed velocity profile and captures a larger number of vortex structures, while the k-ε model smooths out these fluctuations.

Numerical modeling and postprocessing. The simulation was performed using the pisoFoam solver. The simulation time interval was 0.1 seconds. The writeInterval, deltaT, and endTime parameters were specified in the controlDict file. The discretization schemes and convergence parameters were defined in the fvSchemes and fvSolution files. Mesh quality was checked using the checkMesh command, and the results (velocity, pressure, and turbulence fields) were visualized using paraFoam. During the simulation, the probes tool was used to record values at specific points and the fieldAverage tool was used to calculate time-averaged values.

Results analysis. The LES model more accurately depicts the mixing zone and vortex structures. Velocity gradients and the reattachment zone were more clearly defined. The k-ε model provides a faster solution but smooths out flow details, especially in the vortex zone. Figures 3-4 compare the results obtained using the k-ε and LES turbulent models.



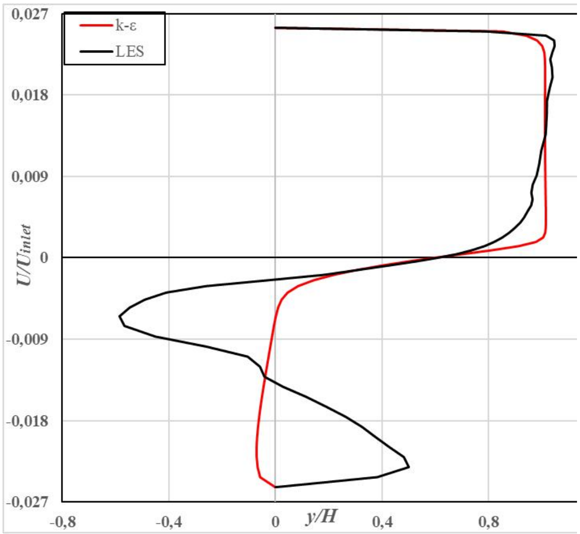
k- ε



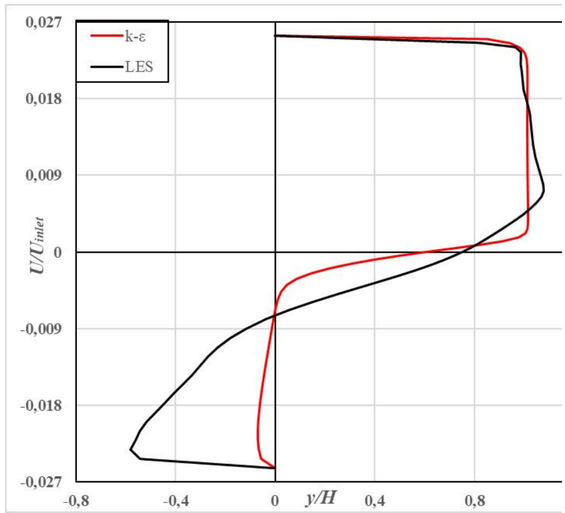
LES

Fig. 3. Streamlines.

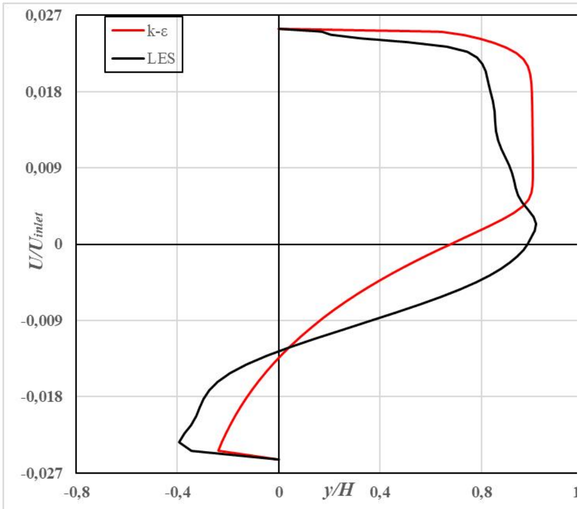
The velocity profiles in the sections x/H =0.02, x/ H =0.06 and x/ H =0.1 are shown in Fig. 4.



x/ H = 0.02



x/ H = 0.06



x/ H = 0.1

Fig. 6. Velocity profile in different sections.

Conclusion. Numerical flow simulation using Pitz-Daily geometry using OpenFOAM was successfully completed. A comparison of the k-ε and LES models allowed us to identify the strengths and weaknesses of each turbulence model. LES provides high physical fidelity, while the k-ε model yields fast and stable solutions. Future applications of the combustion model in conjunction with LES may provide a more realistic simulation of the processes.

LITERATURE

[1] Armaly B. F. et al. Experimental and theoretical investigation of backward-facing step flow //Journal of fluid Mechanics. – 1983. – T. 127. – C. 473-496.

[2] Eaton J. K. Turbulent flow reattachment: an experimental study of the flow and structure behind a backward-facing step. – Stanford University, 1980.

[3] Kim J., Moin P. Application of a fractional-step method to incompressible Navier-Stokes equations //Journal of computational physics. – 1985. – T. 59. – №. 2. – C. 308-323.

[4] Pitz R. W., Daily J. W. Experimental study of combustion in a turbulent free shear layer formed at a rearward facing step //19th Aerospace Sciences Meeting. – 1981. – C. 106.

[5] Versteeg H. K., Malalasekera W. Computational fluid dynamics //The finite volume method. – 1995. – C. 1-26.

[6] Spalart, P. R. A one-equation turbulence model for aerodynamic flows //AIAA 30th aerospace sciences meeting, Reno, 1992.

[7] Launder B. E., Spalding D. B. The numerical computation of turbulent flows //Numerical prediction of flow, heat transfer, turbulence and combustion. – Pergamon, 1983. – C. 96-116.

[8] Маликов З. М., Мадалиев М. Э. Численное моделирование течения в плоском внезапно расширяющемся канале на основе новой двужидкостной модели турбулентности //Вестник Московского государственного технического университета им. НЭ Баумана. Серия «Естественные науки». – 2021. – №. 4 (97). – С. 24-39.

[9] Назаров Ф.Х. Численное исследование потока с обратным уступом при помощи модели турбулентности // Проблемы механики. 2023. №.1. С. 126-129.

Nazarov F.Kh., Navruzov D.P. Pitz–Daily turbulent oqimini OpenFOAM dasturii ta'minoti erdamida sonli tad qikoti.

Abstract: Ushbu makolada 1983 yilda Pitz va Daily tomonidan olib borilgan experiment asosida boskichli kengayishda yuzaga keluvchi azhraluvchi turbulent okimning sonli tadqikoti kўrib chikiladi. Masalaning echimi OpenFOAM dasturii taminotidan foidalanilgan holda amalga oshirilgan. Turbulentlikni ҳhisoblash olish uchun k- ε (k-epsilon) va LES (Large Eddy Simulation – yirik girdobli modellashtirish) modellaridan foidalanildi. Modellashtirish natizhalari experimented ma'lumotlar bilan takkoslanib, okim profillaridagi farklar taklil qilindi.

Kalit suzlar: Pitz–Daily, OpenFOAM, LES, k-epsilon , turbulent okim, CFD (hysoblast hydrodynamics)

Nazarov F.Kh., Navruzov DP Numerical Simulation of the Pitz–Daily Turbulent Flow Using OpenFOAM

Abstract. This paper presents a numerical simulation of a separated turbulent flow behind a backward-facing step, based on the experiment conducted by Pitz and Daily in 1983. The simulation was carried out using the OpenFOAM software. The turbulence was modeled using the k- ε (k-epsilon) model and LES (Large Eddy Simulation). The simulation results were compared with experimental data, and differences in flow profiles were analyzed.

Keywords: Pitz–Daily, OpenFOAM, LES, k-epsilon, turbulent flow, CFD (Computational Fluid Dynamics)