ASSESSMENT OF TURBULENCE MODELS FOR CFD SIMULATION OF PARTIAL FLOW BLOCKAGE IN NUCLEAR FUEL ROD BUNDLES USING LOGOS

Jamoldinov Fozilbek Zaylobidin ugli
Junior researcher, Scientific and technical center for radiation and nuclear safety
jamoldinovfozilbek@gamil.com
Umarvayev Khasan Rakhimjon ugli
Assitant, Tashkent state technical university named after I. Karimov
umarvayevkhasan@gmail.com
Khoshimjonov Khurshidbek Farkhodjon ugli
Assistant, National University of Uzbekistan named after Mirzo Ulugbek
xoshimjonovxurshidbek1999@gmail.com
Juraev Sherzod Kuziboy ugli
Engineer*, Scientific and technical center for radiation and nuclear safety*
zuraev395@gmail.com

***Abstract** – Accurate prediction of coolant flow behavior in nuclear fuel rod bundles under partial flow blockage is essential for reactor safety analysis. This study validates the CFD code LOGOS through simulation of hydrodynamic processes in a 61-rod bundle based on MONJU experimental configurations. Numerical results were compared with experimental velocity measurements to assess turbulence model performance. Calculations were performed using the standard k-ε model and the Explicit Algebraic Reynolds Stress Model (EARSM). The k-ε model reproduces general flow characteristics but fails to adequately predict the recirculation region downstream of the blockage. The EARSM model shows significantly improved agreement with experimental data, particularly in capturing reverse flow behavior. Statistical evaluation confirms higher predictive accuracy of the Reynolds stress–based approach. The results demonstrate the importance of turbulence model selection for reliable CFD-based safety assessment of nuclear fuel assemblies.*

***Keywords:** – CFD, LOGOS, nuclear fuel assembly, partial flow blockage, turbulence modeling, EARSM, rod bundle flow, validation.*

INTRODUCTION

Safe operation of nuclear reactors requires reliable prediction of coolant flow behavior inside fuel assemblies under both normal and abnormal operating conditions. One of the important hypothetical scenarios considered in safety analysis is partial blockage of the flow area in a fuel rod bundle. Such blockage may occur due to deformation of structural elements, foreign objects, or debris accumulation and can lead to flow redistribution, formation of recirculation zones, and local overheating of fuel rods. Accurate assessment of these phenomena is therefore essential for maintaining thermal safety margins.

"Innovations in Science and Technologies" илмий-электрон журнали
ISSN: 3030-3451. 3 / 2026 йил.
www.innoist.uz

Computational fluid dynamics (CFD) methods are increasingly used to investigate complex thermo-hydraulic processes in reactor cores, since they allow detailed analysis of local flow structures that cannot be fully resolved using system codes or experimental measurements alone. However, the reliability of CFD predictions strongly depends on model verification and validation against experimental data, particularly for highly anisotropic turbulent flows typical of rod bundle geometries with local disturbances.

Previous experimental studies, including investigations performed on MONJU-type rod bundle configurations, have provided valuable benchmark data describing velocity fields and recirculation behavior downstream of partial blockages. Despite significant progress in CFD applications, accurate prediction of blockage-induced flow separation and reverse flow remains challenging, mainly due to limitations of commonly used turbulence models.

LITERATURE REVIEW

Computational fluid dynamics (CFD) has become an important tool for investigating thermo-hydraulic processes in nuclear reactor systems, enabling detailed analysis of three-dimensional flow structures and turbulence effects that cannot be adequately resolved using system thermal-hydraulic approaches alone [1]. In nuclear fuel assemblies, local flow disturbances may significantly alter velocity distribution and influence heat transfer characteristics, making reliable numerical prediction essential for safety assessment.

Experimental investigations of coolant flow in rod bundle geometries provide valuable benchmark data for validation of CFD models. In particular, non-intrusive experimental studies using particle image velocimetry (PIV) techniques have demonstrated the formation of complex wake structures and flow redistribution inside rod bundles subjected to geometric disturbances [2]. Such experiments are widely used as reference datasets because direct measurements under reactor conditions are limited.

Despite extensive application of CFD methods, accurate prediction of turbulent flow in disturbed bundle configurations remains challenging. Conventional Reynolds-Averaged Navier–Stokes (RANS) turbulence models, including the standard k-ε model, often show limited capability in reproducing anisotropic turbulence and recirculation regions. Reynolds stress–based approaches provide improved representation of complex turbulent structures and therefore are increasingly considered for validation studies of internal reactor flows [3]. Consequently, verification of CFD tools against experimental benchmarks remains a key requirement for reliable application in nuclear safety analyses.

METHODOLOGY
Experimental Configuration

The experimental benchmark represents a 61-rod bundle installed inside a hexagonal channel corresponding to a MONJU-type fuel assembly geometry. Cylindrical rods were arranged in a triangular lattice, and a partial flow blockage was

"Innovations in Science and Technologies" илмий–электрон журнали
ISSN: 3030-3451. 3 / 2026 йил.
www.innoist.uz

introduced using a flat impermeable plate located within selected subchannels. The blockage thickness was 20 mm, and the distance from the inlet to the blockage position was 200 mm.

Experiments were conducted in a water test loop under controlled flow conditions corresponding to Reynolds numbers of 20,000, 29,000, and 39,000, with inlet velocities of 1.34 m/s, 1.943 m/s, and 2.613 m/s, respectively. Longitudinal velocity distributions downstream of the blockage were measured at several axial locations and used as benchmark data for validation of numerical simulations.

Computational Model

A three-dimensional model reproducing the experimental geometry was developed and implemented in the LOGOS CFD software. The computational domain included the full rod bundle and blockage region.

The numerical mesh consisted of approximately 49.7 million cells, providing sufficient spatial resolution to capture velocity gradients and recirculation structures. Local mesh refinement was applied near rod surfaces and in the blockage region to improve resolution of near-wall flow behavior.

The governing equations were solved within the Reynolds-Averaged Navier–Stokes (RANS) framework under steady-state conditions. Boundary conditions were defined as follows:

- Inlet: uniform velocity corresponding to experimental conditions;
- Outlet: pressure outlet (subsonic);
- Walls and rod surfaces: no-slip condition;
- Coolant temperature: 295 K.

The coolant was modeled as incompressible water with constant thermophysical properties.

Turbulence Modeling

Two turbulence models were investigated to evaluate their predictive capability:

Standard k-ε model, commonly used in engineering CFD simulations due to its numerical robustness and computational efficiency;

Explicit Algebraic Reynolds Stress Model (EARSM), capable of accounting for turbulence anisotropy and improved prediction of separated and recirculating flows.

Both models were applied under identical computational conditions to enable direct comparison with experimental observations.

Validation Procedure

Validation was performed by comparing calculated longitudinal velocity profiles with experimental measurements at several downstream locations normalized by the equivalent blockage diameter x/D_b. The agreement between numerical and experimental results was quantified using the relative deviation parameter:

$$E_n = \frac{S_n - D_n}{U}$$

where S_n is the calculated velocity, D_n is the corresponding experimental value, and U is the bulk inlet velocity. Statistical indicators, including mean relative deviation

"Innovations in Science and Technologies" илмий-электрон журнали
ISSN: 3030-3451. 3 / 2026 йил.
www.innoist.uz

and standard deviation, were used to evaluate the predictive accuracy of the turbulence models.

ANALYSIS AND RESULTS

Flow Structure Behind the Blockage

Numerical simulations performed using the LOGOS CFD code demonstrated the formation of a disturbed flow region downstream of the blockage. The calculated velocity field shows flow separation at the blockage edge followed by development of a recirculation zone. The obtained flow pattern qualitatively agrees with experimental flow visualization results, where reverse flow and wake formation were observed behind the obstruction.
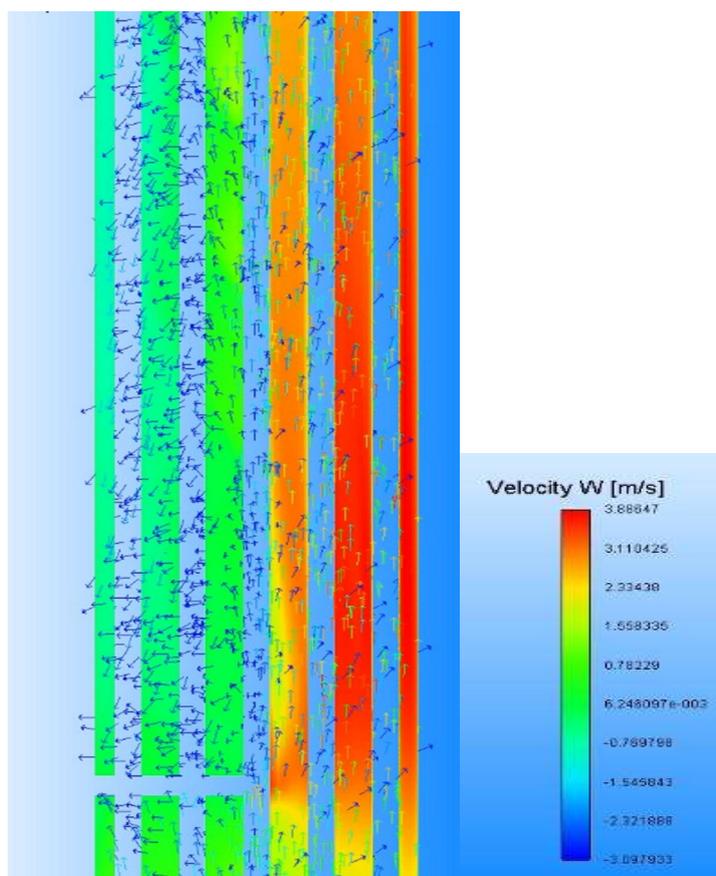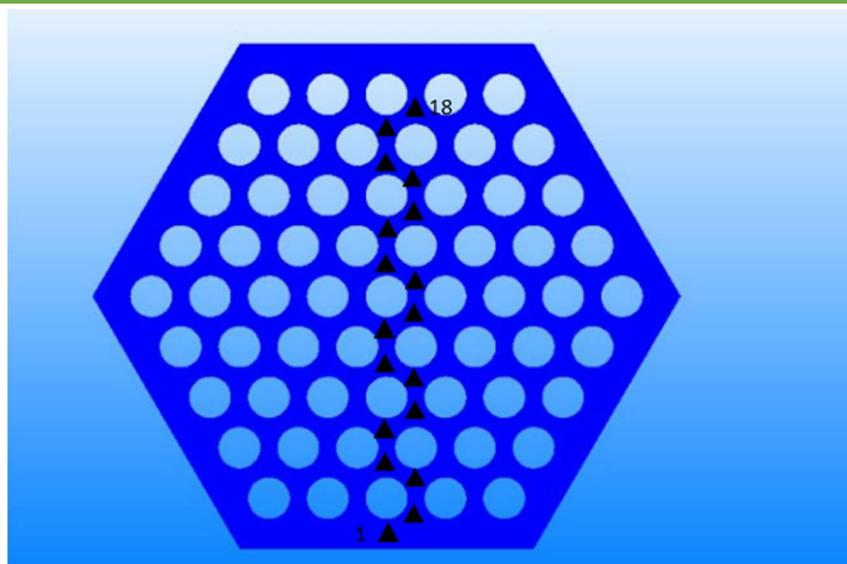

Figure 1 — Velocity field in the longitudinal section of the bundle (EARSM model, inlet velocity 1.34 m/s).

The results confirm that blockage leads to acceleration of the flow in peripheral subchannels and velocity reduction in the central region.

### Measurement Line and Velocity Normalization

Experimental validation was performed along a predefined measurement line downstream of the blockage. The longitudinal velocity was normalized by the bulk inlet velocity, allowing direct comparison between numerical and experimental results.

"Innovations in Science and Technologies" илмий-электрон журнали
ISSN: 3030-3451. 3 / 2026 йил.
www.innoist.uz

Figure 2 — Measurement line used for velocity comparison.

Velocity Profiles: $k - \varepsilon$ **Turbulence Model.** Near-blockage region (x/Db = 0)
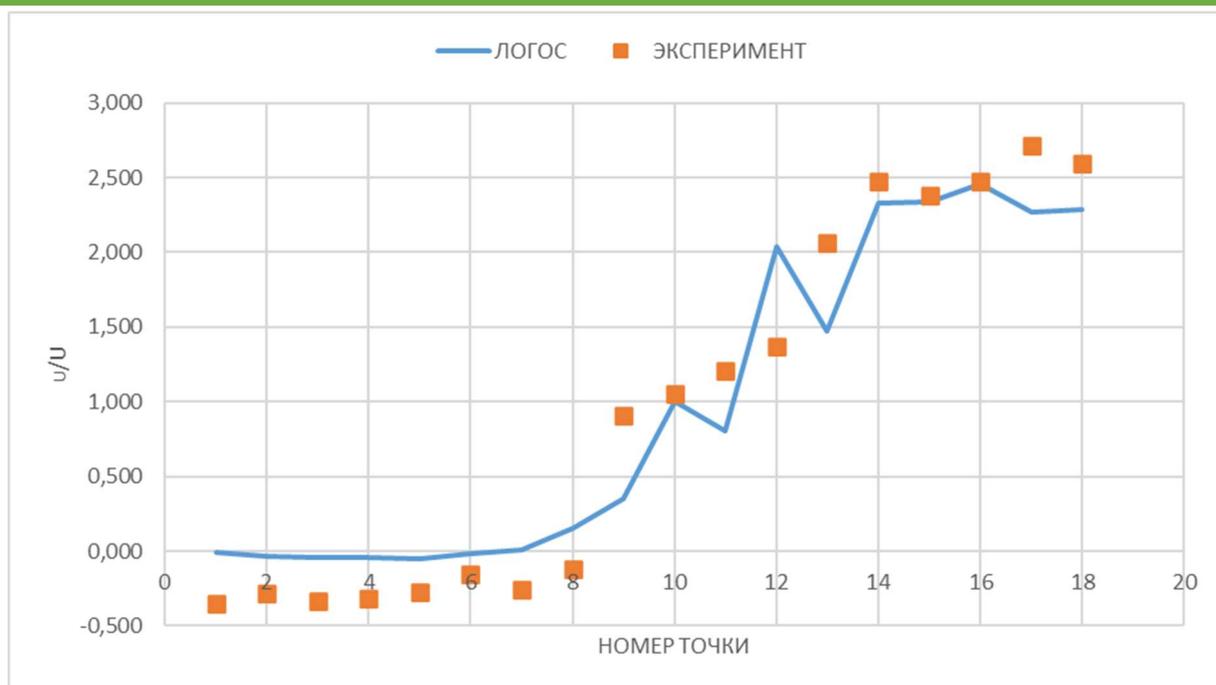
Close to the blockage location, calculated velocity profiles obtained using the k-ε model reproduce the general shape of the experimental distribution. However, quantitative discrepancies are visible, especially in regions of strong velocity gradients.
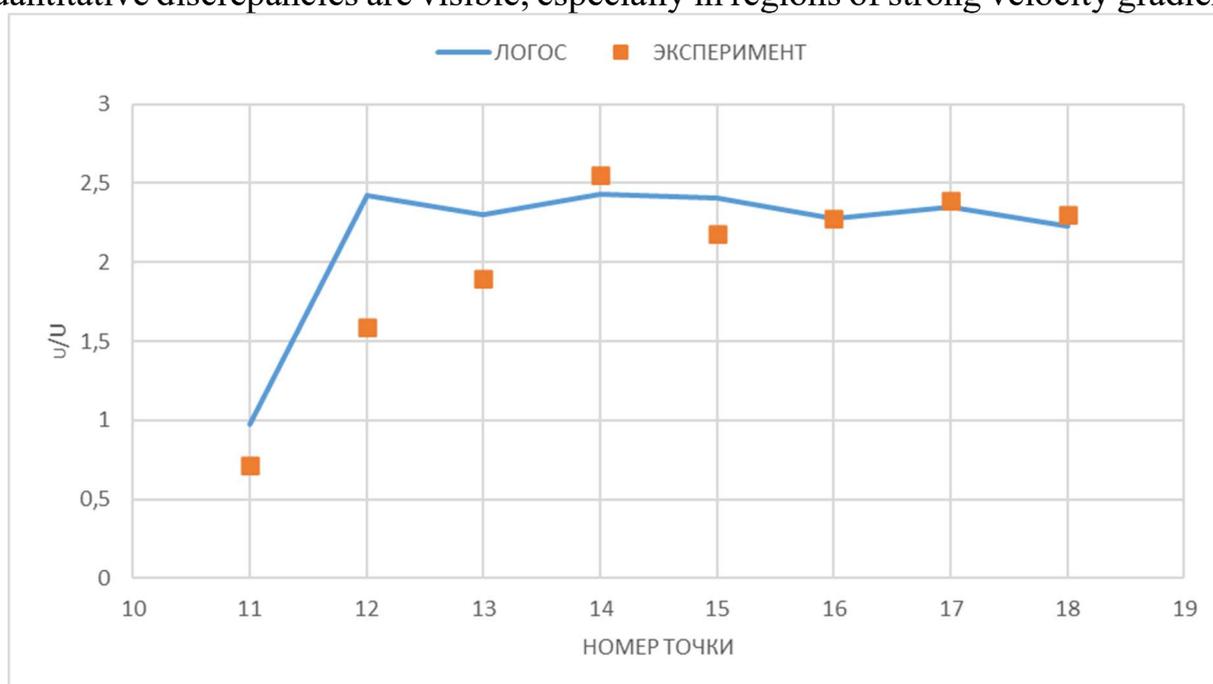
Figure 3 — Normalized longitudinal velocity profile at x/Db = 0 (U = 1.34 m/s, k-ε model).

Recirculation region (x/Db = 0.83)

At a short downstream distance, experiments show clear reverse flow characterized by negative velocity values. The k-ε model predicts significantly weaker reverse flow and, in several locations, fails to reproduce it.

"Innovations in Science and Technologies" илмий–электрон журнали
ISSN: 3030-3451. 3 / 2026 йил.
 www.innoist.uz

Figure 4 — Velocity profile at x/Db = 0.83 (U = 1.34 m/s, k-ε model).
This indicates insufficient representation of turbulence anisotropy.
Intermediate downstream region (x/Db = 2.4)
Further downstream, calculated profiles begin to deviate from experimental data, showing delayed recovery of the velocity field.
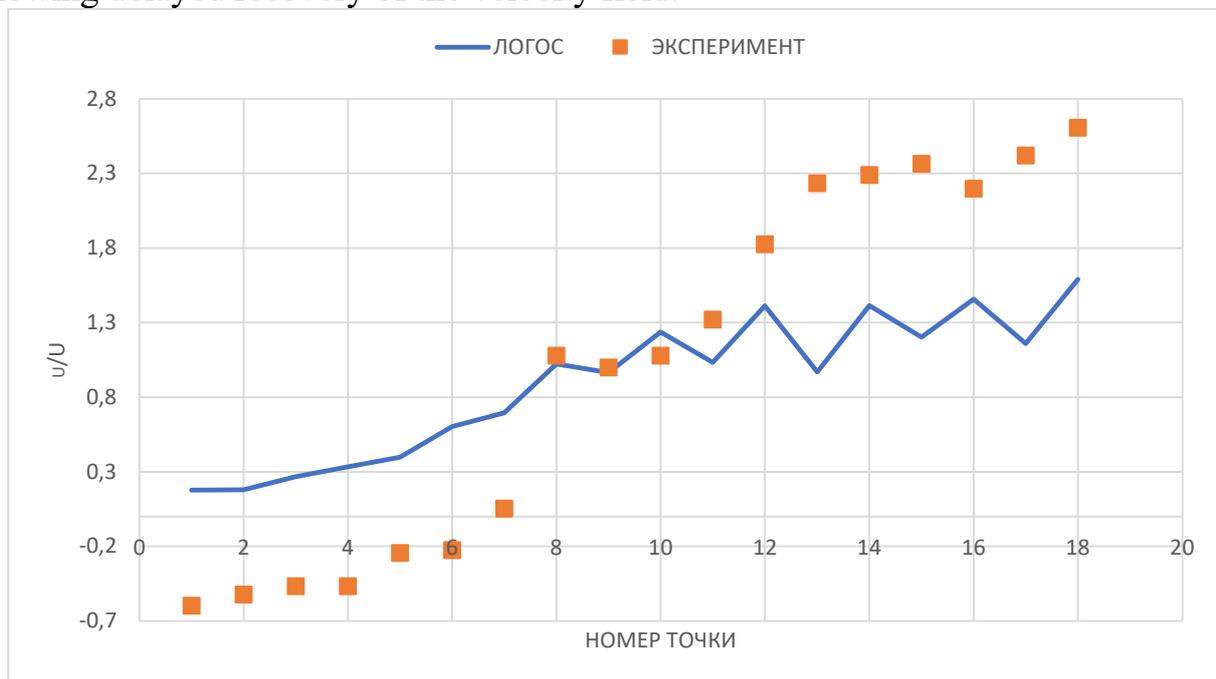
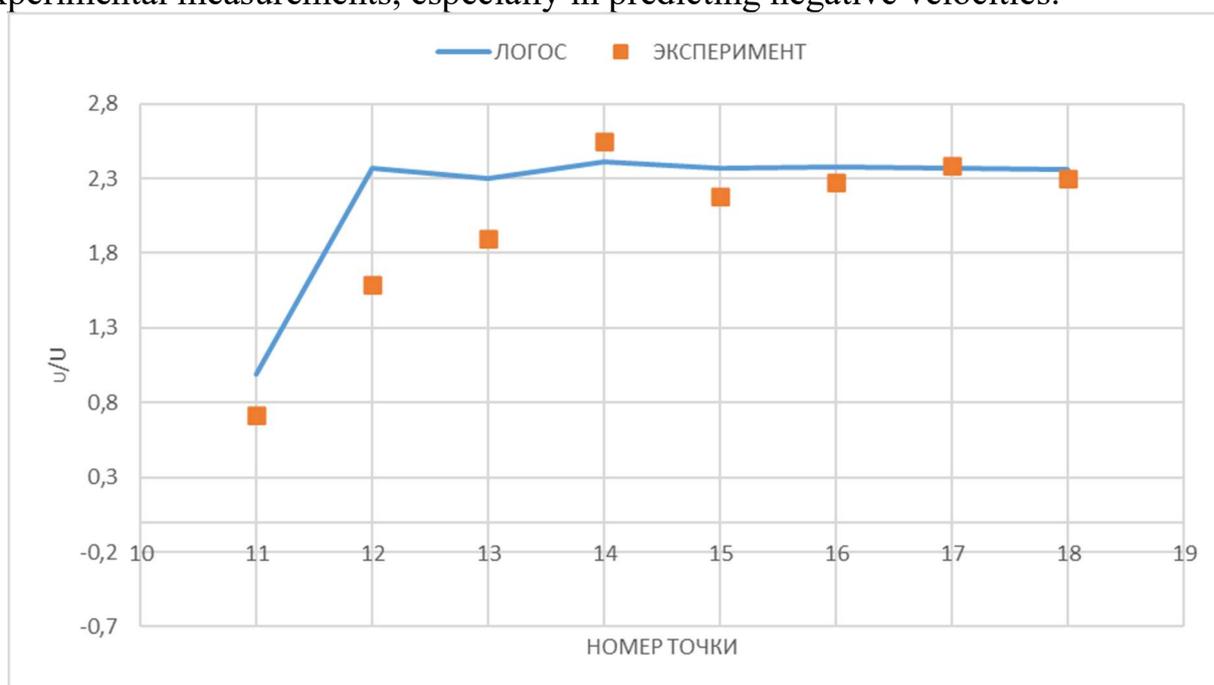**Figure 5** — Velocity profile at x/Db = 2.4 (U = 1.34 m/s, k-ε model).

Velocity Profiles: EARSM Turbulence Model
To improve prediction accuracy, simulations were repeated using the EARSM turbulence model.
Near-blockage agreement

"Innovations in Science and Technologies" илмий-электрон журнали
ISSN: 3030-3451. 3 / 2026 йил.
www.innoist.uz

The EARSM model demonstrates significantly improved agreement with experimental measurements, especially in predicting negative velocities.

Figure 6 — Velocity profile at x/Db = 0 (U = 1.34 m/s, EARSM model).

Recirculation prediction

Unlike the k-ε model, EARSM successfully reproduces the recirculation region and reverse flow magnitude.
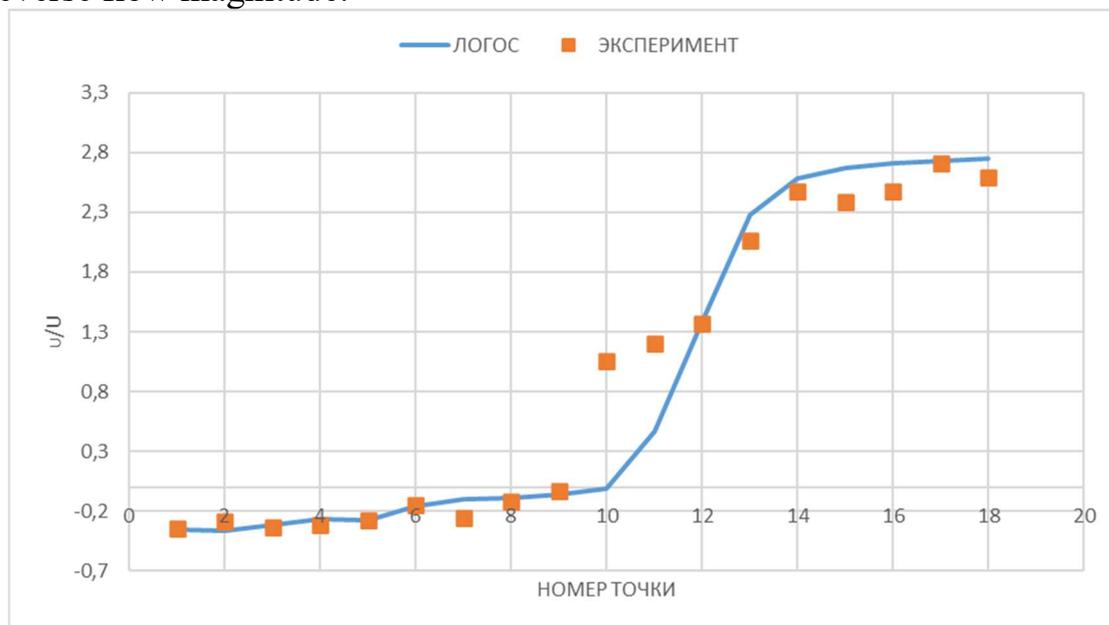
Figure 8 — Velocity profile at x/Db = 0.83 (EARSM model).

Far downstream region

Even with EARSM, calculated disturbances decay slower than in experiments.

"Innovations in Science and Technologies" илмий–электрон журнали
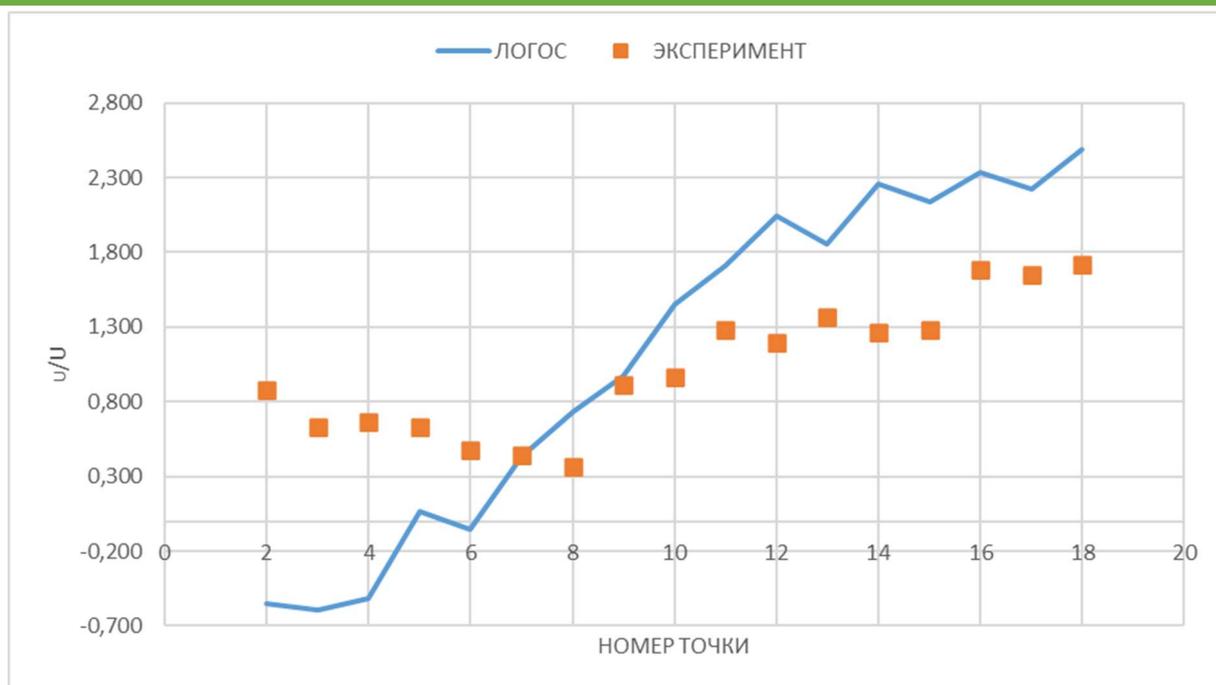**ISSN: 3030–3451. 3 / 2026 йил.**
www.innoist.uz

Figure 9 — Velocity profile at x/Db = 5.55 (EARSM model).

Statistical Validation

Quantitative comparison was performed using relative deviation analysis between calculated and experimental velocities. Statistical results show that EARSM provides smaller deviation and reduced dispersion compared with the k-ε model, confirming improved predictive capability.

Table 1

Statistical Validation Results for the k-ε Model

Model	Inlet velocity	x/Db	$\overline{E}$	u_E	$\overline{E} - u_E$	$\overline{E} + u_E$
$K - \varepsilon$	U=1.34	0	0.141	0.240	-0.099	0.382
		0.83	0.006	0.264	-0.258	0.270
		2.4	-0.077	0.594	-0.671	0.517
		3.97	0.238	0.541	-0.303	0.779
		5.55	-0.036	0.344	-0.380	0.309
	U=1.943	0	0.066	0.234	-0.169	0.300
		0.83	0.036	0.291	-0.327	0.255
		2.4	-0.073	0.525	-0.599	0.452
		3.97	0.133	0.412	-0.279	0.545
		5.55	-0.015	0.218	-0.077	0.359
	U=2.613	0	0.055	0.117	-0.062	0.172
		0.83	-0.014	0.155	-0.169	0.141
		2.4	-0.042	0.325	-0.367	0.284
		3.97	0.115	0.291	-0.176	0.406
		5.55	-0.089	0.138	0.003	0.279

"Innovations in Science and Technologies" илмий-электрон журнали
ISSN: 3030-3451. 3 / 2026 йил.
www.innoist.uz

Table 2

Statistical Validation Results for the EARSM Model

Model	Inlet velocity	x/Db	$\bar{E}$	u_E	$\bar{E} - u_E$	$\bar{E} + u_E$
$K-w$	U=1.34	0	0.155	0.216	-0.061	0.371
		0.83	-0.028	0.25	-0.278	0.223
		2.4	0.028	0.166	-0.138	0.194
		3.97	0.305	0.314	-0.008	0.619
		5.55	0.07	0.589	-0.52	0.659
	U=1.943	0	0.059	0.234	-0.293	0.175
		0.83	0.03	0.198	-0.228	0.168
		2.4	0.029	0.212	-0.241	0.183
		3.97	0.179	0.259	-0.08	0.439
		5.55	0.076	0.424	-0.283	0.565
	U=2.613	0	0.054	0.016	0.107	-0.052
		0.83	0	0.122	-0.122	0.122
		2.4	-0.008	0.135	-0.143	0.127
		3.97	0.171	0.193	-0.022	0.364
		5.55	-0.032	0.334	-0.193	0.475

Discussion of Model Performance

The results demonstrate that the standard k-ε model reproduces overall flow redistribution but fails to capture recirculation physics behind the blockage. The EARSM model accounts for turbulence anisotropy and therefore provides more accurate prediction of separated flow structures. Remaining discrepancies downstream indicate limitations related to turbulence diffusion modeling rather than geometric representation.

The underprediction of reverse flow by the k-ε model is associated with its isotropic turbulence assumption, which limits accurate representation of Reynolds stress redistribution in separated flows. The EARSM model accounts for turbulence anisotropy, allowing improved prediction of wake structures formed behind the blockage.

CONCLUSION

The present study investigated the capability of the LOGOS CFD code to simulate coolant flow behavior in a nuclear fuel rod bundle under partial flow blockage conditions using experimental benchmark data. Numerical simulations successfully reproduced the main hydrodynamic features observed in experiments, including flow redistribution and formation of a wake region downstream of the blockage.

Comparison of turbulence models demonstrated that the standard k-ε model adequately predicts overall velocity trends but fails to accurately capture the recirculation region and reverse flow observed experimentally. In contrast, the Explicit

Algebraic Reynolds Stress Model (EARSM) provided significantly improved agreement with measurements, particularly in regions characterized by strong turbulence anisotropy and flow separation.

Statistical validation confirmed higher predictive accuracy of the Reynolds stress–based approach. Nevertheless, both models showed slower decay of flow disturbances at larger downstream distances compared with experimental observations, indicating limitations related to turbulence diffusion modeling.

The results highlight that turbulence model selection plays a critical role in CFD-based safety analysis of nuclear fuel assemblies. The obtained findings confirm the applicability of the LOGOS code for verification calculations involving complex internal flows and demonstrate the advantages of advanced turbulence modeling for prediction of blockage-induced flow phenomena.

REFERENCES

1. Ferziger J.H., Perić M. Computational Methods for Fluid Dynamics. 3rd ed. — Berlin: Springer, 2002. — 423 p.

2. Dominguez-Ontiveros E.E., Hassan Y.A. Non-intrusive experimental investigation of flow behavior inside a 5×5 rod bundle with spacer grids using PIV and MIR // Nuclear Engineering and Design. — 2009. — Vol. 239, No. 5. — P. 888–898. https://doi.org/10.1016/j.nucengdes.2009.01.009.

3. Wallin S., Johansson A.V. An explicit algebraic Reynolds stress model for incompressible and compressible turbulent flows // Journal of Fluid Mechanics. — 2000. — Vol. 403. — P. 89–132.

4. Todreas N.E., Kazimi M.S. Nuclear Systems I: Thermal Hydraulic Fundamentals. — Boca Raton: CRC Press, 2011. — 891 p.

5. Launder B.E., Spalding D.B. The numerical computation of turbulent flows // Computer Methods in Applied Mechanics and Engineering. — 1974. — Vol. 3. — P. 269–289.

6. Pope S.B. Turbulent Flows. — Cambridge: Cambridge University Press, 2000. — 771 p.

7. Wilcox D.C. Turbulence Modeling for CFD. 3rd ed. — La Cañada: DCW Industries, 2006. — 522 p.

8. Menter F.R. Two-equation eddy-viscosity turbulence models for engineering applications // AIAA Journal. — 1994. — Vol. 32, No. 8. — P. 1598–1605.

9. Baglietto E., Ninokata H. CFD analysis of turbulent flow in wire-wrapped fuel bundles // Nuclear Engineering and Design. — 2005. — Vol. 235. — P. 1183–1195.

10. Xiong J., Yu Y., Liu H. Validation for CFD simulation in rod bundles with split-type mixing vanes // Frontiers in Energy Research. — 2020. — Vol. 8. — Article 43. https://doi.org/10.3389/fenrg.2020.00043.

11. Kim S., Chung M.K. Experimental and numerical investigation of turbulent mixing in rod bundle geometry // International Journal of Heat and Fluid Flow. — 2001. — Vol. 22. — P. 350–363.

"Innovations in Science and Technologies" илмий-электрон журнали
ISSN: 3030-3451. 3 / 2026 йил.
www.innoist.uz

12. OECD/NEA. Benchmark for CFD Codes Applied to Nuclear Reactor Safety (CFD4NRS). — Paris: OECD Nuclear Energy Agency, 2012.

13. IAEA. Safety Assessment for Facilities and Activities. IAEA Safety Standards Series No. GSR Part 4. — Vienna: International Atomic Energy Agency, 2016.