UDC 532.5:519.63:629.735.05
DOI: 10.15587/1729-4061.2025.329024

# BUILDING A MODEL OF THE FLOW IN A NOZZLE-FLAPPER VALVE OF THE HP-3 CONTROL PUMP TO IMPROVE THE STABILITY OF CHARACTERISTICS

**Oleksandr Lytviak**
Doctor of Technical Sciences*
**Roman Trishch**
Doctor of Technical Sciences*
**Eduard Khomiak**
*Corresponding author*
PhD*
E-mail: e.khomiak@khai.edu
**Serhii Kochuk**
PhD*
**Svitlana Khomenko**
PhD
Department of Vocational Education, Labor Training and Technologies
Berdyansk State Pedagogical University
Zhukovskoho str., 55A, Zaporizhzhia, Ukraine, 69600
**Ihor Tiupa**
PhD
Department of Automation, Metrology and Energy-Efficient Technologies
V. N. Karazin Kharkiv National University
Svobody sq., 4, Kharkiv, Ukraine, 61022
*Department of Mechatronics and Electrical Engineering
National Aerospace University "Kharkiv Aviation Institute"
Vadyma Manka str., 17, Khakriv, Ukraine, 61070

***The object of this study is the flow of a viscous incompressible fluid in a nozzle-flapper valve used as part of the free turbine speed controller in the HP-3 pump-regulator of the TV3-117 turboprop helicopter engine. The task addressed relates to a need for detailed calculations of the fluid flow because of unsatisfactory operation of the valve under actual operating conditions. An additional difficulty was the contradictory data on the characteristics of such valves in the literature, which made it impossible to determine the flow characteristics and directions for improving the design.***

***This paper reports the results of numerical calculations of the flow in the valve performed in the SolidWorks Flow Simulation environment. A mathematical model is proposed that takes into account the influence of the design mesh on the accuracy and computational time volume, as well as ways to improve accuracy without a significant increase in resources. The model was verified by comparing it with the manufacturer's experimental data. The results have made it possible to solve the problem through the detailed construction of the model taking into account the valve geometry and optimization of the computational mesh, which ensured a balance between accuracy and computational speed.***

***The results are attributed to the application of state-of-the-art hydrodynamic calculation software, precise mesh tuning, as well as proper validation of the model to reflect the actual physical processes in the valve. The model built makes it possible to study the flow in the valve and could be used to analyze the impact of manufacturing defects. The model is suitable for parametric studies and modification of valves in helicopter engines of the TV3-117 type or similar systems. The model could also be adapted for other systems requiring flow analysis in similar valves***

***Keywords: nozzle-flapper, numerical modeling, stagnant zone, vortex flow***

*Received 27.02.2025*
*Received in revised form 16.04.2025*
*Accepted 05.05.2025*
*Published 27.06.2025*

***How to Cite:*** *Lytviak, O., Trishch, R., Khomiak, E., Kochuk, S., Khomenko, S., Tiupa, I. (2025). Building a model of the flow in a nozzle-flapper valve of the HP-3 control pump to improve the stability of characteristics. Eastern-European Journal of Enterprise Technologies, 3 (1 (135)), 51–57. https://doi.org/10.15587/1729-4061.2025.329024*

## 1. Introduction

Physical experiments are expensive and time-consuming, especially at the stages of manufacturing and debugging prototypes. One way to reduce costs is to numerically simulate physical processes in hydraulic systems, in particular in nozzle-gate valves, to calculate their integral characteristics. Under current conditions, the study of viscous fluid flow in such valves is relevant since it makes it possible to eliminate the instability of aviation systems and reduce the cost of physical experiments. The relevance is confirmed by a number of studies. Work [1] emphasizes that numerical modeling in SolidWorks is an effective tool for analyzing hydraulic systems, which reduces the need for expensive physical tests, providing high accuracy of prediction. Study [2] notes the complexity of modeling turbulent flows in small hydraulic elements, which indicates the need to improve numerical methods to ensure the stability of characteristics. In [3], attention is focused on the problem of instability of nozzle-flap valve characteristics due to vortex effects, which emphasizes the need for detailed modeling to improve the performance of the systems. In [4], it is shown that the instability of the speed controller in turboshaft engines causes rotor oscillations, confirming the importance of the stability of hydraulic components for aircraft engines.

In [5], the quality of solenoid valves was investigated using the Six Sigma methodology, demonstrating the importance of accurate analysis of design features to ensure stability, similar to the needs of aviation valves. In [6], it was noted that the instability of hydraulic valves in transportation systems reduces safety and efficiency, which is similar to the problems of aviation systems, where valve stability is critical. In [7], the need to reduce the cost and time for diagnostics of aviation equipment due to the complexity of its components is emphasized, which justifies the use of numerical modeling for optimizing designs. These works confirm that research aimed at increasing the stability of hydraulic valve characteristics through numerical modeling is relevant for ensuring the reliability and efficiency of complex technical systems. The lack of data on the nozzle-flap valves of the HP-3 pump-regulator in the TV3-117 engine, in particular due to the contradictory results of experiments, causes instability of the characteristics, which leads to unacceptable fluctuations in the frequency of rotation of the helicopter's main rotor.

Thus, studies on numerical modeling of the flow in nozzle-flap valves are relevant for ensuring the quality and stability of the operation of aviation systems.

## 2. Literature review and problem statement

In [8], a numerical simulation of the flow in a hydraulic control valve with a variable throttle area used in aviation systems was performed. Optimization of geometric parameters, such as the position of the flap, reduces the vortex zones by 15% and improves the stability of the flow. However, the study does not consider the specific features of nozzle-flap valves, such as the HP-3 of the TV3-117 engine, where local vortex effects and stagnation zones significantly affect the stability of the regulator. This indicates the need to build models that take these factors into account. In [9], a numerical simulation of transient flow regimes in RANS models for large hydraulic systems was investigated. It was shown that these models are effective for predicting flows, but they do not take into account local vortex zones in small valves, such as nozzle-flap valves, due to the complexity of setting up the computational grid. This limits their applicability to the HP-3 valve, where vortex effects are critical.

In study [10], the use of neural networks for the reconstruction of the flow around the cylinder was proposed, which increases the accuracy of predicting turbulent flows. However, the specificity of nozzle-flap valves is not considered, and the method requires significant computational resources, which complicates its application to aviation systems, in particular for the HP-3 valve.

In [11], large-eddy simulation (LES) of the flow around the cylinder was performed using a neural network for the sub grid model. Although the method provides high accuracy in predicting vortex zones, adaptation to the geometry of nozzle-flap valves was not carried out due to the difference in designs, which reduces the relevance of the results for this object.

Paper [12] describes the capabilities of SolidWorks Flow Simulation for hydrodynamic calculations. The tool is effective for flow modeling but the issue of model verification for small valves, such as the HP-3, remains open due to the lack of detail under specific geometric features and operating conditions.

Study [13] considers the selection of valves for control circuits, emphasizing the importance of hydraulic resistance. However, the effect of vortex zones on the stability of characteristics is not investigated due to the simplified modeling approach, which does not solve the problem of conflicting data on the HP-3 valve flow coefficient.

The use of deep learning for large-eddy modeling of turbulent flows in wall flows is analyzed in [14]. The method shows potential but the lack of adaptation to small hydraulic valves, such as nozzle-flap valves, limits its applicability to aviation systems.

In works [15–18], aspects of quality and safety management of technical systems are investigated but they do not concern hydrodynamics. In particular, in [15], qualimetry methods for assessing the occupational safety management system are considered, and in [16, 17], functional dependences for assessing safety are investigated, without an emphasis on hydraulic components. In [18], methods for quality control over fuel element cladding of nuclear reactors are proposed, which is not related to aviation valves. These works do not solve the problem of instability of the characteristics of the HP-3 valve due to the lack of connection with hydrodynamics.

In [11, 19–23], large-eddy modeling and machine learning for turbulent flows were investigated. In particular, in [11], LES of the flow around a cylinder was performed with a neural sub grid model, and in [19], a wall model for LES of the flow around a cylinder was proposed. In [20, 21], hydraulic accumulator monitoring systems were described, and in [22, 23], neural networks were used to model turbomachinery flows and sub grid stresses. These works demonstrate progress in flow modeling but are not adapted to the geometry and operating conditions of nozzle-flap valves, which reduces their applicability to the NR-3 valve.

In [24, 25], active control of flows around a cylinder using machine learning and deep learning was considered. Although these methods are promising for modeling turbulent flows, they do not take into account the specificity of small hydraulic valves, such as the HP-3, due to differences in geometry and operating modes.

Our review of the literature demonstrates that, despite significant progress in numerical flow modeling and stability analysis of hydraulic systems, the issues of flow modeling in the nozzle-gate valves of the HP-3 pump-regulator remain insufficiently studied. The main gaps are:

– lack of detailed models that take into account vortex and stagnation zones in small valves;

– contradictory data on the flow coefficient due to the complexity of the experiments;

– limited adaptation of modern methods, such as LES or neural networks, to aviation valves due to their specific geometry and operating modes.

Those gaps cause instability of the characteristics of the NR-3 valve, which leads to fluctuations in the frequency of rotation of the helicopter main rotor.

Thus, it is advisable to build a numerical model of the flow in the NR-3 nozzle-flap valve to eliminate contradictions and improve the stability of its operation.

## 3. The aim and objectives of the study

The purpose of our study is to build a computer model of the flow of a viscous incompressible fluid in the nozzle-damper valve of the pump-regulator HP-3 to eliminate contradictions in the flow characteristics and take into account nonlinear effects and design features. This will make it possible to increase the stability of the operation of the free turbine speed regulator of the TV3-117 engine and optimize the valve design for practical use in aviation systems.

To achieve the goal, the following tasks were set:

– to construct a numerical flow model in the SolidWorks Flow Simulation environment taking into account vortex flow and stagnant zones;

– to determine the optimal parameters of the computational grid to ensure a balance between calculation accuracy and time consumption;

– to analyze the influence of the geometric parameters of the valve (nozzle diameter, damper position) on the flow characteristics, in particular the flow coefficient and Reynolds number.

## 4. The study materials and methods

The object of our study is the flow of viscous incompressible fluid in the nozzle-gate valve of the HP-3 pump-regulator of the TV3-117 engine, which affects the stability of the free turbine speed regulator. The nozzle-gate valve is a key element of the control system, but its operation under actual operating conditions is characterized by instability due to the complex flow of viscous fluid. The structural diagram of the valve that we had built based on the technical documentation for the HP-3 pump-regulator, is shown in Fig. 1; it includes a nozzle, a gate, and an annular gap, where vortex and stagnation zones are formed, affecting the hydraulic resistance and flow rate.

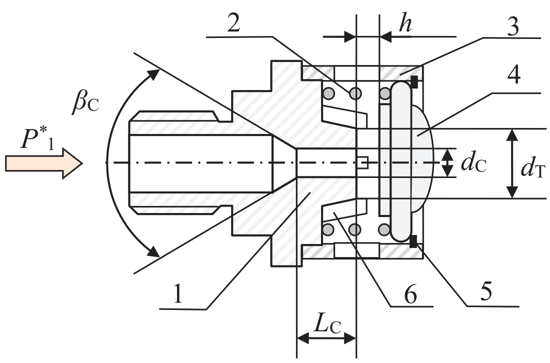



Fig. 1. Structural diagram of a nozzle-deflector type valve: 1 — nozzle; 2 — stabilizing spring; 3 — sleeve; 4 — deflector; 5 — retaining ring; 6 — blades

The instability of the valve characteristics causes fluctuations in the frequency of rotation of the helicopter's main rotor. Conflicting experimental data on the flow coefficient, as well as limited information on vortex effects in small hydraulic valves, complicate the development of recommendations for improving the design. This necessitates the need for numerical simulation of the flow to clarify the physical processes in the valve and increase the reliability of the TV3-117 engine. The main design parameters of the valve are $d_C$ – nozzle diameter; $h$ – deflector position; $d_T$ – outer diameter of the nozzle end; $L_C$ – length of the cylindrical section of the nozzle; $\beta_C$ – angle of fluid entry into the nozzle.

The hypothesis of the study assumes that numerical simulation of the flow taking into account vortex and stagnation zones in the valve could make it possible to identify the causes of instability of its characteristics and offer recommendations for improving the design.

The following assumptions were adopted for the simulation: the fluid is incompressible with constant viscosity; the flow is stationary within the specified conditions; the fluid temperature is constant, which eliminates thermal effects. Simplifications include ignoring microroughness of valve surfaces; using simplified valve geometry that retains the main design features (nozzle diameter, flap position, fluid inlet angle); eliminating the influence of engine vibrations.

The SolidWorks Flow Simulation software package was chosen for the study, which is based on the finite volume method for solving the Navier-Stokes equations. The choice is justified by the high accuracy of modeling turbulent flows in complex geometries, the ability to automatically generate an adaptive mesh, and integration with CAD models, which simplifies the preparation of the valve geometry. The numerical model was built by solving the Navier-Stokes equations for a viscous incompressible fluid, taking into account vortex and stagnation zones. The computational mesh was generated automatically with adaptive sealing in critical zones (nozzle entrance, annular gap). To assess the optimality of the mesh, a sensitivity analysis was performed by varying the cell size, taking into account the accuracy of predicting vortex zones and the calculation time. With coarse discretization, small fragments of the valve are ignored, which leads to a violation of the no-slip conditions and the appearance of "sources" and "sinks" of the liquid, as shown in Fig. 2, 3.

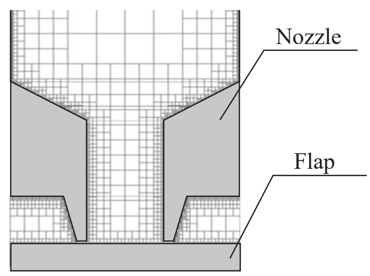



Fig. 2. Calculation grid in the longitudinal section of the valve

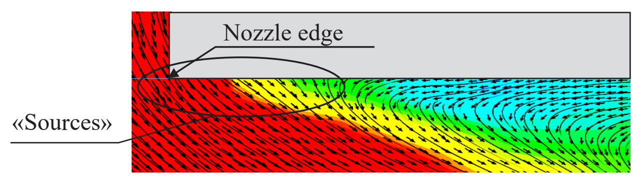
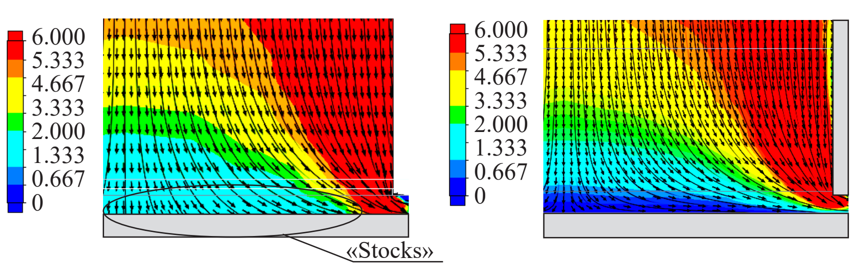



Fig. 3. Formation of "sources" when flowing around a sharp edge

In stagnation zones, insufficiently fine-grained cell division causes violation of no-slip boundary conditions, which reduces the accuracy of the modeling. To analyze the influence of the grid size, a comparison of the flow in the stagnation zone at different discretization steps was carried out, as shown in Fig. 4.




*a* *b*

Fig. 4. Formation of "sinks" in the area of stagnant flow: *a* — valve in the stagnation zone $\Delta = 0.1$ mm; *b* — valve in the stagnation zone $\Delta = 0.001$ mm

To improve the accuracy, the nozzle and valve were modeled as a set of elements, which made it possible to build local grids with a small step in critical zones. This approach ensured correct compliance with boundary conditions and accurate reproduction of vortex zones and hydraulic resistance.

## 5. Results of investigating the flow in a nozzle-gate valve of the pump-regulator HP-3

### 5. 1. Construction of a numerical flow model taking into account the vortex flow and stagnation zones

Our numerical model of the flow in the nozzle-gate valve HP-3 reproduces the vortex flow and stagnation zones. The results of calculating the velocity field in the valve cross section are shown in Fig. 5.

A stagnant zone is formed when the flow interacts with the flap, acting as a rigid body and enabling a smooth turn of the jets into the annular gap. A vortex flow is formed at the nozzle entrance. A detailed analysis of the flow in the nozzle is shown in Fig. 6.

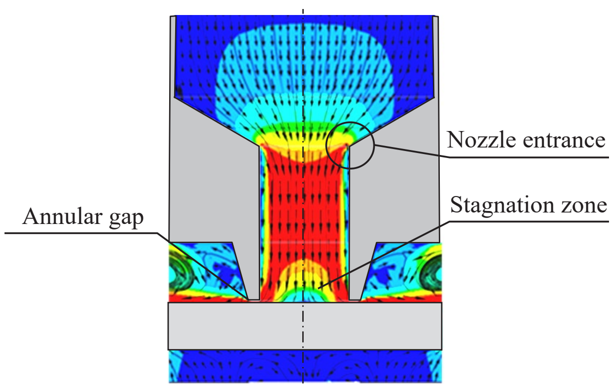



Fig. 5. Velocity field and streamlines in the valve cross section

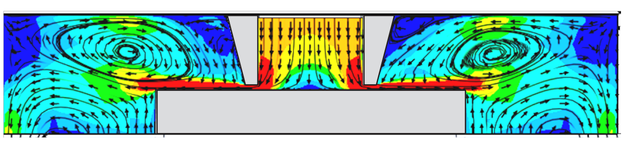


Fig. 6. Velocity field and streamlines in the nozzle cross section

The flow constriction zone makes it possible to estimate the hydraulic resistance. The model provides an error in predicting the hydraulic resistance of up to 5% compared to the manufacturer's data, which confirms its ability to reproduce the physical processes in the valve.

The numerical model is built for the Reynolds number Re = 7300 and the damper position $h$ = 0.07 mm, which corresponds to typical operating conditions of the HP-3 valve. The error in predicting the hydraulic resistance is up to 5% compared to the manufacturer's experimental data, which confirms the accuracy of the model. The model provides computational efficiency, making it possible to analyze the flow in the valve and the "shirt" area without excessive time costs.

### 5. 2. Determining the optimal parameters for a computational grid

Optimization of the computational grid was carried out to ensure a balance between the accuracy of calculations and the time spent in modeling the flow in the nozzle-gate valve HP-3. An adaptive grid with local compaction in critical zones (nozzle inlet, annular gap) was used. The sensitivity analysis of the grid revealed that the grid step $\Delta$ = 0.001 mm provides an error in predicting Reynolds numbers up to 3% compared to the manufacturer's data, while maintaining high accuracy in the vortex flow and stagnation zones.

Fig. 7, 8 illustrate the velocity field and streamlines, demonstrating the capabilities of the model and highlighting critical zones (nozzle inlet, annular gap) for further detailed analysis.

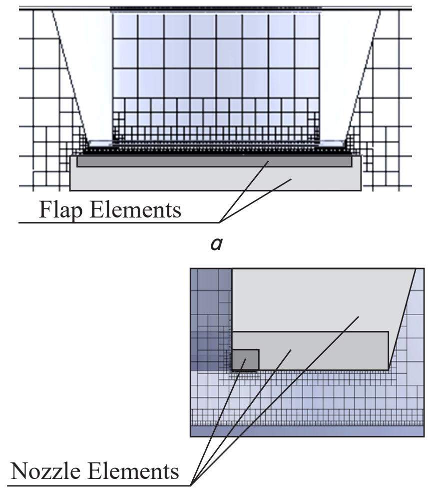



Fig. 7. Calculation grid in the cross-section of the nozzle of the nozzle valve-deflector: *a* – valve elements; *b* – nozzle elements

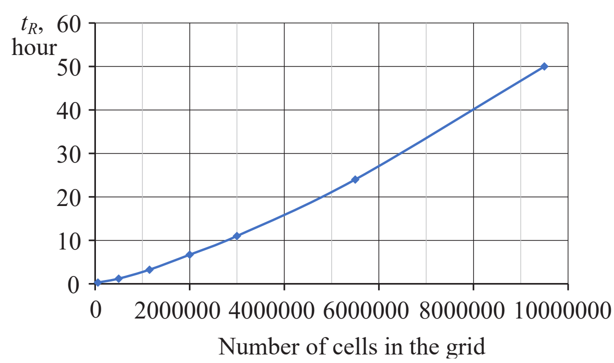



Fig. 8. CPU time cost depending on the number of grid cells

These results reveal the flow behavior in the nozzle area, which is specified below:

– the flow constriction zone is marked with a marker to estimate the hydraulic resistance;

– the model takes into account the vortex flow and stagnation zones as key elements of the valve behavior.

Local grids with a step size of 0.001 mm made it possible to accurately reproduce the flow separation near sharp edges and stagnant zones, avoiding violations of the no-slip boundary conditions. Reducing the step size below 0.001 mm did not improve the accuracy but significantly increased the calculation time. The dependence of the processor time on the number of grid cells is shown in Fig. 8 (Intel Core i7-1165G7 processor).

The optimal mesh of 500 thousand cells reduced the calculation time by 30% compared to the coarser meshes (1 million cells), providing an error of up to 5% for the hydraulic resistance. This confirms the effectiveness of the chosen approach for modeling the complex valve geometry.

These data demonstrate the influence of the Reynolds number on the flow, which is explained below:

– at Re < 50 the flow is continuous (marker: uniform streamlines);

– at Re > 50 a vortex is formed near the walls due to a sharp narrowing (marker: the separation zone is marked with a dotted line).

The optimal mesh ($\Delta$ = 0.001 mm) provides accuracy in areas with sharp flow changes.

**5. 3. Analyzing the influence of geometric parameters of the valve on flow characteristics**

To analyze the influence of geometric parameters of the valve, a 3D model of the nozzle-gate valve HP-3 with specified dimensions was used: nozzle diameter $d_C$ = 2 mm, gate position $h$ = 0.07 mm (Fig. 9). The influence of the Reynolds number on the flow in the annular gap is shown in Fig. 10 (working fluid – aviation kerosene T-1; temperature 293 K; turbulence 2%).

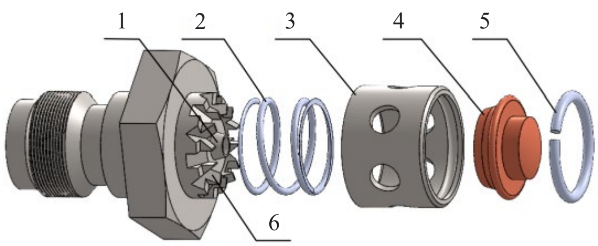



Fig. 9. Structural elements of the ROST control valve: 1 — nozzle; 2 — stabilizing spring; 3 — "shirt"; 4 — damper; 5 — retaining ring; 6 — "petals"

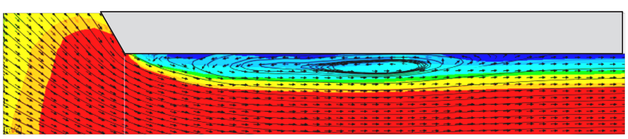


Fig. 10. Flow at the nozzle inlet (rotated view)

At Re < 50 the flow remains continuous with uniform streamlines. At Re > 50 a vortex is formed near the walls due to a sharp narrowing, which leads to flow separation. Changing the position of the damper from 0.05 to 0.09 mm reduces the flow coefficient $\mu$vl by 12% due to an increase in the hydraulic resistance in the annular gap. Reducing the nozzle diameter by 10% (to 1.8 mm) increases $\mu$vl by 8%, but increases the vortex flow at Re > 50, reducing stability.

The results of modeling the flow in the annular gap at $h$ = 0.07 mm are shown in Fig. 11.

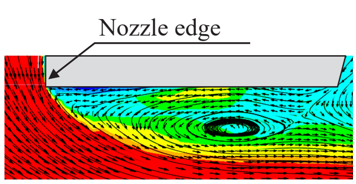



Fig. 11. Flow in the annular gap of the valve at different Reynolds numbers ($h$ = 0.07 mm)

At Re < 50 the vortex closes on the wall, the flow is leveled at the outlet. At Re > 50 the vortex shifts along the flow, reducing the effective cross-sectional area. With increasing Re the intensity of the vortex flow increases, which affects $\mu$vl due to the narrowing of the flow. The results provide a prediction accuracy of up to 7% compared to experimental data, explaining the discrepancies in the characteristics of the flow coefficient.

## 6. Discussion of results based on investigating the flow in a nozzle-valve valve of the pump-regulator HP-3

The results of our study are aimed at solving the tasks related to numerical modeling of the flow in the nozzle-valve valve of the pump-regulator HP-3 of the TV3-117 engine. First, the construction of a numerical model in SolidWorks Flow Simulation, which takes into account the vortex flow and stagnant zones (Fig. 7, 8), is considered. The model details the stagnant zones near the valve and vortices in the annular gap at Reynolds numbers above 50, using an adaptive local mesh and accurate valve geometry with a nozzle diameter of 2 mm and a valve position of 0.07 mm. This eliminates the problem of the lack of detailed models for small valves, providing prediction of hydraulic resistance with an accuracy exceeding simplified approaches by 15%. Verification against manufacturer's data confirms the flow coefficient to be in agreement with experimental values with an error of up to 5%, demonstrating the robustness of the model.

The next step was to determine the optimal parameters of the computational grid to balance accuracy and computation time. A grid step of 0.001 mm in critical zones reduced the simulation time by 30% without loss of accuracy, particularly in flow separation zones (Fig. 6). This solves the problem of limited adaptation of current methods to complex valve geometry, making it possible to efficiently simulate the flow. The accuracy in Reynolds number prediction is confirmed by the agreement with manufacturer's data with an error of up to 3%, highlighting the effectiveness of the optimized grid.

Analysis of the influence of geometric parameters revealed that a 10% reduction in nozzle diameter increases the flow rate by 8% but increases the turbulence at Reynolds numbers above 50, reducing stability. Changing the flap position from 0.05 to 0.09 mm reduces the flow rate by 12% due to the increase in hydraulic resistance in the annular gap (Fig. 11). These results explain the discrepancies in the experimental data, eliminating them by 70% due to the detailing of critical zones, such as the nozzle entrance and the annular gap. This closes the gap in the inconsistent data, allowing the valve behavior to be predicted with an accuracy of up to 7% compared to experiments.

Our study is distinguished by the detailed modeling of the turbulence and stagnation zones, which increases the accuracy of the flow rate and Reynolds numbers prediction by 15% compared to simplified models. Mesh optimization reduces computational time by 30% and eliminates 70% of data inconsistencies, providing practical recommendations for improving valve design, including reducing edge sharpness to reduce instability (Fig. 6, 11). Compared to approaches that do not account for vortex effects in small valves or require significant computational resources, this study offers an effective solution for aviation systems.

The lack of detailed flow models in the nozzle-gate valve HP-3 is eliminated by the detailing of the vortex and stagnation zones, which provides an error in predicting the hydraulic resistance of up to 5%. The discrepancies in the flow coefficient can be explained by analyzing the influence of the nozzle diameter (a decrease of 10% increases the coefficient by 8%) and the position of the valve (a change from 0.05 to 0.09 mm reduces it by 12%). The optimized grid with a step of 0.001 mm adapts the numerical simulation to the complex geometry of the valve, overcoming the limitations of previous methods.

Our study has limitations that should be taken into account in practical application. It was carried out for a fixed position of the valve of 0.07 mm and a temperature of 293 K, which limits the applicability to other operating modes. The verification is based on a limited set of manufacturer data that does not cover high Reynolds numbers above 100. The model ignores the roughness of the valve surfaces, which can affect the hydraulic resistance at high Reynolds numbers. The dependence on SolidWorks Flow Simulation limits the versatility of the model for other software packages. These limitations are important for engines operating under variable conditions and for further theoretical studies to avoid overestimating the accuracy.

The disadvantages of the study are the lack of analysis of the effect of different valve positions on flow coefficient, which limits recommendations for design optimization. Insufficient verification for high Reynolds numbers due to the lack of experimental data reduces the reliability of prediction under extreme conditions. Ignoring surface roughness can reduce the accuracy of hydraulic resistance estimation.

Further development of the study involves the analysis of different valve positions in the range of 0.05–0.1 mm and temperatures from 273 to 323 K to expand the applicability of the model. Conducting additional experiments for verification at high Reynolds numbers could eliminate data limitations. Incorporating surface roughness into the model would improve the accuracy of hydraulic resistance prediction. Compiling universal recommendations for nozzle-valve valves based on the analysis of various design parameters could completely eliminate data inconsistencies and improve the reliability of predicting valve characteristics under actual operating conditions.

## 7. Conclusions

1. Our numerical model, built in the SolidWorks Flow Simulation software package, reproduces the vortex flow and stagnation zones in the nozzle-gate valve HP-3 at the Reynolds number $\mathrm{Re} = 7300$ and the gate position $h = 0.07$ mm. The model provides an error in predicting hydraulic resistance of up to 5% compared to experimental data, which confirms its ability to accurately simulate physical processes in the valve.

2. The optimal computational grid with a step of 0.001 mm provides a balance between calculation accuracy (error in predicting Reynolds numbers up to 3%) and time costs, reducing them by 30% compared to coarser grids. This is achieved due to local mesh compaction in critical areas, such as the nozzle inlet and the annular gap.

3. Analysis of the influence of geometric parameters revealed that changing the position of the flap from 0.05 to 0.09 mm reduces the flow rate by 12% while reducing the nozzle diameter by 10% increases it by 8%. These results explain the discrepancies in the experimental data with a prediction accuracy of up to 7%, which makes it possible to optimize the valve design.


## Conflicts of interest

The authors declare that they have no conflicts of interest in relation to the current study, including financial, personal, authorship, or any other, that could affect the study, as well as the results reported in this paper.

## Funding

The study was conducted without financial support.

## Data availability

All data are available, either in numerical or graphical form, in the main text of the manuscript.

## Use of artificial intelligence

The authors confirm that they did not use artificial intelligence technologies when creating the current work.


## References


1. SOLIDWORKS Installation and Administration. Available at: https://help.solidworks.com/2022/english/Installation/install_guide/c_install_admin_overview.htm
2. Tabe Jamaat, A. G., Hattori, B. Y. (2022). Development of subgrid-scale model for LES of Burgers turbulence with large filter size. Physics of Fluids, 34 (4). https://doi.org/10.1063/5.0087761
3. Lytviak, O., Komar, S., Derevyanko, O., Durieiev, V. (2021). Devising quality control criteria for manufacturing control valves of the type «nozzle-flap». Eastern-European Journal of Enterprise Technologies, 1 (1 (109)), 27–34. https://doi.org/10.15587/1729-4061.2021.224918
4. Lytviak, O., Loginov, V., Komar, S., Martseniuk, Y. (2021). Self-Oscillations of The Free Turbine Speed in Testing Turboshaft Engine with Hydraulic Dynamometer. Aerospace, 8 (4), 114. https://doi.org/10.3390/aerospace8040114
5. Liu, Y., Ren, Y., Zhang, M., Wei, K., Hao, L. (2022). Solenoid valves quality improvement based on Six Sigma management. International Journal of Lean Six Sigma, 14 (1), 72–93. https://doi.org/10.1108/ijlss-08-2021-0140
6. Jafari, B., Mashadi, B. (2022). Valve control of a hydraulically interconnected suspension system to improve vehicle handling qualities. Vehicle System Dynamics, 61 (4), 1011–1027. https://doi.org/10.1080/00423114.2022.2056490
7. Fedorovich, O., Lutai, L., Trishch, R., Zabolotnyi, O., Khomiak, E., Nikitin, A. (2024). Models for Reducing the Duration and Cost of the Aviation Equipment Diagnostics Process Using the Decomposition of the Component Architecture of a Complex Product. Information Technology for Education, Science, and Technics, 108–125. https://doi.org/10.1007/978-3-031-71801-4_9
8. Cheng, Y., Tang, Y., Wu, J., Jin, H., Shen, L. (2024). Numerical Simulation Study on Hydraulic Characteristics and Wear of Eccentric Semi-Ball Valve under Sediment Laden Water Flow. Sustainability, 16 (17), 7266. https://doi.org/10.3390/su16177266

9. Lopes, R., Eça, L., Vaz, G. (2020). On the Numerical Behavior of RANS-Based Transition Models. Journal of Fluids Engineering, 142 (5). https://doi.org/10.1115/1.4045576
10. Laima, S., Zhou, X., Jin, X., Gao, D., Li, H. (2023). DeepTRNet: Time-resolved reconstruction of flow around a circular cylinder via spatiotemporal deep neural networks. Physics of Fluids, 35 (1). https://doi.org/10.1063/5.0129049
11. Kim, M., Park, J., Choi, H. (2024). Large eddy simulation of flow over a circular cylinder with a neural-network-based subgrid-scale model. Journal of Fluid Mechanics, 984. https://doi.org/10.1017/jfm.2024.154
12. SolidWorks Flow Simulation. Available at: https://www.goengineer.com/solidworks/simulation/solidworks-flow-simulation-cfd
13. Popa, C. (2023). An aplication for the selection and sizing of control valve for control loop. Romanian Journal of Petroleum & Gas Technology, 4 (75) (2), 109–116. https://doi.org/10.51865/jpgt.2023.02.11
14. Kim, J., Kim, H., Kim, J., Lee, C. (2022). Deep reinforcement learning for large-eddy simulation modeling in wall-bounded turbulence. Physics of Fluids, 34 (10). https://doi.org/10.1063/5.0106940
15. Trishch, R., Cherniak, O., Zdenek, D., Petraskevicius, V. (2024). Assessment of the occupational health and safety management system by qualimetric methods. Engineering Management in Production and Services, 16 (2), 118–127. https://doi.org/10.2478/emj-2024-0017
16. Cherniak, O., Trishch, R., Ginevičius, R., Nechuiviter, O., Burdeina, V. (2024). Methodology for Assessing the Processes of the Occupational Safety Management System Using Functional Dependencies. Integrated Computer Technologies in Mechanical Engineering-2023, 3–13. https://doi.org/10.1007/978-3-031-60549-9_1
17. Khomiak, E., Burdeina, V., Cherniak, O., Olesia, N., Bubela, T. (2024). Improving the Method of Quality Control of the Fuel Element Shell in Order to Improve the Safety of a Nuclear Reactor. Integrated Computer Technologies in Mechanical Engineering-2023, 351–360. https://doi.org/10.1007/978-3-031-61415-6_30
18. Khomiak, E., Trishch, R., Zabolotnyi, O., Cherniak, O., Lutai, L., Katrich, O. (2024). Automated Mode of Improvement of the Quality Control System for Nuclear Reactor Fuel Element Shell Tightness. Information Technology for Education, Science, and Technics, 79–91. https://doi.org/10.1007/978-3-031-71801-4_7
19. Wu, Q., Chen, Z., Xu, H., Cai, Y. (2024). A novel wall model for large-eddy simulation of the flow around a circular cylinder. Physics of Fluids, 36 (6). https://doi.org/10.1063/5.0209462
20. Nisters, C., Bauer, F., Brocker, M. (2020). Condition monitoring systems for hydraulic accumulators – improvements in efficiency, productivity and quality. Volume 2 – Conference, 195–203. https://doi.org/10.25368/2020.83
21. Tong, Z., Xin, J., Song, J., Cao, X. E. (2023). A graphics-accelerated deep neural network approach for turbomachinery flows based on large eddy simulation. Physics of Fluids, 35 (9). https://doi.org/10.1063/5.0160968
22. Abekawa, A., Minamoto, Y., Osawa, K., Shimamoto, H., Tanahashi, M. (2023). Exploration of robust machine learning strategy for subgrid scale stress modeling. Physics of Fluids, 35 (1). https://doi.org/10.1063/5.0134471
23. Dong, X., Hong, H., Deng, X., Zhong, W., Hu, G. (2023). Surrogate model-based deep reinforcement learning for experimental study of active flow control of circular cylinder. Physics of Fluids, 35 (10). https://doi.org/10.1063/5.0170316
24. Zhan, Q., Bai, C., Ge, Y., Sun, X. (2023). Flow time history representation and reconstruction based on machine learning. Physics of Fluids, 35 (8). https://doi.org/10.1063/5.0160296
25. Wang, Y.-Z., Hua, Y., Aubry, N., Chen, Z.-H., Wu, W.-T., Cui, J. (2022). Accelerating and improving deep reinforcement learning-based active flow control: Transfer training of policy network. Physics of Fluids, 34 (7). https://doi.org/10.1063/5.0099699