UDC 621.224 **doi: 10.20998/2411-3441.2023.1.08**

Y. KRUPA, R. DEMCHUK, A. VOLOBUIEV, S. KIS

COMPARATIVE ANALYSIS OF SOFTWARE SYSTEMS FOR HYDRAULIC TURBINE FLOW SIMULATION

A comprehensive review of modern software complexes used for calculating spatial flow in hydraulic turbine flow parts was conducted. The widely used software system, Ansys, was analyzed. An overview of Ansys was provided, including its history, popularity within the CFD community, key features, and capabilities for analyzing the flow parts of hydraulic turbines. The preprocessing tools, solver parameters, post-processing functions, and visualization capabilities of Ansys were described. The advantages and limitations of using Ansys for calculating spatial flow in hydraulic turbine flow parts were analyzed. The open-source CFD software complex, OpenFOAM, was discussed. The main functions and capabilities of the OpenFOAM program were described. Information about solver libraries, meshing capabilities, advantages, and limitations for analyzing hydraulic turbines was presented, along with insights into the support from the scientific community and resources available to OpenFOAM users. SolidWorks FlowSimulation, which integrates with SolidWorks software, was examined. The unique features of SolidWorks FlowSimulation for analyzing spatial flow in hydraulic turbines were highlighted. The possibilities of CAD integration and the advantages of accurate geometric models were discussed. The capabilities of parametric analysis were explored, and the advantages and limitations of using SolidWorks FlowSimulation for calculating spatial flow in hydraulic turbine flow parts were analyzed. A comparison of the three software complexes was conducted based on their capabilities, ease of use, accuracy, computational resources required, and cost. An assessment of the advantages and disadvantages of each program was provided, along with recommendations for choosing the most suitable program based on specific use cases, objectives, and user requirements. This article serves as a valuable resource for engineers, designers, and researchers seeking insights into the available software systems for analyzing hydraulic turbine flow parts. It enables them to make informed decisions in selecting the most suitable software system based on their specific requirements, ultimately contributing to the optimization of hydraulic turbine performance and efficiency.

Keywords: spatial flow, hydraulic turbine, numerical research, computational fluid dynamics, CFD software system, Ansys, OpenFOAM.

Є. С. КРУПА, Р. М. ДЕМЧУК, А. В. ВОЛОБУЄВ, С. Л. КІСЬ

ПОРІВНЯЛЬНИЙ АНАЛІЗ ПРОГРАМНИХ КОМПЛЕКСІВ ДЛЯ МОДЕЛЮВАННЯ ПОТОКУ В ГІДРОТУРБІНАХ

Проведено комплексний огляд сучасних програмних комплексів, що використовуються для розрахунку просторової течії в проточних частинах гідротурбін. Проаналізовано широко використовувану систему програмного забезпечення Ansys. Виконано огляд даної програми, виділено ключові характеристики та можливості для аналізу проточних частин гідравлічних турбін. Описано інструменти попередньої обробки, параметри розв'язувача, функції постобробки та можливості візуалізації Ansys. Проаналізовано переваги і недоліки використання Ansys для розрахунку просторового потоку в проточних частинах гідравлічних турбін. Розглянуто програмний комплекс CFD з відкритим програмним кодом OpenFOAM. Описано основні функції та можливості програми OpenFOAM. Представлено інформацію про бібліотеки розв'язувачів, можливості сітки, переваги та обмеження для аналізу гідравлічних турбін. Крім того, висвітлюється підтримка наукової спільноти та ресурси, доступні для користувачів OpenFOAM. Розглянуто SolidWorks FlowSimulation, що інтегрується з програмним забезпеченням SolidWorks. Висвітлено унікальні функції SolidWorks FlowSimulation для аналізу просторового потоку в гідравлічних турбінах. Обговорено можливості інтеграції САПР і переваги точних геометричних моделей. Розглянуто можливості параметричного аналізу та проаналізовано переваги й обмеження використання SolidWorks FlowSimulation для розрахунку просторового потоку в проточних частинах гідравлічних турбін. Виконано порівняння трьох програмних комплексів на основі їх можливостей, простоти використання, точності, необхідних обчислювальних ресурсів і вартості. Проведено оцінку переваг та недоліків кожної програми та надано рекомендації щодо вибору найбільш доцільної програми на основі конкретних випадків використання, цілей і вимог користувача.

Ключові слова: просторовий потік, гідравлічна турбіна, чисельне дослідження, обчислювальна гідродинаміка, програмний комплекс CFD, Ansys, OpenFOAM.

Introduction. The efficient design and operation of hydraulic turbines are essential for the generation of hydroelectric power. Central to achieving optimal performance is the accurate calculation and analysis of the spatial flow within the flow parts of hydraulic turbines. The study of flow behavior provides insights into various parameters such as velocity distribution, pressure gradients, turbulence, and energy losses, which are crucial for understanding the performance characteristics and efficiency of hydraulic turbines.

Computational fluid dynamics (CFD) software systems have revolutionized the field of hydraulic turbine analysis by offering powerful tools for simulating and analyzing flow behavior. These software systems enable engineers and researchers to numerically model and predict the flow phenomena within the complex flow parts of hydraulic turbines, providing valuable insights for design optimization and performance enhancement [1–11].

The objective of this article is to present a comprehensive review and comparison of three widely employed CFD software systems – Ansys, OpenFOAM, and SolidWorks FlowSimulation – for the calculation of spatial flow in hydraulic turbine flow parts. Through a thorough examination of these software systems, an assessment of their capabilities, advantages, and limitations is intended to be provided, enabling engineers and researchers to make well-informed decisions in selecting suitable software for their specific analysis requirements.

To achieve this objective, this article examines the features, methodologies, and applications of each software system, highlighting their distinct strengths and weaknesses. The comparison encompasses various factors, including accuracy, computational efficiency, user-friendliness, pre-processing and post-processing capabilities, availability of specialized modules, and integration with other engineering tools.

By gaining a comprehensive understanding of the capabilities and limitations of these software systems, engineers and scientists can effectively choose the most appropriate CFD software for their hydraulic turbine analysis needs, thereby enhancing design optimization, performance prediction, and energy efficiency.

Ansys: A Comprehensive CFD Software System. Ansys is a widely recognized and extensively used commercial software system in the field of computational fluid dynamics. With a rich history and a strong presence in the CFD community, Ansys has become a popular choice for engineers and researchers involved in hydraulic turbine analysis [1–3].

Overview of Ansys. Software Ansys is a comprehensive suite of software tools that encompasses various engineering disciplines, including CFD. It provides a robust platform for simulating and analyzing fluid flow phenomena within complex geometries. With a long-standing reputation in the industry, Ansys has gained the trust of engineers worldwide for its accuracy and versatility [12, 13].

Key Features and Capabilities. Ansys offers a wide range of features and capabilities specifically designed for hydraulic turbine analysis. These features enable engineers to accurately model and analyze the spatial flow within turbine flow parts. Key capabilities of Ansys for analyzing hydraulic turbine flow parts include:

a) Geometry Creation and Mesh Generation: Ansys provides powerful pre-processing tools that facilitate the creation of complex turbine geometries. It offers options for parametric modeling, allowing for easy manipulation of geometric parameters. Additionally, Ansys provides efficient meshing tools that generate high-quality meshes suitable for resolving complex flow behavior in hydraulic turbine flow parts [12–14]. Fig. 1 shows an example of the computational mesh of the flow part of a hydraulic turbine.

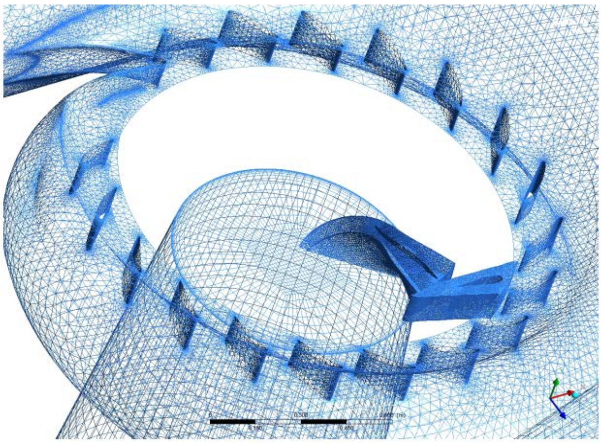

Fig. 1. The computational mesh made in Ansys

b) Solver Options: Ansys offers a variety of solver options tailored for hydraulic turbine simulations. It includes both steady-state and transient solvers that can accurately capture time-dependent flow phenomena. The solvers employ advanced numerical techniques, such as finite volume or finite element methods, to solve the governing fluid flow equations.

c) Turbulence Modeling: Ansys incorporates a range of turbulence models, including the popular Reynolds-averaged Navier-Stokes (RANS) models, to accurately predict turbulent flow behavior within hydraulic turbine flow parts. These models account for turbulence phenomena, such as boundary layer separation, vortices, and secondary flow patterns.

d) Multiphase Flow Analysis: Ansys also provides capabilities for analyzing multiphase flows, allowing engineers to study scenarios involving cavitation, air entrainment, and free surface flows within hydraulic turbines. This enables a more comprehensive understanding of flow behavior and its impact on turbine performance [14].

Pre-processing Tools. Ansys offers a user-friendly pre-processing environment that facilitates geometry creation and mesh generation for hydraulic turbine simulations. The software supports CAD import and provides tools for geometry modification and manipulation. Ansys also offers reliable meshing tools that can generate high-quality meshes with appropriate grid resolution for resolving flow features in turbine flow parts.

Solver Options. Ansys provides a range of solver options suitable for hydraulic turbine simulations. It offers steady-state solvers, such as the pressure-based solver, which are commonly used for analyzing turbine flow parts under steady operating conditions. These solvers ensure accurate and efficient convergence, providing insights into flow velocity, pressure distribution, and other flow parameters [12, 13].

Post-processing Features and Visualization Capabilities. Ansys provides comprehensive post-processing features that enable engineers to analyze and interpret the results of hydraulic turbine simulations. It offers a range of visualization tools to represent flow fields, pressure distributions, velocity vectors, and other parameters of interest. These visualization options aid in the identification of flow patterns, areas of high turbulence, and regions of potential energy losses within the turbine flow parts. Fig. 2 shows an example of visualization of the velocity distribution in the runner of a hydraulic turbine.

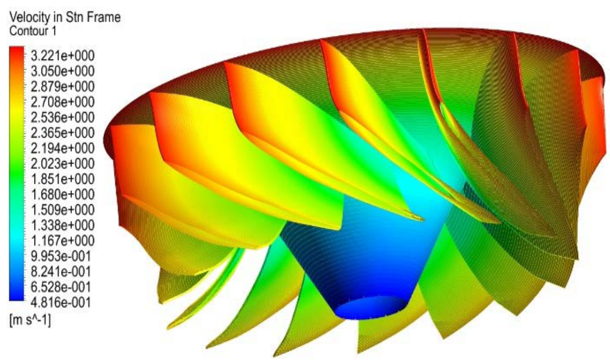

Fig. 2. The velocity distribution in the runner of a hydraulic turbine

Furthermore, Ansys offers advanced post-processing capabilities, such as streamline visualization, contour plots, and cutting planes, which facilitate in-depth analysis and detailed examination of the flow behavior. Engineers can extract quantitative data, generate reports, and compare different simulation cases using the post-processing tools provided by Ansys [12–14].

Advantages and Disadvantages of Using Ansys. Ansys offers several advantages for calculating spatial flow in hydraulic turbine flow parts:

• Wide range of features and capabilities specifically designed for hydraulic turbine analysis.

• Efficient pre-processing tools for reliable geometry creation and mesh generation.

• Comprehensive solver options for steady-state and transient simulations.

• Support for advanced turbulence modeling and multiphase flow analysis.

• User-friendly post-processing features and visualization capabilities for result interpretation.

However, there are also some limitations and considerations when using Ansys:

• Complexity: Ansys is a powerful software system that requires a learning curve to effectively utilize its capabilities.

• Computational Resources: Accurate simulation of hydraulic turbine flow parts may require significant computational resources, such as memory and processing power.

• Cost: Ansys is a commercial software system and may involve licensing fees, which can be a factor for small-scale projects or academic research.

OpenFOAM: An Open-Source CFD Software System. OpenFOAM is a widely used open-source software system in the field of computational fluid dynamics. Its open-source nature has made it popular among researchers and developers in the CFD community [15–17].

Introduction to OpenFOAM. OpenFOAM, short for Open Field Operation and Manipulation, is an open-source CFD software system that provides a flexible and extensible framework for simulating fluid flow phenomena. Being an open-source software, OpenFOAM allows users to access and modify its source code, providing the freedom to customize and adapt the software to their specific needs. This open nature has fostered a vibrant community of developers and users who contribute to its continuous development and improvement [15–17].

OpenFOAM has gained significant popularity in the CFD community due to its versatility, scalability, and robustness. It offers a wide range of solvers, models, and utilities that make it suitable for various fluid flow simulations, including hydraulic turbine analysis.

Core Features and Solver Libraries. OpenFOAM provides a comprehensive set of core features and solver libraries that are relevant to hydraulic turbine analysis. These features include:

a) Solver Libraries: OpenFOAM offers a variety of solver libraries, including the Finite Volume Method (FVM) solvers, which are widely used for simulating fluid flow in hydraulic turbines. These solvers solve the governing equations of fluid flow, such as the Navier-Stokes equations, using the finite volume discretization approach. They allow for accurate and efficient computation of flow behavior within hydraulic turbine flow parts [15–17].

b) Turbulence Models: OpenFOAM incorporates several turbulence models, ranging from Reynolds-averaged Navier-Stokes (RANS) models to more advanced Large Eddy Simulation (LES) models. These models capture the effects of turbulence on flow behavior and are relevant for analyzing the turbulent characteristics within hydraulic turbines.

c) Multiphase Flow Analysis: OpenFOAM offers capabilities for simulating multiphase flows, which are essential for studying phenomena such as cavitation, air-water interaction, and free surface flows within hydraulic turbines [15–17].

Meshing Capabilities. OpenFOAM provides flexible meshing capabilities that are crucial for accurately resolving the flow behavior within hydraulic turbine flow parts. It supports various meshing techniques, including structured, unstructured, and hybrid meshes. OpenFOAM's meshing utilities allow for the generation of high-quality meshes with appropriate resolution near the boundaries and critical flow regions (Fig. 3).

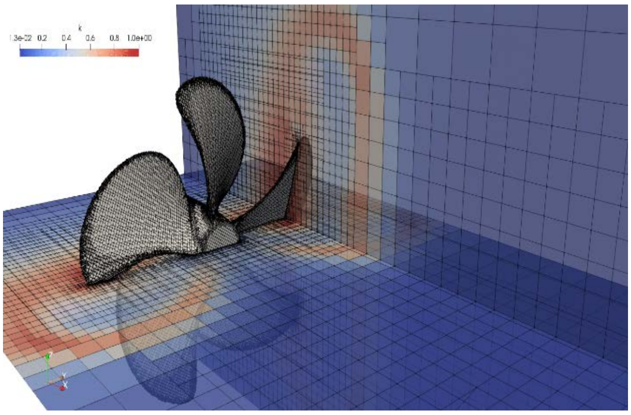

Fig. 3. The computational mesh made in OpenFOAM

Additionally, OpenFOAM supports mesh adaptation, which enables the refinement of the mesh in regions of interest. This feature is particularly useful for capturing flow features and phenomena that require higher resolution, such as boundary layer separation, secondary flows, and flow mixing.

Advantages and Limitations of Using OpenFOAM. Using OpenFOAM for calculating spatial flow in hydraulic turbine flow parts offers several advantages:

• Open-Source Nature: OpenFOAM's open-source nature allows for customization and adaptation to specific analysis needs, providing users with flexibility and control over the software.

• Versatility: OpenFOAM's comprehensive features and solver libraries make it suitable for a wide range of fluid flow simulations, including hydraulic turbine analysis.

• A wide range of options for visualizing the results of a numerical flow study [15–17]. Fig. 4 shows an example of streamlines and velocity magnitude distribution in a Francis turbine.

• Scalability: OpenFOAM is capable of handling complex flow phenomena and can be efficiently scaled for large-scale simulations.

• Community Support: OpenFOAM benefits from an active and supportive user community, providing access to resources, forums, and shared knowledge.

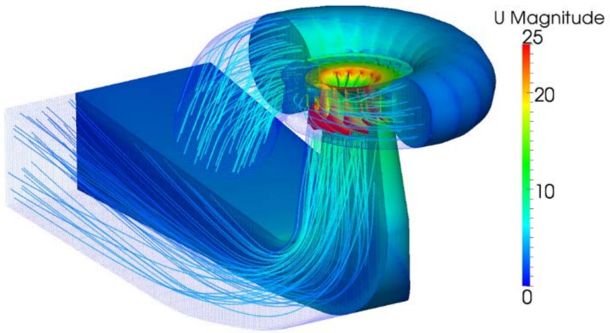

Fig. 4. Francis turbine, complete model view with streamlines and velocity magnitude distribution

However, there are also limitations and considerations when using OpenFOAM:

• Learning Curve: OpenFOAM has a steeper learning curve compared to commercial software systems, requiring familiarity with CFD principles and some programming knowledge.

• Resource Requirements: Accurate simulations of hydraulic turbine flow parts in OpenFOAM may require substantial computational resources, including memory and processing power.

• Limited Commercial Support: OpenFOAM being an open-source software, limited commercial support is available compared to commercial software systems.

Community Support and Resources. One of the strengths of OpenFOAM is its active and vibrant community, which provides extensive support and resources for users. The community offers online forums, mailing lists, and user groups where users can seek assistance, share knowledge, and collaborate on research and development projects. Moreover, numerous online tutorials, documentation, and case studies are available to help users navigate and utilize OpenFOAM effectively for hydraulic turbine analysis.

SolidWorks FlowSimulation: Integration with CAD for Hydraulic Turbine Analysis. SolidWorks FlowSimulation is a powerful computational fluid dynamics software system that seamlessly integrates with SolidWorks CAD software [1, 2, 18].

Introduction to SolidWorks FlowSimulation. SolidWorks FlowSimulation is a CFD software system that enables engineers and designers to simulate fluid flow and heat transfer phenomena within SolidWorks CAD models. The smooth integration between SolidWorks CAD and FlowSimulation allows for efficient analysis and optimization of hydraulic turbine flow parts. This integration eliminates the need for data translation or model reconstruction, ensuring accuracy and consistency throughout the design and analysis process [1, 2, 18].

Unique Features and Functionalities SolidWorks. FlowSimulation offers several unique features and functionalities specifically customized for hydraulic turbine analysis:

a) Intelligent Solution Control: FlowSimulation incorporates an intelligent solution control algorithm that automatically adjusts the solution settings based on the complexity of the flow problem. This feature ensures accurate and efficient simulations, even for complex hydraulic turbine flow parts [19].

b) Thermal Analysis: FlowSimulation provides capabilities for thermal analysis, allowing engineers to study the temperature distribution within hydraulic turbine flow parts. This is particularly relevant for analyzing heat transfer phenomena and ensuring proper cooling and thermal management of the turbine components.

c) Parametric Analysis: FlowSimulation offers parametric analysis capabilities that enable engineers to study the impact of design variations on the performance of hydraulic turbine flow parts. By defining parameters such as geometry dimensions, boundary conditions, or operating conditions, engineers can optimize the design for maximum efficiency and performance [20, 21].

CAD Integration and Benefits. The integration between SolidWorks CAD and FlowSimulation provides numerous benefits for hydraulic turbine analysis:

a) Accurate Geometry Models: FlowSimulation directly utilizes the SolidWorks CAD geometry, ensuring accuracy and consistency between the design and analysis stages. Any changes made to the CAD model automatically propagate to the analysis setup, eliminating the need for manual data transfer or reconstruction.

Fig. 5 shows the computational mesh of the draft tube of a hydraulic turbine, made in FlowSimulation.

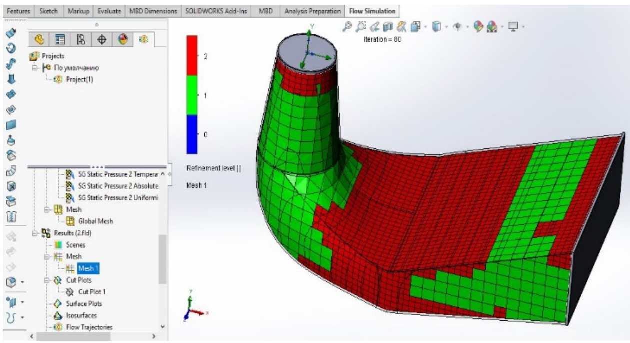

Fig. 5. The computational mesh made in FlowSimulation

b) Smooth Workflow: The integration enables a smooth workflow, facilitating efficient setup and analysis of fluid flow within hydraulic turbine flow parts. The familiar SolidWorks interface and tools enhance productivity and reduce the learning curve for users. Fig. 6 shows the visualization of the velocity distribution in the draft tube of a hydraulic turbine in the form of a fill and vectors.

c) Design Optimization: The CAD integration capabilities facilitate design optimization by enabling engineers to quickly modify the geometry and analyze different design scenarios. This iterative process can lead to improved performance, efficiency, and reliability of hydraulic turbine flow parts.

Parametric Analysis Capabilities. FlowSimulation offers advanced parametric analysis capabilities that are particularly relevant to optimizing hydraulic turbine flow parts [18–21]:

a) Design Exploration: Engineers can define design parameters, such as blade profiles, impeller geometries, or inlet configurations, and perform systematic variations to understand their impact on flow behavior. This allows for a comprehensive exploration of the design space and identification of optimal solutions.

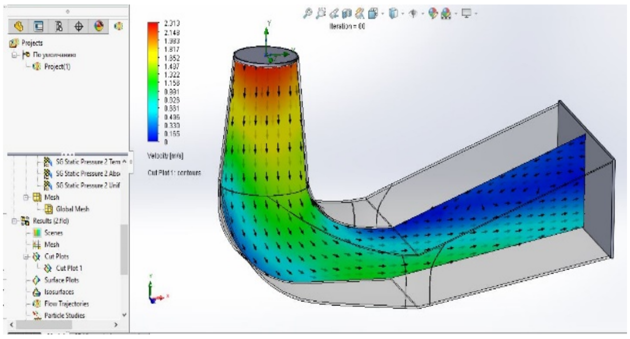

Fig. 6. The visualization of the velocity distribution in the draft tube of a hydraulic turbine

b) Sensitivity Analysis: Parametric analysis in FlowSimulation enables engineers to evaluate the sensitivity of key flow parameters, such as pressure drop, flow rate, or efficiency, to variations in design or operating conditions. This information can aid in identifying critical design factors and guiding design decisions.

c) Optimization Tools: FlowSimulation provides optimization tools that allow for the automated search of design parameters to achieve specific performance goals. By defining design constraints and objectives, engineers can utilize these tools to automatically identify the optimal design configuration for hydraulic turbine flow parts.

Advantages and Limitations of Using SolidWorks FlowSimulation. Using SolidWorks FlowSimulation for calculating spatial flow in hydraulic turbine flow parts offers several advantages:

• Integration with SolidWorks CAD: The seamless integration allows for accurate geometry models and efficient design and analysis workflows.

• Parametric Analysis: The parametric analysis capabilities enable engineers to optimize the design for maximum performance and efficiency.

• Thermal Analysis: The thermal analysis capabilities aid in studying heat transfer phenomena and ensuring proper cooling of the turbine components.

However, there are also limitations and considerations when using SolidWorks FlowSimulation:

• Limited Solver Options: SolidWorks FlowSimulation offers a limited selection of solver options compared to specialized CFD software systems, which may limit the range of complex flow phenomena that can be accurately simulated.

• Simplified Physics Modeling: While FlowSimulation provides a user-friendly interface, it may have simplified physics modeling options compared to more advanced CFD software systems.

• Integration Dependency: The seamless integration with SolidWorks CAD means that users need to have access to both software systems, which may involve additional costs for certain applications.

Comparison of Software Systems and Recommendations. In order to conduct a comparative analysis of Ansys, OpenFOAM, and SolidWorks FlowSimulation, the following factors will be evaluated:

a) Capabilities: Ansys, OpenFOAM, and SolidWorks FlowSimulation offer a range of capabilities for hydraulic turbine analysis. Ansys provides a comprehensive set of solvers and models, including turbulence models and multiphase flow analysis. OpenFOAM, being an open-source software system, offers flexibility and a wide range of solver libraries. SolidWorks FlowSimulation integrates with SolidWorks CAD and provides unique features such as thermal analysis and parametric analysis capabilities.

b) Ease of Use: Ansys and SolidWorks FlowSimulation are known for their user-friendly interfaces and intuitive workflows, making them more accessible to engineers and designers with varying levels of CFD expertise. OpenFOAM, on the other hand, has a steeper learning curve and requires a good understanding of CFD principles and some programming knowledge.

c) Accuracy: All three software systems can provide accurate results when properly configured and validated against experimental data or other benchmark solutions. Ansys and OpenFOAM offer more flexibility in terms of solver options and customization, allowing for finer control over the simulation accuracy.

d) Computational Resources Required: The computational resources required by each software system can vary depending on the complexity of the flow problem and the desired level of accuracy. Ansys and OpenFOAM provide options for parallel computing, enabling faster simulations for large-scale problems. SolidWorks FlowSimulation is generally less computationally demanding, making it suitable for less complex hydraulic turbine analyses.

e) Cost: Ansys is a commercial software system and typically requires a license, which can be costly depending on the desired functionality and usage. OpenFOAM, being an open-source software, is free to use, but may require additional resources for setup, customization, and support. SolidWorks FlowSimulation is a commercial software, and its cost may vary depending on the licensing options and modules required.

Strengths and Weaknesses. Ansys, OpenFOAM, and SolidWorks FlowSimulation have their respective strengths and weaknesses for hydraulic turbine analysis:

• Ansys: Ansys provides a comprehensive range of solvers, models, and pre- and post-processing tools. It offers a user-friendly interface, extensive documentation, and commercial support. However, its cost and the need for computational resources may be limiting factors for some users [12–14].

• OpenFOAM: OpenFOAM's open-source nature and flexibility make it attractive for researchers and developers. It offers a wide range of solver libraries and customization options. However, it has a steeper learning curve, limited commercial support, and may require more computational resources for complex simulations [15–17].

• SolidWorks FlowSimulation: SolidWorks FlowSimulation's smooth integration with SolidWorks

CAD streamlines the design and analysis process. It provides user-friendly interfaces, accurate results, and efficient parametric analysis capabilities. However, its solver options and customization capabilities may be more limited compared to specialized CFD software systems [18–21].

Recommendations. Selecting the most suitable software system for hydraulic turbine analysis depends on specific use cases, objectives, and user requirements. Consider the following recommendations:

a) For engineers and designers seeking a user-friendly interface, efficient workflow, and comprehensive capabilities, Ansys or SolidWorks FlowSimulation can be excellent choices. Ansys provides a wider range of solvers and models, while SolidWorks FlowSimulation offers seamless CAD integration.

b) Researchers and developers with a strong background in CFD and a need for flexibility, customization, and parallel computing capabilities may find OpenFOAM to be a suitable option. It allows for extensive solver customization and provides a platform for code development and research.

c) Cost considerations play a significant role. Ansys may require a higher investment due to licensing fees, while OpenFOAM offers a cost-effective option as an open-source software system. SolidWorks FlowSimulation's cost will depend on the licensing options and modules required.

d) Consider the complexity of the flow problem, the geometry of the hydraulic turbine, and the desired level of accuracy. Ansys and OpenFOAM are well-suited for complex flow phenomena, while SolidWorks FlowSimulation is more suitable for less complex analyses.

e) Evaluate the availability of resources, including user support, documentation, and online communities. Ansys and SolidWorks FlowSimulation benefit from extensive commercial support and resources, while OpenFOAM relies on an active user community for support.

Conclusions. In this article, the use of modern software systems for calculating the spatial flow in hydraulic turbine flow parts was explored. Ansys, OpenFOAM, and SolidWorks FlowSimulation were discussed, highlighting their key features, capabilities, advantages, and limitations in the context of hydraulic turbine analysis. A comparison was made among these software systems based on factors such as capabilities, ease of use, accuracy, computational resources required, and cost. Additionally, recommendations were provided for selecting the most suitable software system based on specific use cases, objectives, and user requirements.

The accurate calculation of spatial flow in hydraulic turbine flow parts is crucial for optimizing turbine performance, efficiency, and reliability. The software system chosen for this analysis plays a vital role in achieving accurate and reliable results. Careful consideration must be given by engineers and designers to the capabilities, ease of use, accuracy, computational resources, and cost factors when selecting a software system.

By comparing Ansys, OpenFOAM, and SolidWorks FlowSimulation, the strengths and weaknesses of each system have been highlighted. A comprehensive range of solvers and models is offered by Ansys, while SolidWorks FlowSimulation provides seamless CAD integration. OpenFOAM, as an open-source software system, offers flexibility and customization options. The choice of software system should align with the specific needs of the hydraulic turbine analysis project.

Looking ahead, the field of CFD software systems is continuously advancing, and future developments may further enhance the capabilities and performance of these tools for hydraulic turbine analysis. Potential advancements include improved solver algorithms, increased computational efficiency, and enhanced visualization and post-processing capabilities. Significant implications for hydraulic turbine analysis will be brought about by these advancements, enabling engineers and designers to tackle more complex flow phenomena and optimize turbine designs more effectively.

References

1. Krupa Y., Demchuk Y. Modern software for the numerical study of flow in hydraulic machines. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv: NTU "KhPI". 2022. No. 1. P. 54–58.
2. Крупа Е. С., Недовесов В. А. Современное состояние программных комплексов CFD для численного исследования пространственного потока в гидромашинах. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv: NTU "KhPI". 2019. No. 1. P. 98–103.
3. Rezvaya K., Krupa E., Shudryk A., Drankovskiy V., Makarov V. Solving the hydrodynamical tasks using CFD programs. *2018 IEEE 3rd International Conference on Intelligent Energy and Power Systems (IEPS).* Kharkiv, 2018. P. 205–209. doi: 10.1109/IEPS.2018.8559548
4. Keck H., Sick M. Thirty years of numerical flow simulation in hydraulic turbomachines. *Acta Mechanica.* 2008. No. 201. P. 211–229.
5. Pinto R., Afzal A., D'Souza L. Computational Fluid Dynamics in Turbomachinery: A Review of State of the Art. *Archives of Computational Methods in Engineering.* 2017. No. 24. P. 467–479.
6. Kreylos O., Tesdall A., Hamann B., Hunter J. Interactive visualization and steering of CFD simulations. *Proceedings of the symposium on Data Visualisation.* 2002. P. 25–34.
7. Liu S., Shao J., Wu S. Numerical simulation of pressure fluctuation in Kaplan turbine. *Science in China. Series E: Technological Science.* 2008. No. 51. P. 1137–1148.
8. Goyal R., Trivedi C., Kumar G. B. Numerical Simulation and Validation of a High Head Model Francis Turbine at Part Load Operating Condition. *Journal of The Institution of Engineers. Series: C.* 2018. No. 99. P. 557–570.
9. Rusanov A., Rusanov R., Lampart P., Designing and updating the flow part of axial and radial-axial turbines through mathematical modeling. *Open Engineering.* 2015. Vol. 5. P. 399–410.
10. Starodubtsev Y. V., Gogolev I. G., Solodov V. G. Numerical 3D model of viscous turbulent flow in one stage gas turbine and its experimental validation. *Journal of Thermal Science.* 2005. Vol. 14. P. 136–141.
11. Birajdar R., Keste A. Prediction of Flow-Induced Vibrations due to Impeller Hydraulic Unbalance in Vertical Turbine Pumps Using One-Way Fluid−Structure Interaction. *Journal of Vibration Engineering & Technologies.* 2020. No. 8. P. 417–430.
12. *ANSYS CFX: CFD Software.* URL: https://www.ansys.com/products/fluids/ansys-cfx (дата звернення: 02.06.2023).
13. *Tutorial Ansys – How to Make Simulation Fluid Flow by CFX YouTube*. URL: https://www.youtube.com/watch?v=PfZ0opXcqAQ (дата звернення: 03.06.2023).
14. Pandimani M., Geddada Y. Numerical nonlinear modeling and simulations of high strength reinforced concrete beams using

ANSYS. *Journal of Building Pathology and Rehabilitation*. 2022. No. 7. P. 22–30.
15. Bychkov I. M. Verification of the OpenFOAM application package on aerodynamic profile flow problems. *XIX school-seminar "Aerodynamics of Aircraft"*. 2008.
16. Liu H., Ren Y., Wang K. Research of Inner Flow in a Double Blades Pump Based on OpenFOAM. *Journal of Hydrodynamics*. 2012. No. 24. P. 226–234.
17. Zhao M., Zhao W., Wan D. Numerical simulations of propeller cavitation flows based on OpenFOAM. *Journal of Hydrodynamics*. 2020. No. 32. P. 1071–1079.
18. *SOLIDWORKS Flow Simulation | SOLIDWORKS*. URL: https:// www.solidworks.com/product/solidworks-flow-simulation (дата звернення: 05.06.2023).
19. *SOLIDWORKS Flow Simulation | MySolidWorks Training*. URL: https: // my.solidworks.com/ training/ elearning/69/solidworks-flow-simulation (дата звернення: 05.06.2023).
20. *Solidworks FlowSimulation 2020 tutorial*. URL: https://www.academia.edu/41718721/Solidworks_flow_simulation_ 2020_tutorial (дата звернення: 05.06.2023).
21. *SOLIDWORKS Flow Simulation Fluid Mixing Tutorial | GoEngineer*. URL: https://www.goengineer.com/blog/solidworks-flow-simulation-fluid-mixing-tutorial (дата звернення: 05.06.2023).

References (transliterated)

1. Krupa Y., Demchuk Y. Modern software for the numerical study of flow in hydraulic machines. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units*. Kharkiv, NTU "KhPI" Publ., 2022, no. 1, pp. 54–58.
2. Krupa E. S., Nedovesov V. A. Sovremennoe sostoyanie programmnykh kompleksov CFD dlya chislennogo issledovaniya prostranstvennogo potoka v gidromashinakh [Actual status of CFD software complexes for numerical research of spatial flow in hydraulic machines]. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units*. Kharkiv, NTU "KhPI" Publ., 2019, no. 1, pp. 98–103.
3. Rezvaya K., Krupa E., Shudryk A., Drankovskiy V., Makarov V. Solving the hydrodynamical tasks using CFD programs. *2018 IEEE 3rd International Conference on Intelligent Energy and Power Systems (IEPS)*. Kharkiv, 2018, pp. 205–209. doi: 10.1109/IEPS.2018.8559548
4. Keck H., Sick M. Thirty years of numerical flow simulation in hydraulic turbomachines. *Acta Mechanica*. 2008, no. 201, pp. 211–229.
5. Pinto R., Afzal A., D'Souza L. Computational Fluid Dynamics in Turbomachinery: A Review of State of the Art. *Archives of Computational Methods in Engineering*. 2017, no. 24, pp. 467–479.
6. Kreylos O., Tesdall A., Hamann B., Hunter J. Interactive visualization and steering of CFD simulations. *Proceedings of the symposium on Data Visualisation*. 2002, pp. 25–34.
7. Liu S., Shao J., Wu S. Numerical simulation of pressure fluctuation in Kaplan turbine. *Science in China. Series E: Technological Science*. 2008, no. 51, pp. 1137–1148.
8. Goyal R., Trivedi C., Kumar G. B. Numerical Simulation and Validation of a High Head Model Francis Turbine at Part Load Operating Condition. *Journal of The Institution of Engineers. Series: C*. 2018, no. 99, pp. 557–570.
9. Rusanov A., Rusanov R., Lampart P., Designing and updating the flow part of axial and radial-axial turbines through mathematical modeling. *Open Engineering*. 2015, vol. 5, pp. 399–410.
10. Starodubtsev Y. V., Gogolev I. G., Solodov V. G. Numerical 3D model of viscous turbulent flow in one stage gas turbine and its experimental validation. *Journal of Thermal Science*. 2005, vol. 14, pp. 136–141.
11. Birajdar R., Keste A. Prediction of Flow-Induced Vibrations due to Impeller Hydraulic Unbalance in Vertical Turbine Pumps Using One-Way Fluid−Structure Interaction. *Journal of Vibration Engineering & Technologies*. 2020, no. 8, pp. 417–430.
12. *ANSYS CFX: CFD Software*. Available at: https://www.ansys.com/ products/fluids/ansys-cfx (accessed 02.06.2023).
13. *Tutorial Ansys – How to Make Simulation Fluid Flow by CFX YouTube*. Available at: https://www.youtube.com/watch?v= PfZ0opXcqAQ (accessed 03.06.2023).
14. Pandimani M., Geddada Y. Numerical nonlinear modeling and simulations of high strength reinforced concrete beams using ANSYS. *Journal of Building Pathology and Rehabilitation*. 2022, no. 7, pp. 22–30.
15. Bychkov I. M. Verification of the OpenFOAM application package on aerodynamic profile flow problems. *XIX school-seminar "Aerodynamics of Aircraft"*. 2008.
16. Liu H., Ren Y., Wang K. Research of Inner Flow in a Double Blades Pump Based on OpenFOAM. *Journal of Hydrodynamics*. 2012, no. 24, pp. 226–234.
17. Zhao M., Zhao W., Wan D. Numerical simulations of propeller cavitation flows based on OpenFOAM. *Journal of Hydrodynamics*. 2020, no. 32, pp. 1071–1079.
18. *SOLIDWORKS Flow Simulation | SOLIDWORKS*. Available at: https: // www.solidworks.com/product/solidworks-flow-simulation (accessed 05.06.2023).
19. *SOLIDWORKS Flow Simulation | MySolidWorks Training*. Available at: https://my.solidworks.com/training/elearning/69/ solidworks-flow-simulation (accessed 05.06.2023).
20. *Solidworks FlowSimulation 2020 tutorial*. Available at: https://www.academia.edu/41718721/Solidworks_flow_simulation_ 2020_tutorial (accessed 05.06.2023).
21. *SOLIDWORKS Flow Simulation Fluid Mixing Tutorial | GoEngineer*. Available at: https://www.goengineer.com/ blog/solidworks-flow-simulation-fluid-mixing-tutorial (accessed 05.06.2023).

Received 05.09.2023

Відомості про авторів / About the Authors

Крупа Євгеній Сергійович (***Krupa Yevhenii***) – кандидат технічних наук, доцент, Національний технічний університет «Харківський політехнічний інститут», доцент кафедри «Гідравлічні машини ім. Г. Ф. Проскури»; м. Харків, Україна; ORCID: https://orcid.org/0000-0003-3997-3590; e-mail: zhekr@ukr.net

Демчук Роман Миколайович (***Demchuk Roman***) – Національний технічний університет «Харківський політехнічний інститут», магістр кафедри «Гідравлічні машини ім. Г. Ф. Проскури»; м. Харків, Україна; e-mail: roman.demchuk@mit.khpi.edu.ua

Волобуєв Артем Володимирович (***Volobuiev Artem***) – Національний технічний університет «Харківський політехнічний інститут», студент кафедри «Гідравлічні машини ім. Г. Ф. Проскури»; м. Харків, Україна; ORCID: https://orcid.org/0009-0004-7062-2996; e-mail: artem.volobuyev@mit.khpi.edu.ua

Кісь Сергій Леонідович (***Kis Serhii***) – Національний технічний університет «Харківський політехнічний інститут», студент кафедри «Гідравлічні машини ім. Г. Ф. Проскури»; м. Харків, Україна; ORCID: https://orcid.org/0009-0007-3838-3990; e-mail: serhii.kis@mit.khpi.edu.ua