ISSN 2411-3441 (print), ISSN 2523-4471 (online)

UDC 621.224 **doi: 10.20998/2411-3441.2024.1.06**

Ye. KRUPA, R. DEMCHUK

NUMERICAL STUDY OF FLOW PARAMETERS IN THE HIGH-HEAD FRANCIS TURBINE

The scientific exploration of numerical computation regarding spatial flow within hydraulic machinery components is examined. A survey of contemporary software systems is conducted, and the benefits of their utilization over experimental studies are evaluated. It is indicated that the optimal approach involves a blend of experimental investigations and numerical simulation. This methodology facilitates the validation of simulation outcomes under real-world conditions and iteratively enhances the model based on acquired data. A review of the widely utilized Ansys software program is provided, emphasizing its pivotal features and capabilities for analyzing flow components of hydraulic turbines. An algorithm for computing flow parameters in hydraulic turbines using the Ansys software suite is outlined. The subject of this study is the high-head Francis hydraulic turbine Fr 500. The turbine's geometry was constructed employing a sector-based approach. This technique allows for significant simplification of calculations within the computational fluid dynamics framework, thereby reducing computational workload while preserving result accuracy. In selecting mathematical and turbulence models, a comprehensive analysis of the problem was undertaken, identifying models most suitable for the specific situation to ensure dependable numerical simulation outcomes. For spatial flow calculations in the turbine's flow component, the standard k-ε turbulence model was adopted. Considerable attention was devoted to mesh generation, as mesh quality strongly influences result accuracy and reliability. An unstructured mesh comprising tetrahedral-shaped cells was employed for discretizing the flow component, with local mesh refinement at the edges of the runner blades and guide vanes. As a result of numerical computations, the values of primary flow parameters for the design operating mode were determined. A visualization of the flow within the flow component is provided, alongside the assessment of hydraulic losses and turbine efficiency. The efficiency values obtained differ from corresponding experimental values by no more than 1 %. A thorough examination of the flow structure within the flow path was conducted, yielding recommendations for adjusting the blade angle β_1 to reduce inlet impact losses.

Keywords: hydraulic turbine, spatial flow, numerical research, computational fluid dynamics, efficiency, hydraulic losses.

Є. С. КРУПА, Р. М. ДЕМЧУК

ЧИСЕЛЬНЕ ДОСЛІДЖЕННЯ ПАРАМЕТРІВ ПОТОКУ В ВИСОКОНАПІРНІЙ РАДІАЛЬНО-ОСЬОВІЙ ГІДРОТУРБІНІ

Розглянуто науковий напрямок чисельного розрахунку просторового потоку в проточних частинах гідравлічних машин. Проведено огляд сучасних програмних комплексів та проаналізовано переваги їх використання у порівнянні з експериментальними дослідженнями. Зазначено, що оптимальним рішенням є поєднання експериментальних досліджень та чисельного моделювання. Це дозволяє перевірити результати моделювання в реальних умовах і вдосконалити модель на основі отриманих даних. Виконано огляд широко використовуваної програми Ansys, виділено її ключові характеристики та можливості для аналізу проточних частин гідравлічних турбін. Представлено алгоритм розрахунку параметрів потоку в гідравлічних турбінах з використанням програмного комплексу Ansys. Об'єктом дослідження в даній роботі є високонапірна радіально-осьова гідротурбіна РО 500. Побудовано геометрію гідравлічної турбіни з використанням секторного підходу. Цей метод дозволяє значно спростити розрахунки в межах обчислювальної динаміки рідини і зменшити об'єм обчислень без втрати точності результатів. Для вибору математичної моделі та моделі турбулентності проведено докладний аналіз задачі та вибрані моделі, які найкраще відповідають конкретній ситуації, щоб забезпечити надійні результати чисельного моделювання. Для розрахунку просторового потоку в проточній частині гідротурбіни було обрано стандартну k-ε модель турбулентності. Велику увагу приділено створенню обчислювальних сіток, якість яких сильно впливає на точність та надійність отриманих результатів. Для дискретизації проточної частини використано неструктуровану сітку з комірками тетраедральної форми, з локальним згущенням сітки біля кромок лопатей робочого колеса та лопаток направляючого апарату. В результаті чисельного розрахунку були розраховані значення основних параметрів потоку для розрахункового режиму роботи. Представлена картина течії в проточній частині, та отримано значення гідравлічних втрат та коефіцієнту корисної дії гідротурбіни. Отримані значення коефіцієнту корисної дії мають розбіжність з аналогічними експериментальними значеннями не більше 1 %. Проведено всебічний аналіз структури потоку в проточній частині. Запропоновано рекомендації щодо зміни кута лопаті β_1 для зменшення ударних втрат на вході в робоче колесо.

Keywords: гідротурбіна, просторовий потік, чисельне дослідження, обчислювальна динаміка рідини, коефіцієєнт корисрної дії, гідравлічні втрати.

Introduction. Modern hydraulic machines, particularly hydraulic turbines and pumps, play a pivotal role in electricity generation and various industrial processes. To optimize their performance and enhance efficiency, detailed modeling of liquid flow within the components of these machines is essential. In this context, contemporary computational fluid dynamics (CFD) software suites have become indispensable tools in engineering research.

There exists a plethora of CFD software packages, including Ansys Fluent, Open FOAM, Star-CCM+, among others. The selection of a specific software suite depends on task requirements, available resources, and the user's expertise level. These tools empower engineers and researchers to analyze and optimize hydraulic machines by simulating fluid flow [1–10].

Advantages of Numerical Modeling.

Experimental studies:

1. Limitations in accuracy. Experimental studies are often constrained by the precision of instruments and external conditions that may influence results.

2. High costs. Conducting experiments can incur significant expenses and time, especially when specialized equipment or conditions are necessary.

3. Restricted variability. Experiments may be limited by factors that can be manipulated or measured, thereby restricting exploration of diverse scenarios.

Numerical Simulation:

1. Enhanced accuracy. Numerical modeling yields highly accurate results as it relies on mathematical models and calculations.

2. Cost-efficiency. Utilizing numerical simulation

software is generally more economical and cost-effective than conducting physical experiments.

3. Increased flexibility. Numerical simulations facilitate easy adjustment of study parameters and conditions, allowing for exploration of a broader spectrum of scenarios.

4. Safety and environmental considerations. In situations where research poses risks to human safety or the environment, numerical simulation offers a safer alternative.

Often, the optimal approach involves a combination of experimental research and numerical modeling. This allows for validation of simulation outcomes under real-world conditions and refinement of the model based on empirical data [3].

In the contemporary scientific and engineering landscape, numerical modeling stands as a crucial tool for acquiring new knowledge and addressing complex challenges. Consequently, numerical modeling continues to hold a central position in the advancement of science and technology.

Object of study. The object of research in this study is the flow path of the Fr 500-V-100 radial-axial (Francis) hydraulic turbine, which includes its inlet, runner, and outlet components.

General concepts of using the Ansys program to study the energy characteristics of hydro turbines. In this work, a numerical study of the spatial flow in the flow part of the radial-axial hydro turbine was performed using the Ansys software complex.

The Ansys program is a powerful engineering tool that has found wide application in researching the energy characteristics of hydraulic machines. With the help of Ansys, engineers and researchers can perform detailed numerical calculations, flow simulations in turbines, and evaluate their performance.

The algorithm for calculating the spatial flow in hydro turbines using Ansys [10]:

1. Preparation of the geometry of the turbine. The first stage is to create a three-dimensional geometry of the hydro turbine. This includes creating an accurate model of the flow part of a hydro turbine in one of the CAD complexes. Geometry plays a key role in the accuracy of calculations, so it is necessary to pay special attention to this stage.

2. Mesh (grid) geometry. For numerical calculations of water flow, it is necessary to create a mesh that divides the geometry into many finite elements. The mesh must be adapted to the characteristics of the hydro turbine and take into account the complex structure of the flow at the edges of the blades and other parts [5].

3. Determination of boundary conditions. Boundary conditions are set according to the operating conditions of the hydro turbine. This includes specifying the initial velocities and pressures, as well as taking into account the inlet and initial boundary conditions [10].

4. Selection of turbulence model. Different turbulence models can be chosen in Ansys depending on the characteristics of the flow. For example, the k-ε or SST (Shear-Stress Transport) model can be used to account for turbulent phenomena [2].

5. Solving the Navier-Stokes equations. After setting the boundary conditions and selecting the turbulence model Ansys solves the system of Navier-Stokes equations that describe the fluid movement inside a hydro turbine. This allows determination of velocities and pressures at each point in the flow.

6. Analysis of the results. After completing the calculations, you can analyze the results. This includes evaluating the turbine's performance, determining the energy performance, and identifying potential problems such as cavitation orvibrations.

Using Ansys for studying the energy characteristics of hydroturbines empowers engineers and researchers to acquire deeper insights into internal turbine processes, facilitating design optimization for enhanced efficiency and reliability.

This tool assumes a significant role in contemporary hydroturbine development, contributing to the advancement of more efficient renewable energy sources.

Creation of a geometric model of the flow section of Fr 500-V-100. Various CAD (Computer-Aided Design) programs are utilized to construct 3D models.

CAD systems, as specialized software tools, facilitate the creation, editing, and analysis of computer models, represented as 2D and 3D images. In mechanical engineering, CAD systems play a pivotal role in assisting engineers and designers in product development and refinement [1–3].

The analysis of flow in hydraulic machines refers to the internal hydraulic flow, necessitating the import of the geometric model of the flow section into the Ansys program. A crucial requirement for the model is its representation of the internal volume as a solid model.

During the initial design phase, a comprehensive 3D model of the Fr 500-V-100 hydro turbine was constructed using the SolidWorks CAD program (refer to Fig. 1).

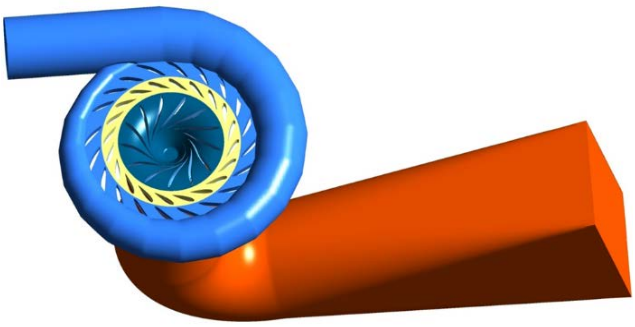


Fig. 1. 3D model of the Fr 500-V-100 hydro turbine

For subsequent calculation of spatial flow within the hydro turbine, a sector-based geometry approach was employed. When modeling flow within the flow path of a hydro turbine featuring complex three-dimensional geometry, such as the Fr 500 hydro turbine, dividing the geometry into sectors proves to be an effective strategy.

This method substantially simplifies calculations within the realm of computational fluid dynamics while reducing computational workload without compromising result accuracy [1–4].

Advantages of sectoring:

- segmenting the turbine into sectors enables the consideration of only a portion of the geometry, thereby decreasing computational effort and resource requirements;

- through meticulous separation and modeling of transitions between sectors, the accuracy of results is maintained at a high level;

- simulating flow within each sector facilitates the examination of interactions between them, enabling evaluation of their collective impact on the overall performance of the hydraulic turbine.

Sectoring process:

1. Geometric segmentation. The hydro turbine undergoes segmentation into sectors, typically in the form of equal angular divisions or divisions tailored to the design's characteristics. These sectors encompass components such as blades, chambers, and other elements.

2. Creation of individual models. A distinct model is generated for each sector, incorporating mesh structures, boundary conditions, and flow parameters, which may vary depending on operational conditions.

3. Modeling within each sector. Initially, calculations are conducted independently within each sector. This approach enables the assessment of flow characteristics within individual sectors.

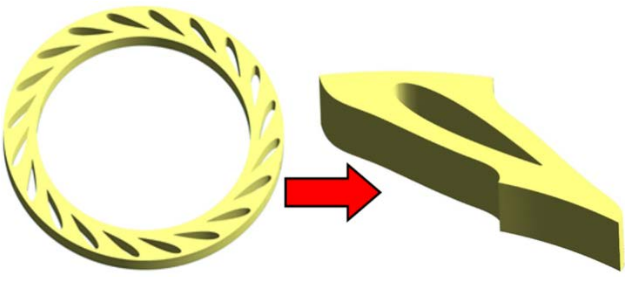
4. Interactions among sectors. Subsequently, results from sector-specific simulations are merged to account for interactions among sectors. This involves considering the influence of flow from one sector on neighboring sectors.

Sectoring constitutes an important method in numerically simulating hydro turbines, facilitating adherence to accuracy requirements while reducing computational complexity.

Consideration of the segmentation of elements within the flow path of the Fr 500-V-100 hydraulic turbine:

a) Guide vane:

The guide vane consists of 20 symmetrically shaped blades and is shown in Fig. 2, *a*. One cut sector (guide vane blade and interblade channels) is shown in Fig. 2, *b*.

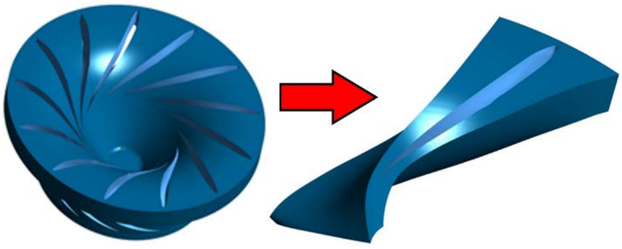


a *b*

Fig. 2. Obtaining the calculated sector of the guide vane:
a – 3D solid model of the guide vane; *b* – one calculated sector of the guide vane

b) Runner:

The runner consists of 13 blades. Its 3D solid model is shown in Fig. 3, *a*. To obtain one calculated sector (runner blade and inter-blade channels), the runner was dissected by two surfaces passing roughly along the middle of the inter-blade channels. This sector is presented in Fig. 3, *b*.



a *b*

Fig. 3. Obtaining the calculated sector of the runner:
a – 3D solid model of the runner; *b* – one calculated sector of the runner

Thus, the finalized geometry of the Fr 500-V-100 hydro turbine, incorporating calculated sectors, takes the following form (refer to Fig. 4).

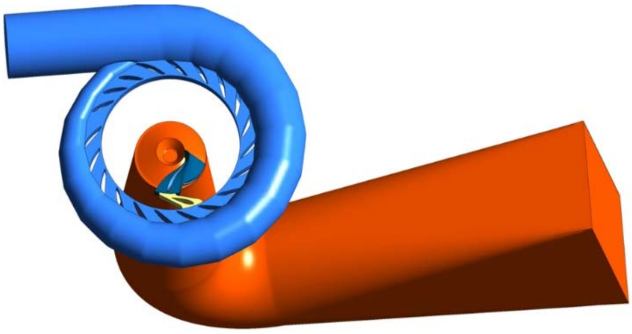


Fig. 4. Three-dimensional model of the Fr 500-V-100 hydro turbine, incorporating calculated sectors

Selection of mathematical and turbulence models. The choice of mathematical and turbulence models is crucial for accurate fluid flow calculations in hydraulic machines using CFD programs, as the reliability of numerical modeling results depends on this selection [10].

Selection of mathematical model:

1. Models of fluid movement. Ansys offers various mathematical models to describe fluid motion, including the Navier-Stokes model, Reynolds-averaged Navier-Stokes (RANS) models, and others. The selection of the model depends on the specific task and flow conditions.

2. Heat transfer calculation. If accounting for heat transfer in a hydraulic machine (e.g., a hydro turbine) is necessary, the choice of a mathematical model should also encompass the energy equations.

Selection of Turbulence Model:

1. k-ε model. This model quantifies turbulent kinetic energy and its dissipation in fluid flow. It is well-suited for turbulent flow calculations but may require significant computing hardware resources [3].

2. SST (Shear-Stress Transport) model. Combining the advantages of the k-ε model with the Reynolds Stress model, this model allows for more accurate consideration of turbulent effects in regions with significant changes in turbulent parameters [4].

3. LES (Large Eddy Simulation) model. Employed for more precise simulation of turbulent flow, particularly in regions with large vortices and irregular structures.

The selection of appropriate mathematical and turbulence models constitutes a pivotal stage in fluid flow calculations in hydraulic machines. It dictates the accuracy and reliability of results while also influencing the computational resources required for simulation. A meticulous analysis of the problem is crucial to choose models that best fit the specific situation, ensuring dependable numerical simulation outcomes.

For the spatial flow calculation within the flow path of the Fr 500-V-100 hydro turbine, this study opts for the standard k-ε turbulence model.

Setting boundary conditions. The numerical experiment was conducted for a hydro turbine model with a runner diameter of $D_1 = 1$ m and a head of $H = 1$ m for the design operating mode of the hydro turbine (according to the universal characteristic Fr 500/3502−V−80).

The guide vane opening for the design mode was set to $a_0 = 62$mm (for the runner diameter of $D_1 = 1$ m). Rotation speed is $n_1' = 70$ rpm.

The following parameters were assigned to the boundaries of the calculation domains:

- at the inlet: the volumetric flow rate $Q_1' = 266$ l/sec;

- on the walls: no-slip condition (velocity equals zero);

- at the outlet: the static pressure $P = 101325$ Pa.

Meshing. The creation of an accurate computational mesh stands as a fundamental stage in numerically modeling a hydro turbine using CFD programs. The quality of the computational mesh significantly impacts the accuracy and reliability of the obtained results [5–8].

Types of computational meshes:

1. Structured mesh. This mesh comprises regular geometric elements like cubes, parallelepipeds, or prisms and finds utility in relatively simple geometry shapes. Structured meshes are straightforward to generate and offer good accuracy in regular areas.

2. Unstructured mesh. Employed for complex geometries with irregular shapes, unstructured meshes incorporate triangles, quadrilaterals, and other irregular elements. This mesh type facilitates the comprehensive description of intricate areas.

Mesh cell types:

1. Tetrahedral cells. Comprising four triangular faces, tetrahedral cells are primarily used for irregular geometries.

2. Hexahedral cells. Featuring six square faces, hexahedral cells are employed in structured grids or to depict regular geometries.

3. Polygon cells. Enabling the creation of highly complex shapes to describe geometry details, polygon cells can pose processing challenges.

In this study, an unstructured mesh with tetrahedral cells, incorporating local mesh refinement near the edges of the runner and guide vanes, was employed to discretize the flow section.

Prismatic cells were utilized on the walls of the calculation domains to effectively capture the boundary layer.

Results of a numerical study on the flow in the Fr 500-V-100 hydro turbine. Ansys provides a broad array of tools for visualizing the results of numerical simulations, facilitating the analysis and interpretation of obtained data [11–12]. The program enables the creation of visual representations of fluid flow, speed distribution, pressure, temperature, and other parameters within a hydro turbine. The Ansys graphical interface simplifies the customization of graphics, the generation of animations, and the creation of user-friendly reports, aiding in a better understanding of results and informed engineering decisions. Fig. 5 through 13 depict visualizations of the results of spatial flow calculations in the Ansys program.

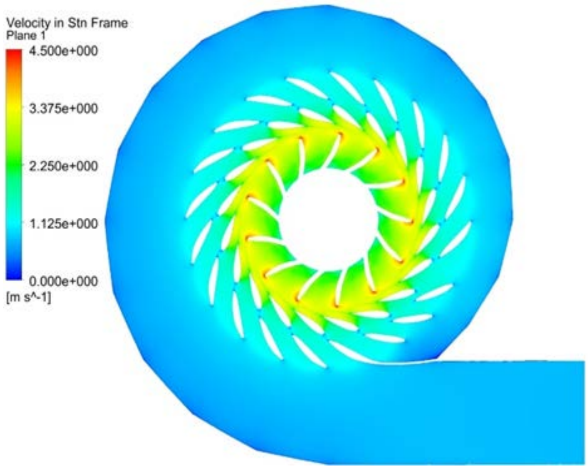


Fig. 5. Distribution of the absolute velocity in the middle section along the height of the guide vane

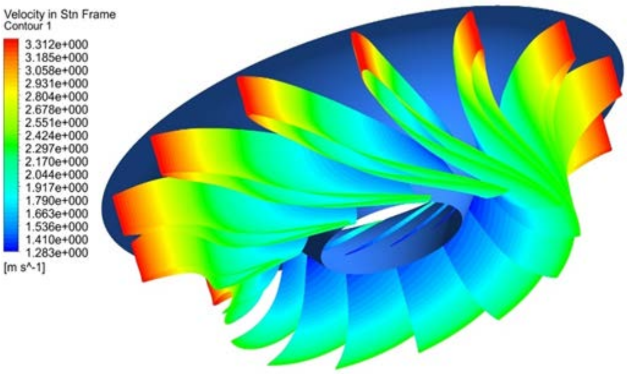


Fig. 6. Distribution of the absolute velocity along the surfaces of the runner blades

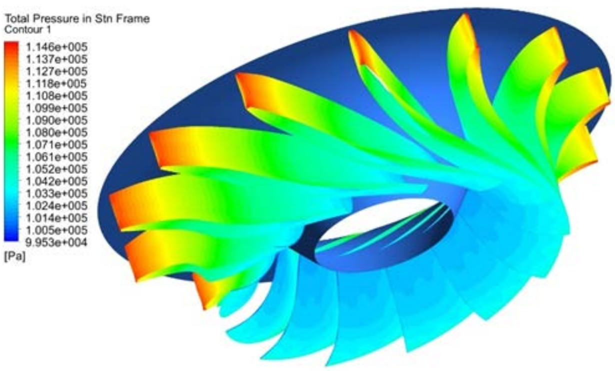


Fig. 7. Distribution of the total pressure along the surfaces of the runner blades

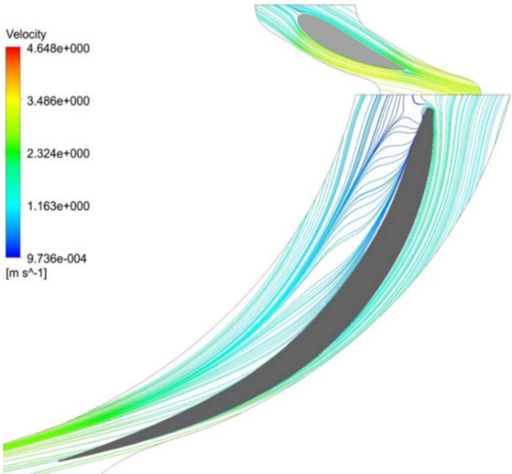


Fig. 8. Relative velocity contours in the guide vane and runner

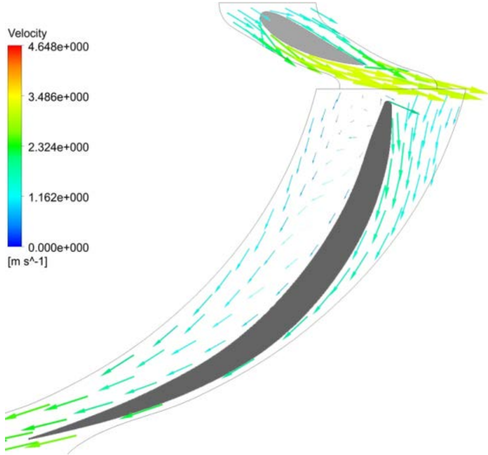


Fig. 9. Relative velocity vectors in the guide vane and runner

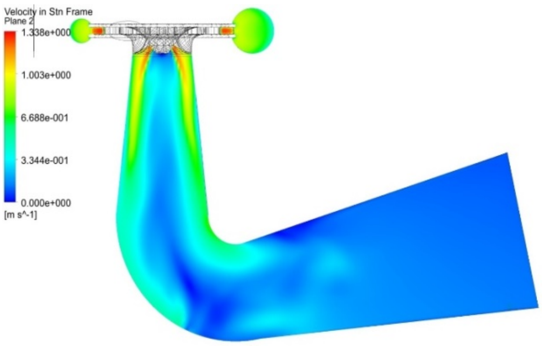


Fig. 10. Distribution of the absolute velocity in the section of the draft tube

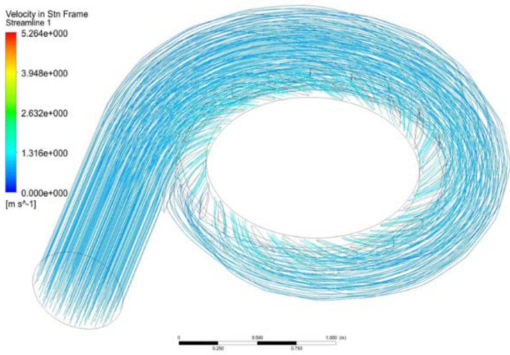


Fig. 11. Fluid movement streamlines in the inlet

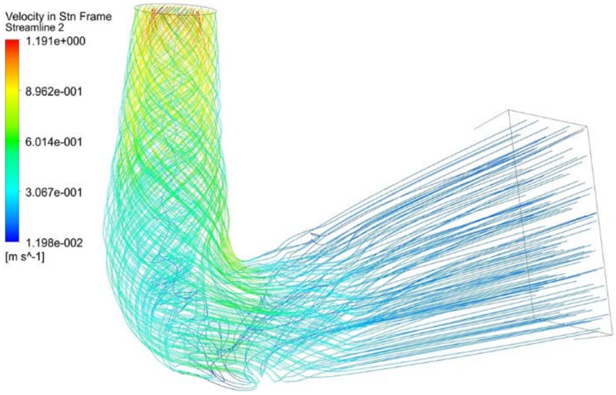


Fig. 12. Fluid movement streamlines in the draft tube

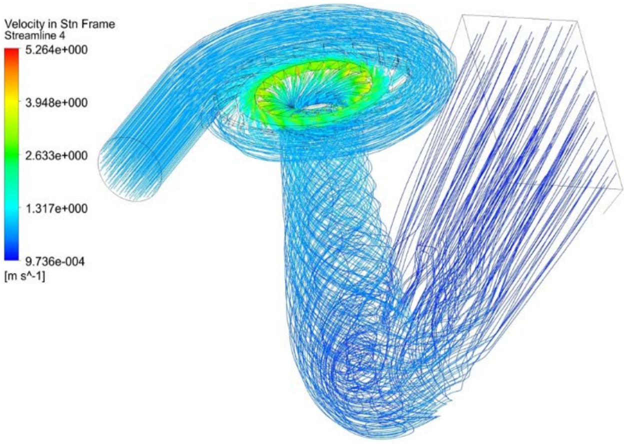


Fig. 13. Fluid movement streamlines in the flow path of the Fr 500-V-100 hydro turbine

Calculation of hydraulic losses. The Ansys program offers tools for accurately computing the hydrodynamic parameters of a hydro turbine, including hydraulic losses. Hydraulic losses encompass pressure losses distributed throughout the hydro turbine's geometry due to fluid resistance and friction. Accurately determining these losses is crucial for assessing the efficiency and operational characteristics of the hydro turbine [10].

The total losses in the inlet and draft tube were computed as the difference between the total energy at the inlet and outlet, divided by the specific gravity [10].

$$h = \frac{P_{\text{in}} - P_{\text{out}}}{\rho g}. \tag{1}$$

The total losses in the runner were determined using the formula:

$$h_r = 1 - \eta_{hyd} = 1 - \frac{N_{ef}}{\rho g Q_r H_r} = 1 - \frac{M\,\omega}{\rho g Q_r H_r}. \tag{2}$$

The hydraulic efficiency of the Fr 500-V-100 hydro turbine was determined using the formula [10]:

$$\eta_{hyd} = \frac{N_{ef}}{\rho g Q H} = \frac{M\,\omega}{\rho g Q H}. \tag{3}$$

After performing the numerical calculation, the value of the moment on the shaft of the hydraulic unit M was obtained. The value of the moment was used to calculate the hydraulic efficiency (formula 3).

Table 1 shows data on the amount of losses in the flow path of the hydro turbine Fr 500-V-100 in the designed mode of operation.

Table 2 shows a comparison of the values of the hydraulic efficiency obtained by numerical experiment and the efficiency based on the universal characteristic in the designed mode.

Table 1 – Hydraulic losses in the flow path

Elements of the flow path	Losses, %
Inlet (stator + SK + NA)	5.0
Runner	3.1
Draft tube	2.9

Table 2 – Hydraulic efficiency of the Fr 500-V-100

	Efficiency, %
Hydro turbine efficiency (as per the universal characteristic Fr 500/3502-V-80)	89
Hydro turbine efficiency (as per the numerical experiment results)	88.4

Analysis of numerical simulation results. The analysis of numerical simulation results in the Ansys program is pivotal in engineering research, offering deep insights into the physical processes occurring inside the hydro turbine and facilitating the evaluation of its performance and efficiency.

Upon analyzing the results of the numerical calculation of three-dimensional flow in the flow section of the Fr 500-V-100 hydro turbine, the following specific observations clarify the flow structure:

- the flow structure in the spiral case appears uniform without significant vortex formations (Fig. 5);
- the uniform flow can be observed in the vicinity of the stay and guide vanes, devoid of noticeable flow breaks (Figs. 5, 8–9);
- velocity peaks manifest at the inlet edges of the runner blades (Fig. 5–6);
- pressure distribution on the surfaces of the runner blades is uniform, with minimum pressure values observed on the backside of the blades along the upper rim and the outlet edge (Fig. 7);
- Figs. 8 and 9 indicate a slight separation of flow on the runner blade, particularly on the front side closer to the inlet edge, where the flow levels off from the middle of the profile. Detecting such flow separations on the blade enables recommendations for adjusting the blade angle β_1 to mitigate shock losses at the runner inlet;
- the liquid flow in the draft tube exhibits non-uniform velocity values, particularly evident in the inlet cone (Figs. 10, 12). A small vortex bundle is observed in the input cone of the draft tube (Figs. 10, 12), with stagnant zones, where the flow rate is zero or close to zero, visible in the elbow and outlet diffuser of the draft tube (Fig. 10).

Figs. 12 and 13 display fluid movement streamlines in the draft tube, revealing leveled and stabilized flow in the outlet diffuser.

Fig. 13 provides a comprehensive overview of liquid movement throughout the flow path, facilitating flow structure analysis and recommendations for geometry adjustments to enhance the hydro turbine's energy performance.

Conclusions. A numerical experiment was conducted using the Ansys software complex for the designed mode of operation of the Fr 500-V-100 radial-axial (Francis) hydro turbine. The visualization of the flow in the flow section is presented, along with the values of hydraulic losses and turbine efficiency. The efficiency values obtained exhibit a discrepancy of no more than 1 % compared to similar experimental values.

A comprehensive analysis of the flow structure in the flow path was performed, leading to recommendations for adjusting the blade angle β_1 to mitigate impact losses at the runner inlet.

In conclusion, the Ansys software complex proves to be effective for numerically simulating the flow in the flow path of hydro turbines, conducting further analysis of the obtained results, and enhancing the structural elements of the flow section to improve turbine efficiency.

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. Prabowoputra D. M., Prabowo A. R., Hadi S., Sohn J. M. Assessment of turbine stages and blade numbers on modified 3D Savonius hydrokinetic turbine performance using CFD analysis. *Multidiscipline Modeling in Materials and Structures*. 2021. Vol. 17, no. 1. P. 253–272. doi: 10.1108/MMMS-12-2019-0224
3. 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.
4. Роговий А. С., Азаров А. С., Толстий П. В. Числове моделювання картин течії газу та характеристики відцентрового компресора. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv: NTU "KhPI". 2022. No. 2. P. 18–23.
5. Zhao M., Zhao W., Wan D. Numerical simulations of propeller cavitation flows based on OpenFOAM. *Journal of Hydrodynamics*. 2020. No. 32. P. 1071–1079.
6. 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.
7. Brijkishore, Khare R., Prasad V. Performance Evaluation of Kaplan Turbine with Different Runner Solidity Using CFD. *Advances in Intelligent Systems and Computing.* Singapore: Springer, 2020. P. 757–767. doi: 10.1007/978-981-13-8196-6_67
8. Kumar P., Saini R. P. CFD on Francis Turbine Under Different Load Conditions. *Lecture Notes in Civil Engineering.* 2024. Vol. 391. P. 461–468. doi: 10.1007/978-981-99-6616-5_52
9. 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.
10. Krupa Y. Calculation of the spatial flow in the Francis high-head turbine using the CFD software package. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units*. Kharkiv: NTU "KhPI". 2021. No. 2. P. 87–93.
11. *ANSYS CFX: CFD Software.* URL: https://www.ansys.com/products/fluids/ansys-cfx (дата звернення: 05.06.2024).
12. *Tutorial Ansys – How to Make Simulation Fluid Flow by CFX YouTube*. URL: https://www.youtube.com/watch?v=PfZ0opXcqAQ (дата звернення: 05.06.2024).

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. Prabowoputra D. M., Prabowo A. R., Hadi S., Sohn J. M. Assessment of turbine stages and blade numbers on modified 3D Savonius hydrokinetic turbine performance using CFD analysis. *Multidiscipline Modeling in Materials and Structures*. 2021, vol. 17, no. 1, pp. 253–272. doi: 10.1108/MMMS-12-2019-0224
3. 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.
4. Rogovyi A. S., Azarov A. S., Tolstyi P. V. Chyslove modelyuvannya kartyn techiyi hazu ta kharakterystyky vidtsentrovoho kompresora. [Numerical modeling of gas flow patterns and characteristics of a centrifugal compressor]. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv, NTU "KhPI" Publ., 2022, no. 2, pp. 18–23.
5. Zhao M., Zhao W., Wan D. Numerical simulations of propeller cavitation flows based on OpenFOAM. *Journal of Hydrodynamics.* 2020, no. 32, pp. 1071–1079.
6. 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.
7. Brijkishore, Khare R., Prasad V. Performance Evaluation of Kaplan Turbine with Different Runner Solidity Using CFD. *Advances in Intelligent Systems and Computing.* Singapore, Springer Publ., 2020, pp. 757–767. doi: 10.1007/978-981-13-8196-6_67
8. Kumar P., Saini R. P. CFD on Francis Turbine Under Different Load Conditions. *Lecture Notes in Civil Engineering*. 2024, vol. 391, pp. 461–468. doi: 10.1007/978-981-99-6616-5_52
9. 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.
10. Krupa Y. Calculation of the spatial flow in the Francis high-head turbine using the CFD software package. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units*. Kharkiv, NTU "KhPI" Publ., 2021, no. 2, pp. 87–93.
11. *ANSYS CFX: CFD Software*. Available at: https://www.ansys.com/products/fluids/ansys-cfx (accessed 05.06.2024).
12. *Tutorial Ansys – How to Make Simulation Fluid Flow by CFX YouTube.* Available at: https://www.youtube.com/watch?v=PfZ0opXcqAQ (accessed 05.06.2024).

Received 12.06.2024

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», аспірант кафедри «Гідравлічні машини ім. Г. Ф. Проскури»; м. Харків, Україна; ORCID: https://orcid.org/0009-0008-3229-0395; e-mail: roman.demchuk@mit.khpi.edu.ua