UDC 621.224 **doi: 10.20998/2411-3441.2021.2.13**

Y. KRUPA

CALCULATION OF THE SPATIAL FLOW IN THE FRANCIS HIGH-HEAD TURBINE USING THE CFD SOFTWARE PACKAGE

At present, the development of software packages for calculating computational fluid dynamics problems has reached a high level of efficiency, accuracy and flexibility, with their help it is possible to solve the most diverse and complex problems. All modern software packages for computational fluid dynamics solve the problems of continuum mechanics using models based on the Navier-Stokes equations. These models are based on three conservation equations: conservation of mass, conservation of momentum and conservation of energy. A numerical simulation of the spatial flow of a high-head radial-axial hydraulic turbine Fr 310 was carried out for two variants of the flow path – with an runner with 15 blades (modification 1) and with 17 blades (modification 2), using the OpenFOAM software package. The OpenFOAM software package is one of the most used products designed to solve fluid dynamics problems and is distributed under a free GPL license (General Purpose License). The process of solving the set hydrodynamic problems using the CFD (Computational fluid dynamics) software package includes the following stages: creating a three-dimensional model of the object under consideration using a computer-aided design system; construction of a computational grid with the required parameters; selection of a mathematical model that most accurately describes the working process in the flow parts of hydraulic machines; selection of a suitable turbulence model; setting boundary conditions. A visualization of the results of a numerical study of two modifications of the Fr 310-V-100 hydraulic turbine is presented. A method for calculating hydraulic losses in the flow path of a hydraulic turbine is presented. The analysis of the results of numerical simulation was performed. This analysis showed that the modification of a hydraulic turbine with a runner with 15 blades is better in terms of efficiency than the modification with 17 blades. Comparison of the two modifications was carried out exceptionally by the values of the hydraulic efficiency of the hydraulic turbine.

Keywords: hydraulic machines, runner, flow path, efficiency, mathematical model, spatial flow.

Є. С. КРУПА

РОЗРАХУНОК ПРОСТОРОВОЇ ТЕЧІЇ У ВИСОКОНАПІРНІЙ РАДІАЛЬНО-ОСЬОВІЙ ГІДРОТУРБІНІ З ВИКОРИСТАННЯМ ПРОГРАМНОГО ПАКЕТУ CFD

В даний час розвиток пакетів прикладних програм для розрахунку задач обчислювальної гідроаеродинаміки досяг високого рівня ефективності, точності і гнучкості, з їх допомогою можна вирішувати самі різноманітні та складні задачі. Всі сучасні пакети програм обчислювальної гідроаеродинаміки вирішують завдання механіки суцільного середовища, використовуючи моделі, побудовані на основі рівнянь Нав'є-Стокса. В основу цих моделей входять три рівняння збереження: збереження маси, збереження імпульсу і збереження енергії. Було проведено чисельне моделювання просторового потоку високонапірної радіально-осьової гідротурбіни РО310 для двох варіантів проточної частини – с робочим колесом, що має 15 лопатей (модифікація 1) та з 17 лопатями (модифікація 2), з використанням пакета прикладних програм OpenFOAM. Програмний комплекс OpenFOAM є одним з найбільш використовуваних продуктів, призначених для вирішення завдань гідродинаміки, що розповсюджуються за вільною ліцензією GPL (General Purpose License). Процес вирішення поставлених гідродинамічних задач за допомогою програмного комплексу CFD (Computational Fluid Dynamics) включає в себе наступні етапи: створення тривимірної моделі розглянутого об'єкта за допомогою системи автоматичного проектування; побудова розрахункової сітки з необхідними параметрами; вибір математичної моделі, яка найточніше описує робочий процес в проточних частинах гідромашин; вибір відповідної моделі турбулентності; завдання граничних умов. Приведено візуалізацію результатів чисельного дослідження двох модифікацій гідротурбіни РО 310-В-100. Представлено методику розрахунку гідравлічних втрат в проточній частині гідротурбіни. Виконано аналіз результатів чисельного моделювання. Даний аналіз показав, що модифікація гідротурбіни з робочим колесом, що має 15 лопатей краща по значенню ККД, ніж модифікація з 17 лопатями. Порівняння двох модифікацій проводилося виключно по значенням гідравлічного ККД гідротурбіни.

Ключові слова: гідравлічні машини, робоче колесо, проточна частина, коефіцієнт корисної дії, математична модель, просторова течія.

Е. С. КРУПА

РАСЧЕТ ПРОСТРАНСТВЕННОГО ПОТОКА В ВЫСОКОНАПОРНОЙ РАДИАЛЬНО-ОСЕВОЙ ГИДРОТУРБИНЕ С ИСПОЛЬЗОВАНИЕМ ПРОГРАММНОГО ПАКЕТА CFD

В настоящее время развитие пакетов прокладных программ для расчета задач вычислительной гидроаэродинамики достигло высокого уровня эффективности, точности и гибкости, с их помощью можно решать самые разнообразные и сложные задачи. Все современные пакеты программ вычислительной гидроаэродинамики решают задачи механики сплошной среды, используя модели, построенные на основе уравнений Навье-Стокса. В основу этих моделей входят три уравнения сохранения: сохранения массы, сохранения импульса и сохранения энергии. Было проведено численное моделирование пространственного потока высоконапорной радиально-осевой гидротурбины РО 310 для двух вариантов проточной части – с рабочим колесом, имеющим 15 лопастей (модификация 1) и с 17 лопастями (модификация 2), с использованием пакета прикладных программ OpenFOAM. Программный комплекс OpenFOAM является одним из наиболее используемых продуктов, предназначенных для решения задач гидродинамики, и распространяется по свободной лицензии GPL (General Purpose License). Процесс решения поставленных гидродинамических задач с помощью программного комплекса CFD (Computational fluid dynamics) включает в себя следующие этапы: создание трехмерной модели рассматриваемого объекта с помощью системы автоматизированного проектирования; построение расчетной сетки с необходимыми параметрами; выбор математической модели, которая наиболее точно описывает рабочий процесс в проточных частях гидромашин; выбор подходящей модели турбулентности; задания граничных условий. Приведена визуализация результатов численного исследования двух модификаций гидротурбины РО 310-В-100. Представлена методика расчета гидравлических потерь в проточной части гидротурбины. Выполнен анализ результатов численного моделирования. Данный анализ показал, что модификация гидротурбины с рабочим колесом, имеющим 15 лопастей лучше по значению КПД, чем модификация с 17 лопастями. Сравнение двух модификаций проводилось исключительно по значениям гидравлического КПД гидротурбины.

Ключевые слова: гидравлические машины, рабочее колесо, проточная часть, коэффициент полезного действия, математическая модель, пространственное течение.

Introduction. With the development of methods of mathematical modeling and the widespread introduction into computer engineering practice, it became possible to replace the physical experiment with a numerical one. The use of numerical modeling greatly expands the possibilities of analyzing the influence of geometric parameters of the hydraulic machine on the kinematic and energy characteristics. Numerical experiment allows to estimate influence of separate geometrical parameters not only on power characteristics of the hydro turbine as a whole, but also on kinematic characteristics of its separate elements, and also on categories of losses in these elements. The latter allow us to identify ways to reduce certain types of losses, which provides a basis for improving the energy performance of the turbine as a whole.

One of the advantages of numerical modeling is the ability to study the properties of the object in a wide range of variable parameters and with different combinations [1–20].

Currently, the development of commercial packages for computational hydro aerodynamics, such as ANSYS CFX, ANSYS FLUENT, OpenFOAM, CD-adapco STAR-CD and START-CCM + or FlowVision package, has reached a high level of efficiency, accuracy and flexibility; thus they can be used to solve a variety of complex problems. Moreover, today it is difficult to imagine the development of such industries as energy engineering, automotive, aerospace or aerospace without the use of these packages [1–20].

All modern CFD software packages solve problems of continuous environment mechanics using models based on Navier-Stokes equations. These models are based on three conservation equations: mass conservation, momentum conservation, and energy conservation.

The solution of the computational hydrodynamic problem within the CFD package takes place in three stages. At the first geometry is created, the computer grid is constructed, boundary conditions (preprocessor) are set, at the second stage the decision with the use of the solver corresponding to a concrete solved problem, and, at last, by means of special software tools - post-processors, the obtained results are presented graphically and analyzed [1–20].

In this work, the OpenFOAM software package was used for numerical research of the flow in the flow path (FP) of the Fr 310 turbine.

OpenFOAM software is currently widely used to solve applied problems of fluid dynamics [10–13, 20].

Object of study. Terms of calculations. The flow path of the hydro turbine Fr 310 (spiral casing, stay vanes, guide vanes, runner, draft tube) was accepted as the object of research.

The main geometric characteristics of the elements of the flow path:

1) Spiral casing (SC) with round and elliptical meridional sections; girth angle $\varphi_{sc} = 345°$. The stay ring consists of 19 stay vanes, including a spiral tooth.

2) The profile of the vane of the distributor is symmetrical, height $b_0 = 0{,}12 \cdot D_1$, the diameter of the axis of rotation of the vanes $D_0 = 1{,}2 \cdot D_1$, the number of vanes $z_0 = 24$.

3) The number of runner blades (RB) $z_1 = 15$ (modification 1), $z_1 = 17$ (modification 2).

4) Parameters of the draft tube (DT): height $h = 3{,}95 \cdot D_1$, length $L = 4D_1$, bend type KU-3RO.

Creating a geometric model of the flow path of the Fr 310. Calculation of water flow in the hydraulic unit is a task of internal flow, so in OpenFOAM it is enough to import the geometry of the FP. The internal volume of the FP should be represented as a solid model (Fig. 1).

Fig. 1. Solid model of the flowing part of the hydro turbine Fr 310-V-100

Mathematical model. One of the main stages of solving the problem is the choice of calculation model.

At the first calculation stage, the medium is assumed to be a single-phase uncompressed viscous, the flow is turbulent. The simulated process is assumed to be isothermal.

The criterion of turbulence is the Reynolds number.

A system of Reynolds-averaged continuity equations (2) and Navier-Stokes (1) is used for mathematical modeling of turbulent flow in the FP of the Fr 310 turbine.

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + f_i; \quad (1)$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0, \quad (2)$$

where i, j = 1–3 – summation of the same indices; x_1, x_2, x_3 – coordinate axes; u_1, u_2, u_3 – time-averaged values of velocities; f_i – expresses the action of mass forces. The flow in rotating working elements is considered in the relative coordinate system, and the term *f*i in the right part of equation (1) expresses the action of centrifugal and Coriolis forces: $\vec{f}_i = -\rho\left(2\vec{\omega}\times\vec{u} + \vec{\omega}\times\left(\vec{\omega}\times\vec{r}\right)\right)$, where $\vec{\omega}$ – angular speed of rotation; $\vec{r}$ – radius-vector (the modulus of which is equal to the distance from a given point to the axis of rotation).

To close the system of equations (1–2) in this work, the *k-ε* turbulence model was used.

The *k-ε* model of turbulence, as well as its modifications, is widely used in modern software products. When using this model, the system of equations of fluid motion is supplemented by two differential equations describing the transfer of the kinetic energy of turbulence k and the dissipation rate ε, respectively [3, 4, 9].

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}\left(\rho \overline{u_j} k\right)=\frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right)+P_k-\rho\varepsilon ; \qquad (3)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_j}\left(\rho \overline{u_j}\varepsilon\right)=\frac{\partial}{\partial x_j}\left(\Gamma_\varepsilon \frac{\partial \varepsilon}{\partial x_j}\right)+\frac{\varepsilon}{k}\left(C_{\varepsilon 1}P_k-\rho C_{\varepsilon 2}\varepsilon\right), \quad (4)$$

where $P_k=-\rho\overline{u_i' u_j'}\frac{\partial \overline{u_i}}{\partial x_j}$ – a member expressing the energy generation k, $\Gamma_k=\mu+\frac{\mu_t}{\sigma_k}$, $\Gamma_\varepsilon=\mu+\frac{\mu_t}{\sigma_\varepsilon}$.

Tasks of boundary conditions. Numerical studies were performed for the model with a diameter of runner D_1 = 1 m at a head of H = 1 m for the optimal mode of operation of the turbine (according to the universal characteristic RO 310/1107-V-38.8). The opening of the guide vanes for the optimal mode a_0 = 51,55 mm (for D_1 = 1 m). Rotate n_I' = 63,5 r/m.

The following parameters were set at the boundaries of the calculation areas:

- at the entrance – mass regime consumption Q_I' = 400 l/s;

- on the wall – the condition of adhesion (speed is zero);

- at the outlet – static pressure P = 101325 Pa.

The task of the original calculation grid. One of the most important stages in creating a calculation model is to build a calculation grid. Construction of the calculation grid is a process of dividing the calculation area into many separate cells. Grid cells are polyhedral, usually tetrahedral, hexahedrons, prisms or pyramids. The edges of these cells form the lines of the calculation grid, and the points located on the edges or in the center of the cells – the nodes of the calculation grid. As a result of the numerical solution of the equations of the mathematical model, it is in the nodes of the computational grid that the required flow parameters are determined [6–8, 11].

Sampling of the studied flow part was performed using an unstructured grid with tetrahedral cells with local thickening at the edges of the stay vanes, guide vanes, RB.

The total number of elements is 7,5 million: SC with stay vanes – 3,1 million; one interscapular canal of the guide vane – 500 thousand; one interscapular canal of the runner – 2,7 million; DT – 1,2 million

For a qualitative description of the boundary layer, prismatic cells were built on the walls of the regions (Fig. 2).

Numerical research of the flow in the flow path of the Fr 310 for two modifications of the runner. According to the recommendations adopted in hydraulic turbine construction, for heads above 300 m the number of runner blades is taken as z = 13–19 [15].

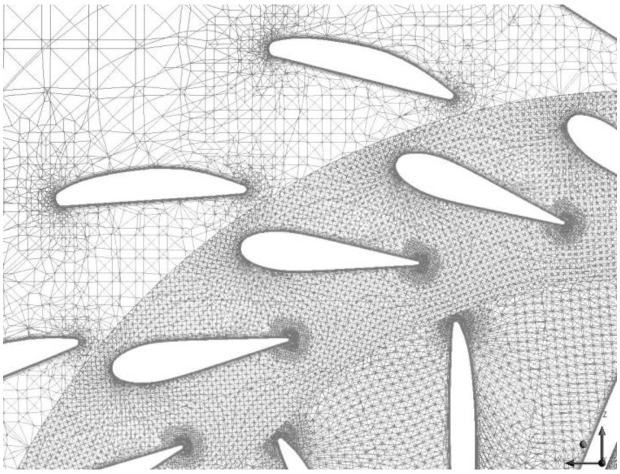

Fig. 2. Calculated grid of the model

In this work, a numerical study of the spatial flow was performed for two variants of the FP:

1) FP of the Fr 310 hydro turbine with runner with 15 blades (modification 1);

2) FP of the Fr 310 hydro turbine with runner with 17 blades (modification 2).

As a result of the numerical experiment, the following data were obtained and analyzed for two modifications of the runner:

- visualization of the flow (velocity and pressure fields) in the characteristic cross sections of FP of the hydro turbine Fr 310;

- plots of speed and pressure distribution along the front and rear surfaces of the runner blade;

- hydraulic torque on the shaft of the hydraulic unit;

- hydraulic efficiency of the hydro turbine.

Modification 1. Fig. 3 shows the geometry of the model of the FP of the hydraulic turbine Fr 310-V-100 with the runner having 15 blades (modification 1).

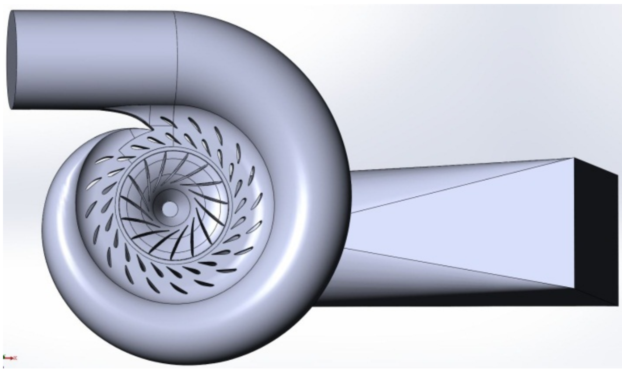

Fig. 3. The geometry of the flow path with z_1 = 15 (modification 1)

Figures 4–7 show a visualization of part of the results of the calculation of the spatial flow in the package OpenFOAM for modification 1.

Modification 2. Fig. 9 shows the geometry of the model of the FP of the hydraulic turbine Fr 310-V-100 with the runner having 17 blades (modification 2).

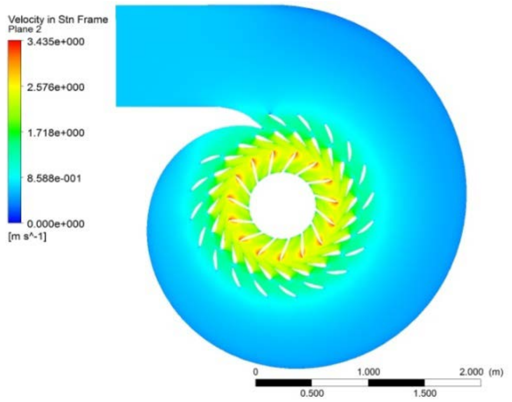

Fig. 4. Distribution of absolute velocity in the middle section on height of the guide vanes (modification 1)

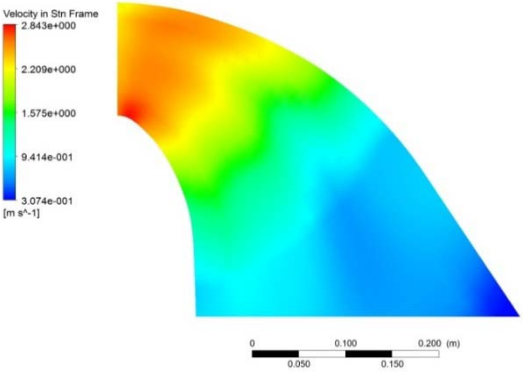

Fig. 5. Distribution of absolute velocity in the meridional section of the runner (modification 1)

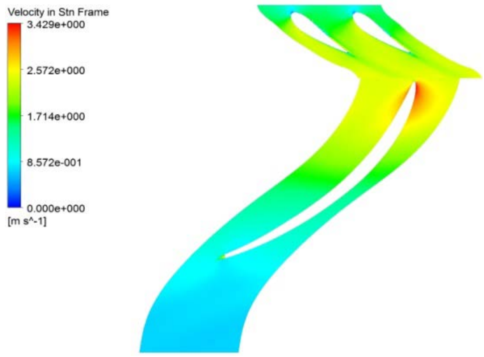

Fig. 6. Absolute velocity field in the gratings of the guide vanes and the runner (modification 1)

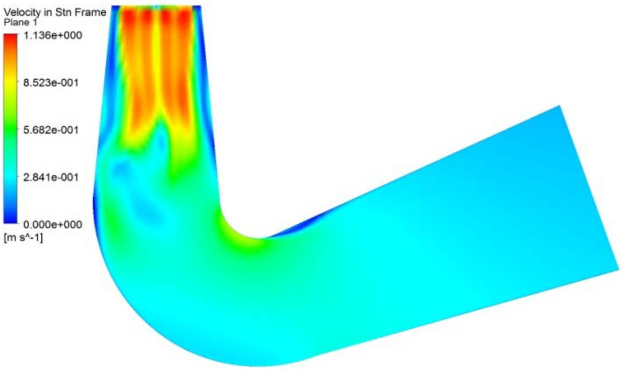

Fig. 7. Absolute velocity field in the draft tube of the hydro turbine (modification 1)

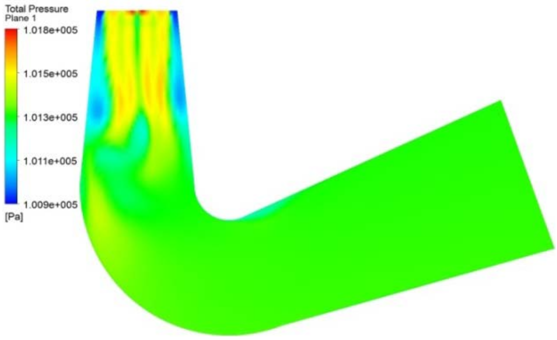

Fig. 8. Total pressure field in the the draft tube of the hydro turbine (modification 1)

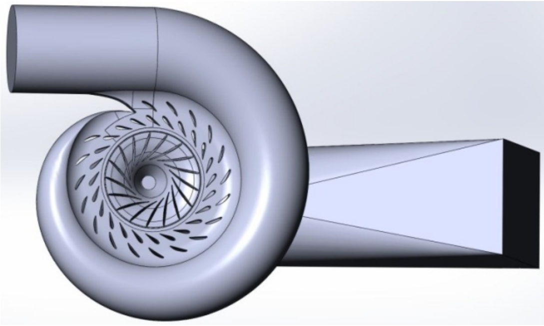

Fig. 9. The geometry of the flow path with z_1 = 17 (modification 2)

Figures 10–14 show a visualization of part of the results of the calculation of the spatial flow in the package OpenFOAM for modification 2.

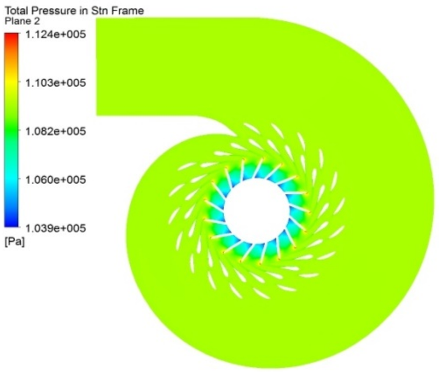

Fig. 10. Distribution of total pressure in the middle section on height of the guide vanes (modification 2)

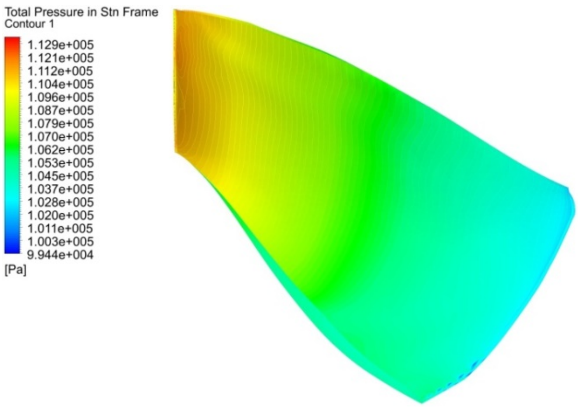

Fig. 11. Distribution of total pressure along the front surface of the runner blade (modification 2)

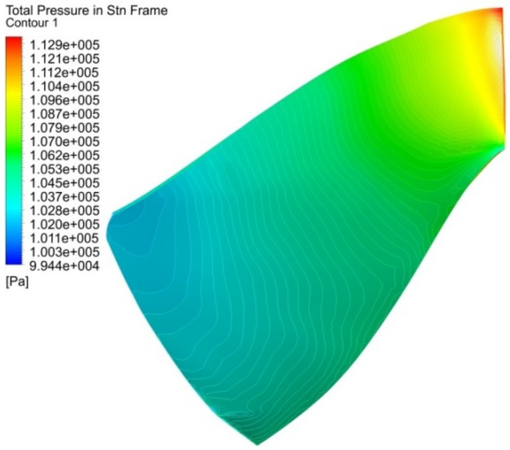

Fig. 12. Distribution of total pressure along the rear surface of the runner blade (modification 2)

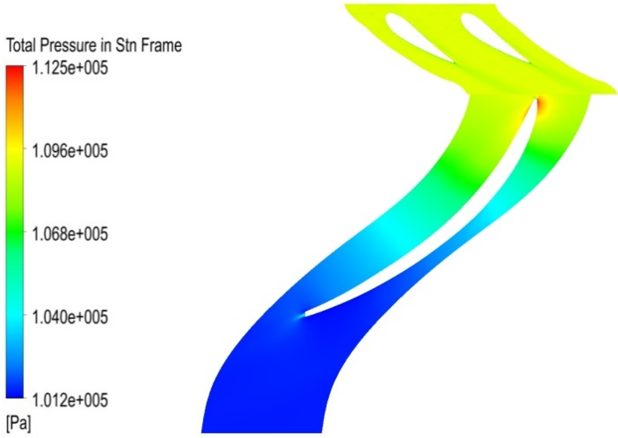

Fig. 13. Total pressure field in the gratings of the guide vanes and the runner (modification 2)

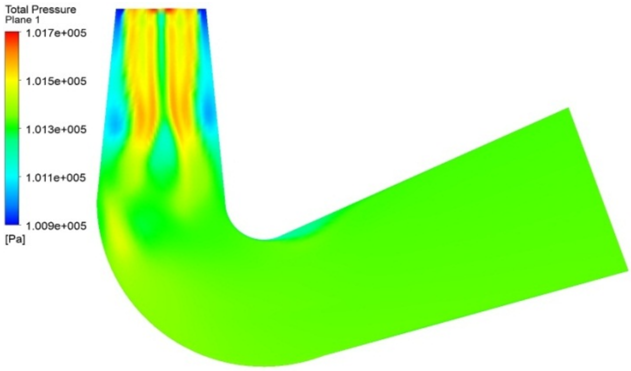

Fig. 14. Total pressure field in the draft tube of the hydro turbine (modification 2)

Calculation of hydraulic losses in the elements of the flow path. The total losses in the supply (spiral casing, stay ring, distributor), draft tube were calculated as the difference of total energy at the inlet and outlet, divided by the specific gravity.

$$h = \frac{P_{\text{in}} - P_{\text{out}}}{\rho g}. \qquad (5)$$

Total losses in the runner were calculated by the formula:

$$h_r = 1 - \eta_g = 1 - \frac{N_{ef}}{\rho g Q_r H_r} = 1 - \frac{M\omega}{\rho g Q_r H_r}. \qquad (6)$$

The hydraulic efficiency of the Fr 310 turbine was determined by the formula:

$$\eta_h = \frac{N_{ef}}{\rho g Q H} = \frac{M\omega}{\rho g Q H}. \qquad (7)$$

The value of torque M on the shaft of the hydraulic unit was obtained as a result of a numerical experiment.

Table 1 shows the value of losses in the elements of the flow path of the turbine Fr 310 for two modifications of the runner.

Table 2 presents a comparison of the values of hydraulic efficiency obtained by a numerical experiment with the efficiency of the universal characteristic.

Table 1 – Hydraulic losses in the flow path

Elements of the flow path	Modification 1 (15 blades)	Modifications 2 (17 blades)
Supply (SC + stay vanes + guide vanes), %	2,0	2,1
Runner, %	3,8	4,0
Draft tube, %	2,1	2.3

Table 2 – Hydraulic efficiency of the hydro turbine Fr 310

	Modification 1 (15 blades)	Modification 2 (17 blades)
Efficiency of the hydro turbine (according to the universal characteristic Fr 310/1107-V-38,8), %	92,4	
Efficiency of the hydro turbine (according to the results of a numerical experiment), %	92	91,6

Analysis of the obtained results of numerical modeling. Analyzing the results of numerical calculation of the flow in the FP of the turbine Fr 310 we can highlight the following points:

- the nature of the flow in all elements of the flow path is virtually the same for both modification 1 and modification 2, only the numerical values of the quantities differ;
- the flow in the spiral case is uniform (Fig. 4, 10);
- the flow around the stay vanes and the guide vanes takes place without significant flow gaps (Fig. 4, 6, 10, 13);
- in the runner in the area of the inlet edge there is a peak of speeds (Fig. 6, 13);
- the minimum values of pressure are observed on the back side of the blade along the output edge (Fig. 11, 12);
- fluid flow in DT is characterized by uneven velocity values (Fig. 7). In the vertical diffuser behind the fairing there is a vortex harness (Fig. 7). In the bend and

in the output diffuser there are so-called stagnant zones, where the flow rate is close to zero (Fig. 7).

If we compare the two modifications, we can conclude that with the same mode, the same elements of the flowing part, the modification with an runner having 15 blades is slightly better in terms of efficiency. For a more accurate comparison, it is necessary to take into account the issues of cavitation and strength, but in this work there was no such task.

Therefore, based on the results of numerical study of the flow in the flow path of the Fr 310 it was found that with increasing number of blades, the hydraulic efficiency of the turbine decreases.

Conclusions. A numerical experiment was performed using the OpenFOAM software package for two modifications of the hydro turbine Fr 310.

After calculating two modifications of the flow path, it was concluded that the best option is modification 1 with the runner having 15 blades.

The comparison was performed exclusively on the efficiency of the turbine. Issues of cavitation and strength were not considered in this work.

Thus, the OpenFOAM software can be used for numerical simulation of the flow in the flow path of the hydro turbine, further analysis of the results and optimization of the elements of the flow path to improve the energy performance of the hydro turbine.

References

1. Drankovskiy V. E., Rezvaya K. S., Krupa E. S. Calculating three-dimensional fluid flow in the spiral casing of the reversible hydraulic machine in turbine mode. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv: NTU "KhPI". 2016. No. 20 (1192). P. 53–57.
2. Khare R., Prasad V., Kumar S. CFD approach for flow characteristics of hydraulic Francis turbine. *International Journal of Engineering Science and Technology.* 2010. Vol. 2 (8). P. 3824–3831.
3. 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
4. Wahidullah H. S., Prasad V. Design and permance analysis of Francis turbine for hydro power station on Kunar river using CFD. *International Journal of Advanced Research.* 2017. No. 5 (5). P. 1004–1012.
5. Pankaj G., Rajeshwer S. Numerical Study of Cavitation in Francis Turbine of a Small Hydro Power Plant. *Journal of Applied Fluid Mechanics.* 2016. No. 9 (1). P. 357–365. doi: 10.18869/ acadpub.jafm.68.224.24080
6. Rezvaya K., Krupa E., Drankovskiy V., Potetenko O., Tynyanova I. The numerical reseach of the flow in the inlet of the high-head hydraulic turbine. *Bulletin of the National Technical University "KhPI". Series: New solution in modern technologies.* Kharkiv: NTU "KhPI". 2017. No. 7 (1229). P. 97–102. doi: 10.20998/2413-4295.2017.07.13
7. Dehkharqani A. S., Cervantes M. J., Aidanpää J. O. Numerical analysis of fluid-added parameters for the torsional vibration of a Kaplan turbine model runner. *Advances in Mechanical Engineering*. 2017. Vol. 9, issue 10. P. 1–10. doi: 10.1177/1687814017732893
8. Brekke H. Design, Performance and Maintenance of Francis Turbines. *Global Journal of Researches in Engineering Mechanical and Mechanics Engineering.* 2013. Vol. 13 (5). P. 28–40.
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. *OpenFOAM. The open source CFD toolbox*. URI: http://www.openfoam.com (дата звернення: 04.03.2020).
11. Zhang H., Zhang L. Numerical simulation of cavitating turbulent flow in a high head Francis turbine at part load operation with OpenFOAM. *Procedia Engineering*. 2012. Vol. 31. P. 156–165. doi: 10.1016/j.proeng.2012.01.1006
12. Крупа Є. С. Чисельне моделювання просторового потоку в підводі осьової поворотно-лопатевої гідротурбіни. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv: NTU "KhPI". 2017. No. 42 (1264). P. 77–83.
13. Krasnopolsky B., Medvedev A. Acceleration of large scale OpenFOAM simulations on distributed systems with multicore cpus and gpus. *Parallel Computing: On the Road to Exascale. Series: Advances in Parallel Computing*. Amsterdam: IOS Press. 2016. Vol. 27. P. 93–102. doi: 10.3233/978-1-61499-621-7-93
14. Кочевский А. Н., Неня В. Г. Современный подход к моделированию и расчету течений жидкости в лопастных гидромашинах. *Вісник Сумського державного університету: Сер.: Технічні науки*. Суми: СумДУ. 2003. № 13 (59). С. 195–210.
15. Миронов К. А., Олексенко Ю. Ю. Применение CFD при проектировании элементов проточной части гидротурбин. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv: NTU "KhPI". 2016. No. 20 (1192). P. 116–121.
16. Nilsson H., Cervantes M. Effect of inlet boundary conditions, on the computed flow in the Turbine-99 draft tube, using OpenFOAM and CFX. *26th IAHR Symposium on Hydraulic Machinery and Systems. IOP Conference. Series: Earth and Environmental Science. Vol. 15*. Bristol: IOP, 2012. P. 1–9. doi: 10.1088/1755-1315/15/3/032002
17. Duan X. H., Kong F. Y., Liu Y. Y., Zhao R. J., Hu Q. L. The numerical simulation based on CFD of hydraulic turbine pump. *IOP Conference Series: Materials Science and Engineering. Vol. 129*. 2016.
18. Elin A., Lugova C., Kolesnik E. Testing of the CFX-5 package on the examples of flow of liquid and gas in the running parts of VNIIAEN specialization pumps: flow modeling in the flow part of the intermediate stage of the multistage centrifugal pump. *Scientific and practical journal "Pumps and equipment".* 2007. Vol. 6 (47). P. 42–46.
19. 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.
20. Bychkov I. M. Verification of the OpenFOAM application package on aerodynamic profile flow problems. *XIX school-seminar "Aerodynamics of Aircraft"*. 2008.

References (transliterated)

1. Drankovskiy V. E., Rezvaya K. S., Krupa E. S. Calculating three-dimensional fluid flow in the spiral casing of the reversible hydraulic machine in turbine mode. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units*. Kharkiv, NTU "KhPI" Publ., 2016, no. 20 (1192), pp. 53–57.
2. Khare R., Prasad V., Kumar S. CFD approach for flow characteristics of hydraulic Francis turbine. *International Journal of Engineering Science and Technology*. 2010, vol. 2 (8), pp. 3824–3831.
3. 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
4. Wahidullah H. S., Prasad V. Design and permance analysis of Francis turbine for hydro power station on Kunar river using CFD. *International Journal of Advanced Research.* 2017, No. 5 (5), pp. 1004–1012.
5. Pankaj G., Rajeshwer S. Numerical Study of Cavitation in Francis Turbine of a Small Hydro Power Plant. *Journal of Applied Fluid Mechanics.* 2016, no. 9 (1), pp. 357–365. doi: 10.18869/ acadpub.jafm.68.224.24080
6. Rezvaya K., Krupa E., Drankovskiy V., Potetenko O., Tynyanova I. The numerical research of the flow in the inlet of the high-head hydraulic turbine. *Bulletin of NTU "KhPI". Series: New solutions in modern technologies.* Kharkiv, NTU "KhPI" Publ., 2017, no. 7 (1229), pp. 97–102. doi:10.20998/2413-4295.2017.07.13

7. Dehkharqani A. S., Cervantes M. J., Aidanpää J. O. Numerical analysis of fluid-added parameters for the torsional vibration of a Kaplan turbine model runner. *Advances in Mechanical Engineering*. 2017, vol. 9, issue 10, pp. 1–10. doi: 10.1177/1687814017732893
8. Brekke H. Design, Performance and Maintenance of Francis Turbines. *Global Journal of Researches in Engineering Mechanical and Mechanics Engineering.* 2013, vol. 13 (5), pp. 28–40.
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. *OpenFOAM. The open source CFD toolbox*. Available at: http://www.openfoam.com (accessed 04.03.2020).
11. Zhang H., Zhang L. Numerical simulation of cavitating turbulent flow in a high head Francis turbine at part load operation with OpenFOAM. *Procedia Engineering*. 2012, vol. 31, pp. 156–165. doi: 10.1016/j.proeng.2012.01.1006.
12. Krupa Ye. S. Chysel'ne modelyuvannya prostorovoho potoku v pidvodi os'ovoyi povorotno-lopatevoyi hidroturbiny [Numerical simulation of the spatial flow in the approach of the Kaplan turbine]. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv, NTU "KhPI" Publ., 2017, no. 42 (1264), pp. 77–83.
13. Krasnopolsky B., Medvedev A. Acceleration of large scale OpenFOAM simulations on distributed systems with multicore cpus and gpus. *Parallel Computing: On the Road to Exascale. Series: Advances in Parallel Computing*. Amsterdam, IOS Press Publ., 2016, vol. 27, pp. 93–102. doi: 10.3233/978-1-61499-621-7-93
14. Kochevskiy A. N., Nenya V. G. Sovremenny podkhod k modelirovaniyu i raschetu techenij zhidkosti v lopastnykh gidromashinakh [Modern approach to modeling and calculating fluid flow in blade hydraulic machines]. *Visnyk Sums'koho derzhavnoho universytetu. Seriya: Tekhnichni nauky* [Sumy State University Bulletin: Technical Sciences Series]. Sumy, SumDU Publ., 2003, no. 13 (59), pp. 195–210.
15. Mironov K. A., Oleksenko Yu. Yu. Primenenie CFD pri proektirovanii elementov protochnoy chasti gidroturbin [Application of CFD in the design of elements of the flow path of hydraulic turbines]. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv, NTU "KhPI" Publ., 2016, no. 20 (1192), pp. 116–121.
16. Nilsson H., Cervantes M. Effect of inlet boundary conditions, on the computed flow in the Turbine-99 draft tube, using OpenFOAM and CFX. *26th IAHR Symposium on Hydraulic Machinery and Systems. IOP Conference. Series: Earth and Environmental Science. Vol. 15.* Bristol, IOP Publ., 2012, pp. 1–9. doi: 10.1088/1755-1315/15/3/032002
17. Duan X. H., Kong F. Y., Liu Y. Y., Zhao R. J., Hu Q. L. The numerical simulation based on CFD of hydraulic turbine pump. *IOP Conference Series: Materials Science and Engineering. Vol. 129*. 2016.
18. Elin A., Lugova C., Kolesnik E. Testing of the CFX-5 package on the examples of flow of liquid and gas in the running parts of VNIIAEN specialization pumps: flow modeling in the flow part of the intermediate stage of the multistage centrifugal pump. *Scientific and practical journal "Pumps and equipment".* 2007, vol. 6 (47), pp. 42–46.
19. 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.
20. Bychkov I. M. Verification of the OpenFOAM application package on aerodynamic profile flow problems. *XIX school-seminar "Aerodynamics of Aircraft"*. 2008.

Received 16.10.2021

Відомості про авторів / Сведения об авторах / About the Authors

Крупа Євгеній Сергійович (***Крупа Евгений Сергеевич***, ***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