UDC 621.224

K. A. MIRONOV, YU. YU. OLEKSENKO, V. K. MIRONOV

STUDY OF THE SPATIAL FLOW IN THE FLOW PART OF THE HIGH-PRESSURE FRANCIS TURBINE

The paper presents some results of a computational study of the spatial turbulent flow of a viscous fluid in the flow part of the high-pressure Francis turbine Fr500, made using the CFX-TASCflow application program package. To improve the energy performance at the preliminary design stage of the turbine, numerical flow simulations should be carried out. This CFD approach reduces costs and time in comparison with the experimental approach and makes it possible to improve and analyze turbine performance and its design before the model is manufactured. The computational complex of programs provides an opportunity to see the picture of pressure distribution, the field of velocity vectors and the movement of fluid particles for substantiation and analysis of results. Numerical modeling of the spatial flow in the flow part of the turbine was carried out to determine changes in the energy characteristics, therefore, the k-ε turbulence model was chosen. As a result of the calculation, the distribution of speeds and pressures in the various elements of the hydraulic turbine was determined at different openings of the guide vane. The analysis of energy losses in the flow part of a Francis turbine: a spiral case, a stator with flat rings, a guide vane, a runner and a draft tube on the optimal operating mode of the hydraulic turbine, as well as an analysis of the effect of opening the guide vane on changes in energy losses in various elements of the flow parts . The results of the computational study confirm that the hydraulic efficiency of a hydraulic turbine largely depends on the losses in the guide vane and the runner, which means it is these elements that should be given the most attention, their design and coordination of the flow in them. The issue of increasing the energy performance of the flow parts of a high-pressure Francis turbine was also considered.

Keywords: runner, spiral case, guide vanes, draft tube, stator, CFD, energy losses.

К. А. МИРОНОВ, Ю. Ю. ОЛЕКСЕНКО, В К. МИРОНОВ

ДОСЛІДЖЕННЯ ПРОСТОРОВОЇ ТЕЧІЇ В ПРОТОЧНІЙ ЧАСТИНІ ВИСОКОНАПІРНОЇ РАДІАЛЬНО-ОСЬОВОЇ ГІДРОТУРБІНИ

В роботі представлені деякі результати розрахункового дослідження просторової турбулентної течії в'язкої рідини в проточній частині високонапірної радіально-осьової гідротурбіни РО500, виконаного за допомогою комплексу прикладних програм CFX-TASCflow. Для поліпшення енергетичних показників на попередньому етапі проектування гідротурбіни проводиться чисельне моделювання потоку. Даний підхід CFD знижує витрати і час в порівнянні з експериментальними підходом і дає можливість удосконалити і аналізувати показники турбіни і її конструкцію до моменту виготовлення моделі. Розрахунковий комплекс програм надає можливість побачити картину розподілу тиску, поле векторів швидкості і руху частинок рідини для обґрунтування та аналізу результатів. Чисельне моделювання просторового потоку в проточній частині гідротурбіни було проведено для визначення зміни енергетичних характеристик, тому була обрана k-ε модель турбулентності. В результаті розрахунку були визначені розподіл швидкостей і тисків в різних елементах гідравлічної турбіни, при різних відкриттях напрямного апарату. Виконано аналіз втрат енергії в проточній частині радіально-осьової гідротурбіни: спіральній камері, статорі з плоскими кільцями, напрямному апараті, робочому колесі і відсмоктуючій трубі на оптимальному режимі роботи гідротурбіни, а також аналіз впливу відкриття напрямного апарату на зміни втрат енергії в різних елементах проточної частини. Наведені результати розрахункового дослідження підтверджують, що гідравлічний коефіцієнт корисної дії гідравлічної турбіни в значній мірі залежить від втрат в напрямному апараті і робочому колесі і означає саме цим елементам варто приділяти найбільші увагу, їх конструкції та узгодженню потоку в них. Також було розглянуто питання підвищення енергетичних показників проточної частини високонапірної радіально-осьової гідротурбіни.

Ключові слова: робоче колесо, спіральна камера, напрямний апарат, відсмоктуюча труба, статор, CFD, енергетичні втрати.

К. А. МИРОНОВ, Ю. Ю. ОЛЕКСЕНКО, В К. МИРОНОВ

ИССЛЕДОВАНИЕ ПРОСТРАНСТВЕННОГО ТЕЧЕНИЯ В ПРОТОЧНОЙ ЧАСТИ ВЫСОКОНАПОРНОЙ РАДИАЛЬНО-ОСЕВОЙ ГИДРОТУРБИНЫ

В работе представлены некоторые результаты расчетного исследования пространственной турбулентного течения вязкой жидкости в проточной части высоконапорной радиально-осевой гидротурбины РО500, выполненного с помощью комплекса прикладных программ CFX-TASCflow. Для улучшения энергетических показателей на предварительном этапе проектирования гидротурбины проводиться численное моделирования потока. Данный подход CFD снижает затраты и время в сравнении с экспериментальными подходом и дает возможность усовершенствовать и анализировать показатели турбины и ее конструкцию до момента изготовления модели. Расчетный комплекс программ предоставляет возможность увидеть картину распределения давления, поле векторов скорости и движения частиц жидкости для обоснования и анализа результатов. Численное моделирование пространственного потока в проточной части гидротурбины было проведено для определения изменения энергетических характеристик, поэтому была выбрана k–ε модель турбулентности. В результате расчета были определены распределение скоростей и давлений в различных элементах гидравлической турбины, при различных открытиях направляющего аппарата. Выполнен анализ потерь энергии в проточной части радиально-осевой гидротурбины: спиральной камере, статоре с плоскими кольцами, направляющем аппарате, рабочем колесе и отсасывающей трубе на оптимальном режиме работы гидротурбины, а также анализ влияния открытия направляющего аппарата на изменения потерь энергии в различных элементах проточной части. Приведенные результаты расчетного исследования подтверждают, что гидравлический коэффициент полезного действия гидравлической турбины в значительной мере зависит от потерь в направляющем аппарате и рабочем колесе и значит именно этим элементам стоит уделять наибольшие внимание, их конструкции и согласованию потока в них. Также был рассмотрен вопрос повышения энергетических показателей проточной части высоконапорной радиально-осевой гидротурбины.

Ключевые слова: рабочее колесо, спиральная камера, направляющий аппарат, отсасывающая труба, статор, CFD, энергетические потери.

Introduction. The process of developing new flow parts of hydraulic turbines consists of several successive stages, including:

- selection of design parameters of hydro turbine for given conditions of HPP and design of elements of the flow parts (as a rule, several options are considered, differing by calculation methods, geometrical, kinematic and other parameters);

- computational study of flow characteristics, force and torque characteristics, energy losses in the elements of the flow parts and the determination of the best options for experimental studies;
- an experimental study of model blocks of the hydro turbine at energy-cavitational stands, and the size of the model, test conditions and parameters of the stands must meet the requirements of the international standard IEC.

Based on the results of experimental and computational studies of the model, characteristics are calculated and guarantees of full-scale hydro turbine are issued.

Literature review. Improving the flow parts of the hydro turbine is based on conducting extensive numerical investigations, during which the search for the most rational variants. The basis of this research is the multivariate numerical analysis of the influence of geometric and operating parameters on the energy performance of the hydro turbine. To solve this problem, both simplified flow models [1] and a more complex kinematic flow description using quasi-three-dimensional and three-dimensional fluid flow models [2] are used. The use of quasi-three-dimensional and three-dimensional methods allows, for given specific conditions, to find the distribution of the parameters of the spatial flow and more accurately find the energy loss. In modern software products, mathematical models of liquid and gas flows are used, in a three-dimensional formulation, describing various models of turbulence [3].

Currently, for modeling and calculating fluid flows in hydraulic machines, modern commercial software products are used: CFX-TASCflow, Fluent, etc., which allow to adequately simulate complex physical effects [3, 4].

Research methodology. The traditional practice of predicting the effectiveness of the hydro turbine relies either on a theoretical approach or on experimental testing of a reduced model. The theoretical approach allows us to determine the value of efficiency, but it is difficult to determine the cause of problems and disrepairs. Conversely, testing the model is costly and time consuming.

So far, the theoretical analysis of the flow in the individual the hydro turbine element is used to assess the losses and flow studies. Modern CFD application software packages are becoming a cost-effective tool used to obtain detailed information about the properties of a stream in the hydro turbine, taking into account the interaction of its various elements. CFD is widely used by designers and researchers to optimize the project, as well as to predict the work of Francis turbine (Fr) as a whole. Three-dimensional modeling of a viscous flow makes it possible to determine the pressure and velocity distribution in the flow parts of the hydro turbine.

CFD is a tool for the numerical solution of complex incomplete differential equations, known as the governing equations of fluid motion, used to describe in detail the flow motion in a given flow field [5, 6]. The physical laws governing fluid flow can be described by means of an exact mathematical description that forms the basis of any analysis. Known numerical methods used for sampling in CFD are the finite difference method (FDM), the finite volume method (FVM), and the finite element method (FEM) [7].

The most important reason for increasing the use of CFD to solve most movement of liquid problems is that this method is much cheaper than experimental testing of the hydro turbine model. In short, CFD has the following advantages:

1) Costs and time spent on the creation of the project and its development is much less.

2) CFD allows you to simulate the conditions movement of liquid, which are difficult to reproduce in experimental testing of the model.

3) CFD allows you to get more detailed and complete information about the flow.

4) Does not require scaling.

Results. This article presents the results of a design study of fluid flow in the spiral case and in the area of the stator and the guide vanes arrays of a high-pressure Francis turbine (Fr) Fr500, performed using the CFX-TASCflow program. The picture of the spatial flow in characteristic sections of the flow part is considered.

Numerical modeling of the spatial flow in the flow parts of the hydro turbine was carried out to determine the change in the energy characteristics; therefore the *k-ε* model of turbulence was chosen. This model was developed in the 70s [8, 9]. There are also modifications.

When using this model, the system of equations of fluid motion is supplemented by two differential equations describing the transfer, respectively, of the kinetic turbulence energy *k* and dissipation rate *ε* [10, 11].

k is the kinetic energy of turbulence, defined as the dispersion of velocity fluctuations; *ε* is the dissipation rate. We write two equations for k and *ε*:

$$\frac{\partial pk}{\partial t} + \nabla(pUk) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right] + P_k - p\varepsilon, \qquad (1)$$

$$\frac{\partial p\varepsilon}{\partial t} + \nabla(pU\varepsilon) = \nabla\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right] + \frac{\varepsilon}{k\left(C_{S1}P\right)_k} - C_{S2}p\varepsilon, \qquad (2)$$

where $\mu_t = C_\mu p \frac{k^2}{\varepsilon}$, $C_\mu = 0,09$, $C_{S1} = 1,44$, $C_{S2} = 1,92$, $\sigma_k = 1,0$, $\sigma_\varepsilon = 1,3$, P_k – takes into account the occurrence of turbulence due to viscous friction forces and is determined

$$P_k = \mu_t \nabla U \left(\nabla U + \nabla U^T \right) - \frac{2}{3} \nabla U \left(3\mu_t \nabla U + pk \right) + P_{kb} \cdot$$

Calculations show that near the solid paries there is a very sharp change in the parameters k and ε. To properly resolve these changes, we must use a very dense computational grid. Instead, an approach is often used in which a small area is allocated to the paries, in which the numerical solution of equations (1) and (2) is not performed, and instead, the desired parameters are calculated using algebraic formulas describing typical parietal layers.

When designing Francis turbine, before building a geometric model, air defense, it is necessary to carry out their coordination with each other [6, 9].

The assembly of the geometrical model obtained as a result of the flow parts design of a high-pressure Francis turbine is shown in Fig. 1.

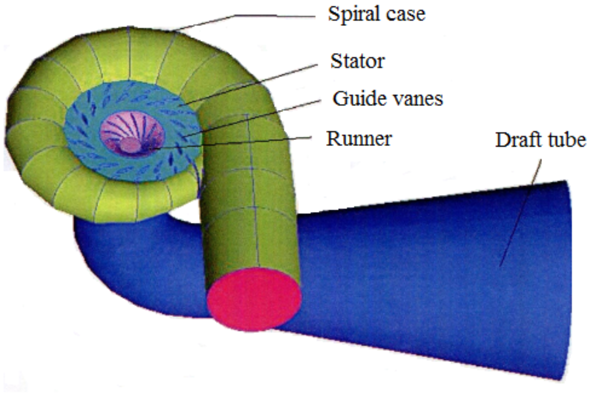

Fig. 1. Assembling a geometric model Francis turbine

As a result of the calculation, we determined the distribution of velocities and pressures in various elements of the hydro turbine, at various discoveries of guide vanes. The figures show graphs for the optimal mode ($n_I' = 65$ об/мин; $Q_I' = 154$ л/с; $\alpha_2 = 15^0$), which give an idea of the change in pressures and velocities within the considered area of flow.

Numerical simulation of the flow in the flow parts of the hydro turbine Fr500 was carried out for the design area, including the intervene channel formed by stator columns, shoulder guide vanes, runner blades and draft tube for a model with a diameter runner D_1=500 mm.

The obtained results of the calculation of the spatial flow are presented in the form of averaged values of the total and static pressures of flow, averaged flow angles in relative and absolute motion, and values of losses in individual elements of the flow parts. For runner at a mode point with minimal total losses close to optimal, a static and total pressure field in the computational domain, the distribution of the components of the meridional and peripheral components of the full velocity before entering and output the runner, as well as the trajectory of fluid particles in draft tube

The flow of fluid in the spiral case has a complex spatial character, the pressure change in the spiral case and the stator is shown in Fig. 2.

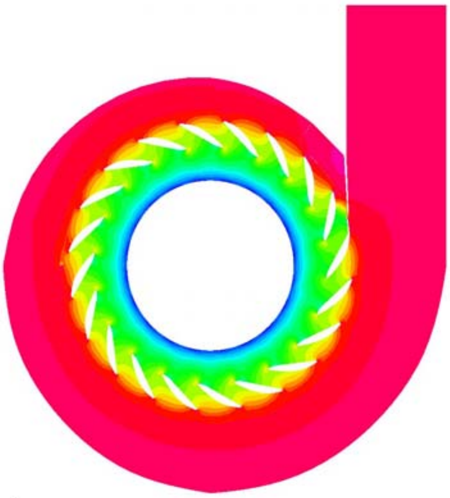

Fig. 2. Pressure distribution in the spiral case

From Fig. 2 that the pressure is greater at the outer paries of the spiral case and decreases towards the exit from the spiral. An increase in pressure is also observed at the end of the spiral.

The flow of fluid in the area of the stator columns shoulder guide vanes and runner is shown in Fig. 3, and the distribution of total pressure is shown in Fig. 4.

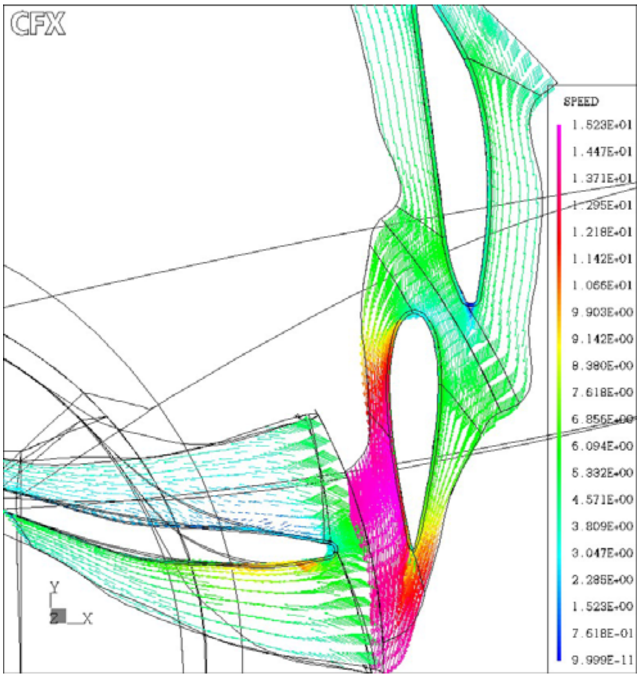

Fig. 3. The field of the vectors of the velocity of the spatial flow of fluid in the region of the stator columns, the shoulder guide vanes and runner in the optimal mode

The data obtained (see Fig. 3) show that the geometry of the blade system of the runner in the area of the entrance edge is not consistent with the flow angle behind guide vanes, which means the presence of impact losses at the entrance edge of the runner, therefore further work will address issues related to the modification of the input element of the blade runner.

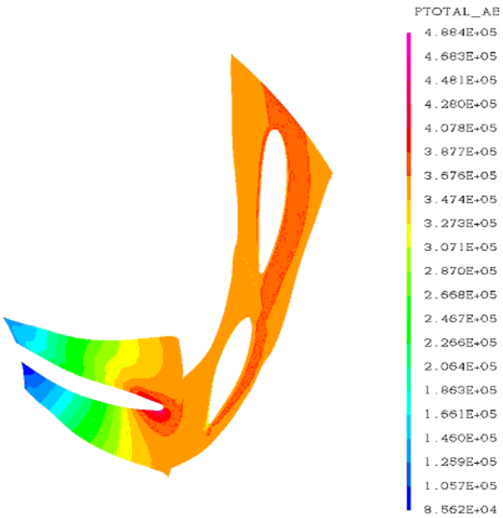

Fig. 4. Isolines of total pressure in the middle part of blade systems

The pressure continuously decreases along the meridional direction from the entrance to the stator to the exit from runner, as can be seen from Fig. 4. The pressure becomes negative at the exit from the runner due to the influence of the draft tube.

In Fig. 5 shows the trajectories of the movement of fluid particles in the draft tube at the optimal mode based on the calculation of the spatial flow.

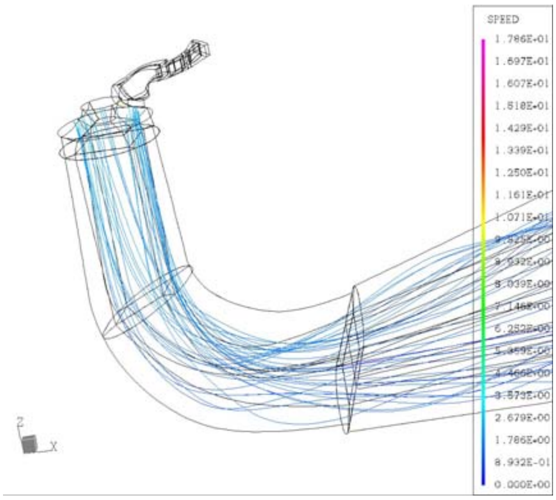

Fig. 5. The trajectories of the movement of fluid particles in the draft tube

The location of the current lines in the draft tube Fig. 5 shows that the speed decreases from the inlet to the outlet of the draft tube, due to which the kinetic energy is converted into pressure energy. There is a gradual drop in pressure from inlet to outlet along the suction and pressure side of the runner blades.

The pattern of fluid motion also shows the orderly nature of the flow in the draft tube (secondary flows in the draft tube are weak). This improves the recovery of static pressure in the draft tube and does not lead to additional losses. The reason for the favorable flow in the peripheral region of the draft tube is a sufficient swirl of flow beyond the runner.

The obtained calculated data correspond to the previously known experimental recommendations on the positive effect of a small swirl flow at the entrance to the draft tube on the amount of losses in it [6, 13] and on the optimal, from the point of view of minimizing inductive losses, distribution pattern of the tangential velocity component an increase in its values in the peripheral region [13].

The results of the calculation of the energy loss (at the optimal mode) in the flow parts of a high-pressure Francis turbine Fr500 are shown in the table.

Table 1 – The results of the calculation of the energy loss in the flow parts of a high-pressure Francis turbine

Turbine type	Energy losses, %				Σ
	Spiral case + Stator	Guide vanes	Runner	Draft tube	
Fr500	0,77	2,5	1,66	0,2	5,13

As can be seen, from the results of the calculation (see Table 1) of the energy characteristics, the greatest energy losses in the high-pressure Francis turbine are accounted for to a greater extent in the guide vanes and the runner. It is of interest to consider in more detail the question of the effect of the guide vanes (the shape of the blade profile, the law of profile thickness variation, the axis of rotation, etc.) on the formation of the energy characteristics of the hydro turbine. To reduce losses in the runner, it is necessary to carefully coordinate the flow coming from the blades of the guide vanes with the geometry of the input element of the runner.

Conclusion. 1. A pilot approach for evaluating the performance of a hydro turbine is expensive and time consuming. On the other hand, CFD approach is fast and cost effective.

2. Losses in various elements of the flow parts depend on the discovery of the guide vanes. The losses in the spiral case and the stator decrease, while the losses in the guide vanes and the draft tube increase with the opening of the guide vanes. Losses in the runner are minimal at optimum performance. Hydraulic efficiency of a hydro turbine largely depends on the losses in the guide vanes and the runner.

3. Analysis of losses in the supply shows that the greatest energy losses occur in the guide vanes. Losses in the spiral case and in the stator constitute no more than one third of the losses of the guide vanes. The total losses in the supply change smoothly with increasing flow and have a minimum, in absolute values up to 65 % of all hydraulic losses in the flow parts, which is a characteristic feature of high-pressure Francis turbine.

References

1. Колычев В. А. *Кинематические характеристики потока в лопастных гидромашинах*. Киев: ИСИО, 1995. 272 с.
2. Барлит В. В. *Современные гидродинамические методы расчета лопастных систем и САПР гидромашин.* Киев: УМК МО Украины, 1992. 180 с.

3. Wilcox David C. *Turbulence Modeling for CFD.* DCW Industries, Inc., 1993. 460 p.
4. Черный С. Г., Чирков Д. В., Лапин В .Н. *Численное моделирование течений в турбомашинах.* Новосибирск: Наука, 2006. 202 с.
5. Chung T. J. *Computational fluid dynamics*. Cambridge university press, 2002. 1012 p.
6. Барлит В. В., Миронов К. А., Власенко А. В., Яковлева Л. К. *Расчет и проектирование проточной части реактивных гидротурбин на основе численного моделирования рабочего процесса*. Харьков: НТУ "ХПИ", 2008. 216 с.
7. Миронов К. А., Олексенко Ю. Ю. Використання CFD для розрахунку спіральної камери та колон статора висконапірної радіально-осьової гідротурбіни. *Вестник Нац. техн. ун-та "ХПИ": сб. науч. тр. Серія: Гідравлічні машини та гідроагрегати. Харків: НТУ «ХПИ».2018.* № 17(1293). С. 50–53.
8. Launder B. E., Spalding D. B. The Numerical Computation of Turbulent Flows. *Comp. Meth. Appl. Mech. Eng.* 1974. Vol. 3. С. 269–289.
9. Миронов К. А., Олексенко Ю. Ю. Применение CFD при проектировании элементов проточной части гидротурбин. *Вестник Нац. техн. ун-та "ХПИ": сб. науч. тр. Серія: Гідравлічні машини та гідроагрегати. Харків: НТУ «ХПИ».*2016. № 20(1192). С. 116–121.
10. *Ansys 16.0 Release Documentation, Theory and Modelling Guide.* ANSYS Inc.: Canonsburg, PA, USA. 2015.
11. Versteeg H., Malalasekera W. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method (2nd Edition).* Pearson Education Limited, 1995. 257 p.
12. Колычев В. А., Миронов К. А., Тыньянова И. И. Согласование элементов проточной части при проектировании радиально-осевых гидротурбин. *Проблемы машиностроения.* 2009. Т. 12, № 5. С. 3–8.
13. Этинберг И. Э., Раухман Б. С. *Гидродинамика гидравлических турбин*. Ленинград: Машиностроение, 1978. 280 с.

References (transliterated)

1. Kolychev V. A. *Kinematicheskie harakteristiki potoka v lopastnyh gidromashinah* [Kinematic characteristics of flow in blade hydraulic machines]. Kiev, ISIO Publ., 1995. 272 p.
2. Barlit V. V. *Sovremennye gidrodinamicheskie metody rascheta lopastnyh sistem i SAPR gidromashin* [Modern hydrodynamic methods for calculating blade systems and CAD of hydraulic machines]. Kiev, UMK MO Ukrainy Publ., 1992. 180 p.
3. Wilcox David C. *Turbulence Modeling for CFD.* DCW Industries, Inc. Publ., 1993. 460 p.
4. Chernyj S. G., Chirkov D. V., Lapin V. N. [et al.]. *Chislennoe modelirovanie techenij v turbomashinah* [Numerical simulation of currents in turbomachines]. Novosibirsk, Nauka Publ., 2006. 202 p.
5. Chung T. J. *Computational fluid dynamics*. Cambridge university press Publ., 2002. 1012 p.
6. Barlit V. V., Mironov K. A., Vlasenko A. V., Jakovleva L. K. *Raschet i proektirovanie protochnoj chasti reaktivnyh gidroturbin na osnove chislennogo modelirovanija rabochego processa* [Calculation and design of the flow parts of jet turbines based on numerical simulation of the workflow]. Kharkov, NTU "HPI" Publ., 2008. 216 p.
7. Myronov K. A., Oleksenko Yu. Yu. *Vykorystannia CFD dlia rozrakhunku spiralnoi kamery ta kolon statora vysokonapirnoi radialno-osovoi hidroturbiny* [Using CFD to calculate the spiral case and stator columns of the high-pressure Francis turbine]. *Vestnik Nats. tekhn. un-ta "KhPI": sb.nauch. tr. Seriya: Gidravlichni mashini ta gidroagregati* [Bulletin of the National Technical University "KhPI": a collection of scientific papers. Series: Hydraulic machines and hydraulic units] Kharkov, NTU"KhPI" Publ., 2018, no. 17(1293), pp. 50–53.
8. Launder B. E., Spalding D. B. The Numerical Computation of Turbulent Flows. *Comp. Meth. Appl. Mech. Eng.* 1974, vol. 3, pp. 269–289.
9. Myronov K. A., Oleksenko Yu. Yu. *Primenenie CFD pri proektirovanii jelementov protochnoj chasti gidroturbin* [The use of CFD in the design of elements of the flow part of hydraulic turbines]. *Vestnik Nats. tekhn. un-ta "KhPI": sb.nauch. tr. Seriya: Gidravlichni mashini ta gidroagregati* [Bulletin of the National Technical University "KhPI": a collection of scientific papers. Series: Hydraulic machines and hydraulic units] Kharkov, NTU"KhPI" Publ., 2016, no. 20(1192), pp. 116–121.
10. *Ansys 16.0 Release Documentation, Theory and Modelling Guide.* ANSYS Inc.: Canonsburg, PA, USA, 2015.
11. Versteeg H., Malalasekera W. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method (2nd Edition).* Pearson Education Limited Publ., 1995. 257 p.
12. Kolychev V. A., Mironov K. A., Tyn'janova I. I. Soglasovanie jelementov protochnoj chasti pri proektirovanii radial'no-osevyh gidroturbin [Coordination of elements of the flow part in the design of Francis turbines]. *Problem mashinostroenija.* 2009, vol. 12, no. 5. pp. 3–8.
13. Jetinberg I. Je., Rauhman B. S. *Gidrodinamika gidravlicheskih turbin* [Hydrodynamics of hydraulic turbines]. Lenyngrad: Mashinostroenie Publ., 1978. 280 p

Received 09.11.2018

Відомості про авторів / Сведения об авторах / About the Authors

Миронов Костянтин Анатолійович (Миронов Константин Анатольевич, Mironov Konstantin Anatolievich) – 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/0000-0002-6034-410X; e-mail: cosmir@i.ua

Олексенко Юлія Юріївна (***Олексенко Юлия Юрьевна***, ***Oleksenko Yuliia Yuriivna***) – Національний технічний університет «Харківський політехнічний інститут», аспірант кафедри «Гідравлічні машини ім. Г.Ф. Проскури»; м. Харків, Україна; ORCID: https://orcid.org/0000-0003-4467-7833; e-mail: yuliayo@ukr.net

Миронов Вадим Костянтинович (Миронов Вадим Константинович, Mironov Vadim Konstantinovich)– Національний технічний університет «Харківський політехнічний інститут», студент кафедри «Технологія жирів і продуктів бродіння»; м. Харків, Україна; e-mail: vadiki1200@gmail.com.