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

UDC 621.224 **doi: 10.20998/2411-3441.2019.1.11**

***K. A. MIRONOV, YU. YU. OLEKSENKO***

# RESEARCH OF FLUID FLOW IN TWO-DIMENSIONAL AND THREE-DIMENSIONAL FORMULATION IN THE FLOW PART OF A HIGH-PRESSURE FRANCIS TURBINE

The paper presents some results of a computational study of the spatial flow of a viscous fluid in a high-pressure Francis turbine Fr500 (in the optimal mode). To improve the energy performance at the preliminary design stage of the turbine, numerical flow simulations should be carried out. The difficulty of solving the problem posed is due both to the complex spatial geometry of the blade system of the runner and the varying degree of influence of the working bodies on the formation of energy characteristics. 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, this model is the most successful model of first-level turbulence of the circuit. 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. Analysis of the energy loss in the flow part of the Francis turbine was carried out using programs for calculating fluid flow in two-dimensional and three-dimensional formulation. The obtained calculated data correspond to the previously known experimental recommendations for high-pressure Francis turbine. The issues of increasing the energy performance of a projected high-pressure Francis turbine were considered.

**Keywords:** runner, spiral case, guide vanes, draft tube, stator, CFD, flow part, Francis turbine, energy losses.

***К. А. МИРОНОВ, Ю. Ю. ОЛЕКСЕНКО***

## ДОСЛІДЖЕННЯ ПОТОКУ РІДИНИ В ДВОВИМІРНІЙ І ТРИВИМІРНІЙ ПОСТАНОВЦІ 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. Для поліпшення енергетичних показників на попередньому етапі проектування гідротурбіни проводиться чисельне моделювання потоку. Складність вирішення поставленого завдання обумовлена як складної просторової геометрією лопатевої системи робочого колеса, так і різним ступенем впливу робочих органів на формування енергетичних характеристик. 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, дана модель є найбільш вдалою моделлю турбулентності першого рівня замикання. 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. Аналіз втрат енергії в проточній частині радіально-осьової гідротурбіни був проведений з використанням програм для розрахунку течії рідини в двовимірній і тривимірній постановці. Отримані розрахункові дані відповідають відомим раніше експериментальним рекомендаціям для високонапірної радіально-осьової гідротурбіни. Були розглянуті питання підвищення енергетичних показників спроектованої високонапірної радіально-осьової гідротурбіни.

**Ключові слова:** робоче колесо, спіральна камера, напрямний апарат, відсмоктуюча труба, статор, CFD, проточна частина, радіально-осьова гідротурбіна, енергетичні втрати.

***К. А. МИРОНОВ, Ю. Ю. ОЛЕКСЕНКО***

## ИССЛЕДОВАНИЕ ТЕЧЕНИЯ ЖИДКОСТИ В ДВУМЕРНОЙ И ТРЕХМЕРНОЙ ПОСТАНОВКЕ 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. Для улучшения энергетических показателей на предварительном этапе проектирования гидротурбины проводиться численное моделирования потока. Трудность решения поставленной задачи обусловлена как сложной пространственной геометрией лопастной системы рабочего колеса, так и различной степенью влияния рабочих органов на формирование энергетических характеристик. 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, данная модель является наиболее удачной моделью турбулентности первого уровня замыкания. 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. Анализ потерь энергии в проточной части радиально-осевой гидротурбины был проведен с использованием программ для расчета течения жидкости в двумерной и трехмерной постановке. Полученные расчетные данные соответствуют известным ранее экспериментальным рекомендациям для высоконапорной радиально-осевой гидротурбины. 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, проточная часть, радиально-осевая гидротурбина, энергетические потери.

**Introduction.** The generally accepted approach to improving the flow parts of hydro turbines is to form the geometry of the hydro turbine by introducing changes in the original version, obtained as a result of an approximate solution of the inverse problem, or, in adopted as an analogue. Comparison of design options is based on the estimated assessment of their kinematic and energy characteristics. Finding the best option makes it extremely difficult to improve the flow path, since such an approach requires going through a significant number of geometric parameters and their combinations. The difficulty of solving the problem posed is due both to the complex spatial geometry of the runner blade system and the varying degree of influence of the working bodies on the formation of energy characteristics [1, 2].

The lack of methods for coordinating the elements of the flow part in the process of its formation on the basis of solving the direct problem greatly complicates the process of improving the flow part and increases the amount of research and design work.

When designing the flow part of the turbine using the calculated and experimental research methods. Recently, in order to reduce the amount of physical experiment, great attention has been given to a numerical experiment. This allows you to reduce the time and cost of design work, which leads to the comprehensive introduction of automated hydro turbine design systems into engineering practice.

Strengthening the role of the numerical experiment became possible in connection with the development of more advanced mathematical models of flow, hydrodynamic methods for designing the flow part and flow calculation, as well as numerical methods and algorithms.

**Literature review.** In order to ensure high energy-cavitational parameters of the flow part of the hydro turbine, it is necessary to conduct a comprehensive hydrodynamic analysis of the flow part using modern CFD application software packages. These packages allow us to calculate the viscous turbulent flow in the cavity of a hydro turbine of any complexity [3–7].

Along with the development of workflow modeling methods that use the results of solving a three-dimensional viscous flow problem, methods for calculating energy characteristics based on simplified flow models are widely used.

The use of simplified models makes it possible at the initial design stages (during the design of the flow part) to calculate the parameters of the optimal mode, determine the kinematic parameters of the flow at the inlet and outlet of the runner and determine the value of the energy loss in this mode [1, 8].

These workflow models are used in solving problems of selection and optimization of the main parameters of hydro turbine [1, 2].

They do not require flow calculations and therefore can be used in the initial design stages in the absence of complete information about the geometry of the blade systems.

**Research methodology.** The flow simulation in the hydraulic machine can be carried out in various approximations. One of the most common and effective approaches is the stationary cyclic statement, in which it is assumed that the currents in all interscapular channel of the guide vane and in the inter-blade channels of the runner are the same [3]. In this case, the calculation is carried out only in one of the channels of the guide vane and the runner, and on the side borders of the channels the conditions for the periodicity of the flow are set. To transfer flow parameters from rotating segments to fixed and vice versa, their values are averaged in the circumferential direction. Such an approach significantly saves computational resources, but it does not make it possible to take into account the circular irregularity of the flow and the non-stationary effects associated with it.

In each element of the flow part of the hydraulic turbine in the flow is dominated by physical processes characteristic of this element. Accordingly, it is necessary to choose suitable models for describing the currents in them. On the one hand, the model should display the main features of the flow, and on the other – be economical. Thus, the main role in the runner of the hydro turbine is played by the process of transferring the torque to the runner by the fluid [3]. This process is quite accurately described by the stationary model of an inviscid fluid.

Viscous properties of the fluid have a significant influence on the energy loss in the draft tube. The dominant role is played by viscosity in the mechanism of formation of the precessing vortex bundle for the runner, which has a significant impact on the work of the entire hydro turbine. For an adequate description of the flow in the draft tube, an effective model of turbulence is required [3, 9].

**Results.** The article presents the results of a computational study of fluid flow in a spiral case and in the area of stator grids and guide vane of the high-pressure Francis turbine Fr500, performed using the CFX-TASCflow program [10–12] and the model developed at the hydraulic machines department [13, 14].

Numerical modeling of the spatial flow in the flow part of the hydro turbine was carried out to determine the change in energy characteristics, therefore the k - ε model of turbulence was chosen, this model is the most successful model of first-level turbulence of the circuit [8–10]. To describe the turbulent quantities, it uses a system of two nonlinear diffusion equations - for the mass density of turbulent energy k and the dissipation rate of turbulent energy ε.

This model was developed in the 70s. 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 energy of turbulence k and dissipation rate ε [2, 15–17].

Calculations show that near the solid walls there is a very sharp change in the parameters k and ε. For the proper resolution of these changes, it is necessary to use a very dense computational grid. Instead, an approach is often used in which a small area is allocated to the wall, in which the numerical solution of equations is not performed, but instead the desired parameters are calculated using algebraic formulas describing typical wall layers.

When designing a Francis turbine, before building a geometric model, in order to reduce the search for possible options for the geometry of the flow part elements, it is necessary to reconcile them with each other [6, 14].

A schematic of the model of high-pressure Francis turbine is shown in Fig. 1.

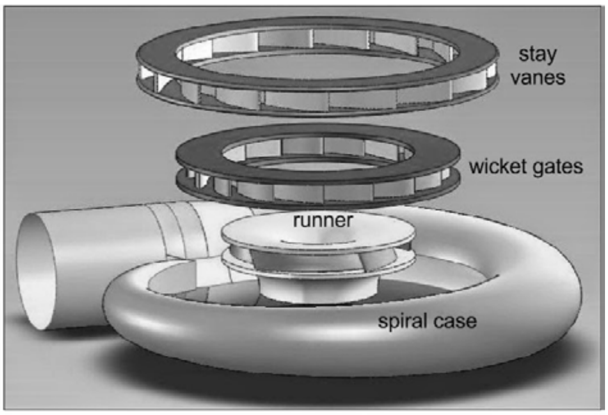



Fig. 1. A schematic of the model of high-pressure Francis turbine

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 (mode with maximum efficiency), 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 $D1$ = 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 area of the stator columns, blades of the guide vane and the runner is shown in Fig. 2, and the distribution of total pressure is shown in Fig. 3.

The data obtained (see Fig. 2) show that the geometry of the runner blade system in the area of the inlet edge is not consistent with the flow angle behind the guide vane, which means the presence of impact losses at the inlet edge of the runner, therefore in further work we will consider issues related to the modification of the inlet element of the runner blade.

The pressure continuously decreases along the meridional direction from the entrance to the stator to the outlet from the runner, as can be seen from Fig. 3. The pressure becomes negative at the outlet from the runner due to the influence of the draft tube.

In Fig. 4 shows the trajectories of the movement of the fluid particles in the draft tube (when the fluid flow from the runner falls out) at the optimum mode based on the calculation of the spatial flow.

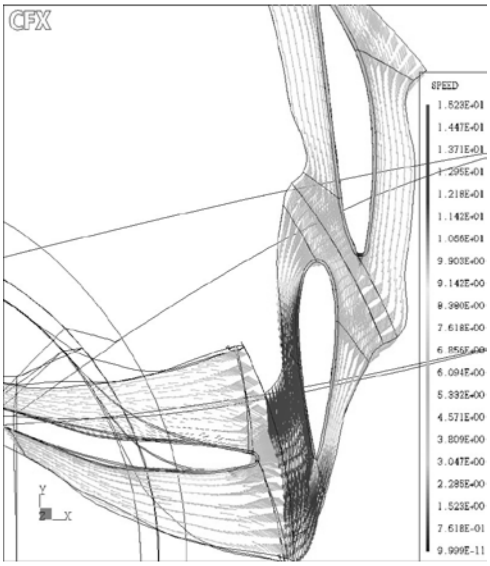



Fig. 2. The field of the vectors of the velocity of the spatial flow of fluid in the region of the stator columns, the blade guide vanes and runner in the optimal mode

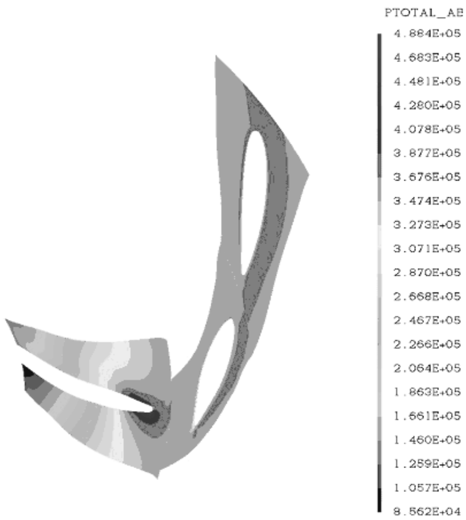
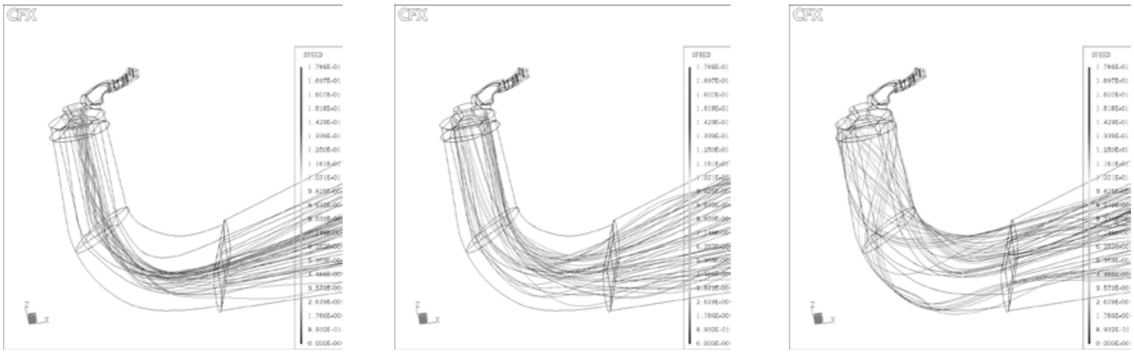



Fig. 3. Isolines of total pressure in the blade systems

The location of the current lines in the draft tube Fig. 4 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.



*a* *b* *c*

Fig. 4. The trajectories of the movement of fluid particles in the draft tube:
*a* – upper rim; *b* – middle; *c* – lower rim

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 [7, 18–20] 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.

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 1.

**Conclusion.** 1. To reduce the amount of physical experiment, it is necessary to pay more attention to the numerical experiment. This will reduce the time and cost of design work.

2. Considered in detail the nature of the movement of fluid in the flow part high-pressure hydro turbine.

3. The results of the calculation optimal mode of the hydro turbine using two-dimensional and three-dimensional flow models are given, the obtained data are in good agreement with each other.

4. To improve the energy performance of high-pressure Francis turbine, it is necessary to study in more detail the effect of the geometry of the guide vane on the formation of losses in the hydro turbine.

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

| Turbine type | Calculation program | Energy losses, % | | | | Σ |
|---|---|---|---|---|---|---|
| | | Spiral case + Stator | Guide vane | Runner | Draft tube | |
| Fr500 | Two-dimensional model | 0,6 | 3,02 | 1,61 | | 5,23 |
| | 3D model | 0,77 | 2,5 | 1,66 | 0,2 | 5,13 |


**References**

1. Колычев В. А. *Кинематические характеристики потока в лопастных гидромашинах*. Киев: ИСИО, 1995. 272 с.
2. Колычев В. А., Тыньянова И. И., Миронов К. А. Моделирование энергетических характеристик гидротурбин на начальном этапе проектирования. *Восточно-европейский журнал передовых технологий.* 2010. Т. 43, № 1/6. С. 27–38.
3. Черный С. Г., Чирков Д. В., Лапин В .Н. *Численное моделирование течений в турбомашинах.* Новосибирск: Наука, 2006. 202 с.
4. Chung T. J. *Computational fluid dynamics*. Cambridge: Cambridge university press, 2002. 1012 c.
5. Minkowycz W. J. Sparrow E. M., Murthy J. Y. *Handbook of Numerical Heat Transfer*. Wiley, 2006. 984 c.
6. Paul G. Tucker. *Computation of Unsteady Internal Flows – Fundamental Methods with Case Studies*. New York: Springer US, 2001. 376 c.
7. 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.
8. 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.
9. Лапин В. Н., Черный С. Г., Скороспелов В. А., Турук П. А. Проблемы моделирования течений в турбомашинах. *Вестник Казахского Нац. ун-та им. аль-Фараби. Сер.: Математика, механика, информатика.* Алматы: КазНУ. 2004. Т. 42, вып. 3. С. 57–66.
10. Сухоребрый П. Н, Барлит В. В., Дранковский В. Э., Рао В. С., Харвани Л. К. Характеристики пространственного турбулентного потока и потери энергии в элементах проточной части гидротурбины РО500. *Проблемы машиностроения.* 2004. Т. 7, № 3. С. 13–20.
11. ANSYS. *Ansys 16.0 Release Documentation, Theory and Modelling Guide.* ANSYS Inc.: Canonsburg, PA, USA, 2015.
12. Jošt D., Škerlavaj A., Morgut M., Mežnar P., Nobile E. Numerical simulation of flow in a high head Francis turbine with prediction of efficiency, rotor stator interaction and vortex structures in the draft tube. *Journal of Physics: Conference Series.* 2015. Vol. 579.
13. Колычев В. А., Миронов К. А, Тыньянова И. И. Расчет и анализ баланса потерь энергии в высоконапорной радиально-осевой гидравлической турбине. *Східно-Європейський журнал передових технологій*. 2005. Т. 13, № 1/2. С. 95–106.
14. Колычев В. А., Дранковский В. Э. *Расчет гидродинамических характеристик направляющих аппаратов гидротурбины: учебн. пособ.* Харьков: НТУ «ХПИ», 2002. 268 с.
15. Daneshkah K., Zangeneh M. Parametric design of a Francis turbine runner by means of a three-dimensional inverse design method. *25-th IAHR Symposium on Hydraulic Machinery Systems. Vol. 12.* 2010.

16. 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.
17. Юн А. А., Крылов Б. А. *Расчет и моделирование турбулентных течений с теплообменом, смешением, химическими реакциями и двухфазных течений в программном комплексе Fastest-3D: учебн. пособ.* Москва: МАИ, 2007. 116 с.
18. Ayli E., Kaplan A., Cetinturk H. CFD analysis of 3D flow for 1.4 MW Francis turbine and model turbine manufacturing. *ASME 2015 International Design Engineering Technical Conferences and Computers and Information in Engineering Conference. Vol. 1A. (2–5 August 2015, Boston, Massachusetts, USA).* Boston: ASME, 2015.
19. Kurosawa S., Lim S. M., Enomoto Y. Virtual model test for a Francis turbine. *IOP Conference Series: Earth and Environmental Science*. 2010. Vol. 12, no. 1.
20. Ayli E., Celebioglu K., Aradag S. Determination and generalization of the effects of design parameters on Francis turbine runner performance. *Engineering Applications of Computational Fluid Mechanics*. 2016. Vol. 10:1. P. 545–564.

**References (transliterated)**

1. Kolychev V. A. *Kinematicheskie kharakteristiki potoka v lopastnykh gidromashinakh* [Kinematic characteristics of flow in blade hydraulic machines]. Kiev, ISIO Publ., 1995. 272 p.
2. Kolychev V. A., Tyn'janova I. I., Mironov K. A. Modelirovanie energeticheskikh kharakteristik gidroturbin na nachal'nom etape proektirovaniya [Modeling the energy characteristics of hydroturbines at the initial design stage]. *Vostochno-evropeyskiy zhurnal peredovykh tekhnologiy.* 2010, vol. 43, no. 1/6, pp. 27–38.
3. Chernyj S. G., Chirkov D. V., Lapin V .N. *Chislennoe modelirovanie techeniy v turbomashinakh* [Numerical simulation of currents in turbomachines]. Novosibirsk, Nauka Publ., 2006. 202 p.
4. Chung T. J. *Computational fluid dynamics*. Cambridge: Cambridge university press Publ., 2002. 1012 p.
5. Minkowycz W. J. Sparrow E. M., Murthy J. Y. *Handbook of Numerical Heat Transfer*. Wiley, 2006. 984 p.
6. Paul G. Tucker. *Computation of Unsteady Internal Flows – Fundamental Methods with Case Studies*. New York, Springer US Publ., 2001. 376 p.
7. Myronov K. A., Oleksenko Yu. Yu. Primenenie CFD pri proektirovanii elementov protochnoy chasti gidroturbin [The use of CFD in the design of elements of the flow part 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.
8. Barlit V. V., Mironov K. A., Vlasenko A. V., Jakovleva L. K. *Raschet i proektirovanie protochnoy chasti reaktivnykh gidroturbin na osnove chislennogo modelirovaniya rabochego protsessa* [Calculation and design of the flow parts of jet turbines based on numerical simulation of the workflow]. Kharkov, NTU "KhPI" Publ., 2008. 216 p.
9. Lapin V. N., Chernyj S. G., Skorospelov V. A., Turuk P. A. Problemy modelirovaniya techeniy v turbomashinakh [Current modeling problems in turbomachines]. *Vestnik Kazakhskogo Nats. un-ta im. al'-Farabi. Seriya: Matematika, mekhanika, informatika* [Bulletin of Al-Farabi Kazakh National University. Series: Mathematics, mechanics, computer science]. Almaty, KazNU Publ., 2004, vol. 42, issue 3, pp. 57–66.
10. Suhorebryj P. N, Barlit V. V., Drankovskij V. Je., Rao V. S., Harvani L. K. Kharakteristiki prostranstvennogo turbulentnogo potoka i poteri energii v elementakh protochnoy chasti gidroturbiny RO500 [Characteristics of spatial turbulent flow and energy loss in the elements of the flow part of the PO500 hydro turbine]. *Problemy mashinostroeniya.* 2004, vol. 7, no. 3, pp. 13–20.
11. ANSYS. *Ansys 16.0 Release Documentation, Theory and Modelling Guide.* ANSYS Inc.: Canonsburg, PA, USA, 2015.
12. Jošt D., Škerlavaj A., Morgut M., Mežnar P., Nobile E. Numerical simulation of flow in a high head Francis turbine with prediction of efficiency, rotor stator interaction and vortex structures in the draft tube. *Journal of Physics: Conference Series.* 2015. Vol. 579.
13. Kolychev V. A., Mironov K. A., Tyn'janova I. I. Raschet i analiz balansa poter' energii v vysokonapornoy radial'no-osevoy gidravlicheskoy turbine [Calculation and analysis of the energy loss balance in a high-pressure radial-axial hydraulic turbine]. *Skhidno-Yevropeys'kyy zhurnal peredovykh tekhnolohiy.* 2005, vol. 13, no. 1/2, pp. 95–106.
14. Kolychev V. A., Drankovskij V. Je. *Raschet gidrodinamicheskikh kharakteristik napravlyayushchikh apparatov gidroturbiny* [Calculation of the hydrodynamic characteristics of the guide vanes of a turbine]. Kharkov, NTU "KhPI" Publ., 2002. 268 p.
15. Daneshkah K., Zangeneh M. Parametric design of a Francis turbine runner by means of a three-dimensional inverse design method. *25-th IAHR Symposium on Hydraulic Machinery Systems. Vol. 12.* 2010.
16. 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.
17. Jun A. A., Krylov B. A. *Raschet i modelirovanie turbulentnykh techeniy s teploobmenom, smesheniem, khimicheskimi reaktsiyami i dvukhfaznykh techeniy v programmnom komplekse Fastest-3D* [Calculation and modeling of turbulent flows with heat exchange, mixing, chemical reactions and two-phase flows in the Fastest-3D software package]. Moscow, MAI Publ., 2007. 116 p.
18. Ayli E., Kaplan A., Cetinturk H. CFD analysis of 3D flow for 1.4 MW Francis turbine and model turbine manufacturing. *ASME 2015 International Design Engineering Technical Conferences and Computers and Information in Engineering Conference. Vol. 1A. (2–5 August 2015, Boston, Massachusetts, USA).* Boston, ASME Copyright, 2015.
19. Kurosawa S., Lim S. M., Enomoto Y. Virtual model test for a Francis turbine. *IOP Conference Series: Earth and Environmental Science*. 2010, vol. 12, no. 1.
20. Ayli E., Celebioglu K., Aradag S. Determination and generalization of the effects of design parameters on Francis turbine runner performance. *Engineering Applications of Computational Fluid Mechanics*. 2016, vol. 10:1, pp. 545–564.

*Received 19.05.2019*

*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», заступник директора ННІ МІТ; 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