UDC 621.224 **doi: 10.20998/2411-3441.2022.1.08**

Y. KRUPA, Y. DEMCHUK

MODERN SOFTWARE FOR THE NUMERICAL STUDY OF FLOW IN HYDRAULIC MACHINES

In the past few decades, the field of developing computer software systems has been actively developing, which in turn leads to competition in the software market. Qualified engineers working in the hydroturbine industry must be able to use a computer not only at the user level, but also at the programmer level in order to program modules for their own needs based on existing software systems. Recently, numerical simulation has become applicable to an ever wider class of flows, replacing experimental research methods. Certain numerical models are characterized by different areas of applicability and expenditures of computational resources. The paper provides an analytical review of modern CFD software systems. The advantages and disadvantages of these programs are analyzed in terms of building a three-dimensional model of the object of study, creating a computational grid, setting boundary conditions and visualizing the calculation results. The analysis and comparison of existing mathematical models that used to calculate the spatial flow in the flow path of hydraulic machines has been carried out. There are many different programs for solving hydrodynamic problems, some of the advanced commercial software systems are Ansys, SolidWorks Flow Simulation, Autodesk CFD. There are also open source software products. These automatic design systems make it possible not only to perform high-quality modeling of systems of various physical nature, but also to study the response of these systems to external influences in the form of distributions of pressures, temperatures, and velocities. The calculation algorithms in the programs are similar; the distinctive features of the programs can be evaluated according to the following criteria: grid generation, accuracy, reliability (convergence), calculation speed, model physics, system flexibility. The use of modern software packages for studying the hydrodynamic characteristics of the flow in hydraulic machines significantly reduces the time and material resources in comparison with physical modeling.

Keywords: hydraulic machines, numerical research, hydrodynamic calculation, mathematical model, spatial flow, CAD systems.

Є. С. КРУПА, Є. О. ДЕМЧУК

СУЧАСНЕ ПРОГРАМНЕ ЗАБЕЗПЕЧЕННЯ ДЛЯ ЧИСЕЛЬНОГО ДОСЛІДЖЕННЯ ПОТОКУ В ГІДРОМАШИНАХ

В останні кілька десятиліть активно розвивається сфера розробки обчислювальних програмних комплексів, що у свою чергу призводить до конкуренції на ринку програмного забезпечення. Кваліфіковані інженери, що працюють в галузі гідротурбобудування повинні володіти комп'ютером не лише на рівні користувача, а й на рівні програміста, щоб на базі існуючих програмних комплексів програмувати модулі для власних потреб. Останнім часом чисельне моделювання стає застосовним до дедалі ширшого класу течій, замінюючи собою експериментальні методи дослідження. Ті чи інші чисельні моделі характеризуються різними областями застосування та витратами обчислювальних ресурсів. У роботі проведено аналітичний огляд сучасних програмних комплексів CFD. Проаналізовано переваги та недоліки даних програм у частині побудови тривимірної моделі об'єкта дослідження, створення розрахункової сітки, завдання граничних умов та візуалізації результатів розрахунку. Проведено аналіз та порівняння існуючих математичних моделей, що застосовуються для розрахунку просторової течії у проточних частинах гідромашин. Для вирішення гідродинамічних задач існує багато різних програм, одними з передових комерційних програмних комплексів є Ansys, SolidWorks Flow Simulation, Autodesk CFD. Так само існують програмні продукти з відкритим вихідним кодом. Дані системи автоматичного проектування дозволяють як виконати якісне моделювання систем різної фізичної природи, так і досліджувати відгук цих систем на зовнішні впливи, таких як: розподіл тисків, температур, швидкостей. Алгоритми проведення розрахунку у програмах схожі, відмінні риси програм можна оцінити за такими критеріями: генерація сітки, точність, надійність (збіжність), швидкість обчислень, фізика моделі, гнучкість системи. Використання сучасних пакетів програм для дослідження гідродинамічних характеристик потоку в гідромашинах значно зменшує витрати часу та матеріальних ресурсів, порівняно з фізичним моделюванням.

Ключові слова: гідравлічні машини, чисельне дослідження, гідродинамічний розрахунок, математична модель, просторова течія, системи автоматичного проектування.

Introduction. At the moment, software systems for numerical calculations are rapidly improving in the world. By optimizing the program code, the time spent on computational processes is reduced. The use of improved algorithms leads to the expansion of scalability, as well as a reduction in the time of reading and writing files, and improves convergence.

The application interface is constantly being improved, which makes the work easier, faster and more convenient. The main processes of algorithms for calculating the hydrodynamic medium in all similar software systems consist of the following items:

- creating the geometry of the object under study;
- setting a mathematical calculation model;
- creation of a computational grid;
- setting introductory boundary conditions;
- launching the created computational model with the specified parameters in the solver;
- viewing calculation results in graphical and numerical form [1–11].

Overview of CAD programs for creating flow geometry. The largest companies such as Autodesk, Dassault Systemes, and others, along with programs for calculating the flow, also create CAD programs in which you can build the geometry of the object of study.

Consider some CAD products that can be used to build 3D models.

The most popular and demanded programs on the market at the moment are SolidWorks, Autodesk Inventor, Kompas 3D, etc. [1–11].

In the SolidWorks 2022 program from Dassult Systemes, it became possible to transfer sections not only to cylindrical surfaces, but also to figures of varying complexity. New features of Slicing have appeared, which allows you to create 2D sections at the intersections of the selected element and a set of planes, which in turn facilitates work when joining complex parts [3, 12].

In Workbench 19.1 of the ANSYS software package, model geometry can be built in the SpaceClaim section, Geometry, or the Design Modeler module. It should be noted that the set of tools in these programs is much smaller than those of the above, but they have all the most basic functions for creating geometry, and these programs also lack an interface in Russian [13–16].

ModuleWorkbench is an innovative project management scheme, where it is possible to link different stages of calculations with different blocks of studies; the project is presented in the form of interconnected systems in the form of block diagrams [13–16]. In Autodesk and SolidWorks, the project is managed in the project tree [12, 17, 18].

Autodesk this year updated the Inventor build to version 2022. The new version, according to the development team, is much more productive than previous versions. The features of the new version will be preserved if the product is used on the recommended system requirements. It should also be noted that AutoCAD 2022 was recently introduced – the most powerful two- and three-dimensional CAD system, with new additional features [17].

Examples of the interfaces of the programs described above are shown in Fig. 1–3.

Creation of the computational grid. In SolidWorks Flow Simulation and Autodesk CFD, the simulation grid is built automatically and the grid editor is very simplified compared to ANSYS subroutines. Also in Autodesk CFD there is no clear sequence for setting parameters, the grid is generated when the solver is launched [15]. In SolidWorks Flow Simulation, the grid is generated in the project tree along with the entry points for boundary conditions [1, 12].

The ICEM CFD subprogram is included in the ANSYS software package; this is an excellent solution for creating complex computational grids specifically for hydrodynamic calculations, where there are complex curvilinear elements. ICEM CFD allows building structured (hexagonal) and unstructured (tetra) grids [13–16]. In the program, it is possible to cut flow paths from the general geometry of the hydraulic unit, to thicken the calculated cells, thereby creating a boundary layer. The creation of a computational grid is a very important stage for the calculation, since incorrect construction of the grid will lead to damage to the calculation area (Fig. 4).

Fig. 5–7 show the results of the generated computational grids in various software packages.

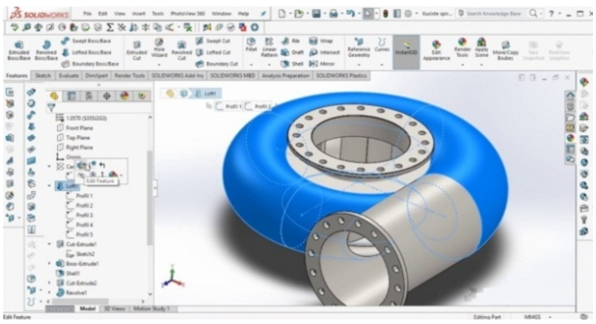

Fig. 1. Solid model of the spiral case of the hydroturbine, made in SolidWorks

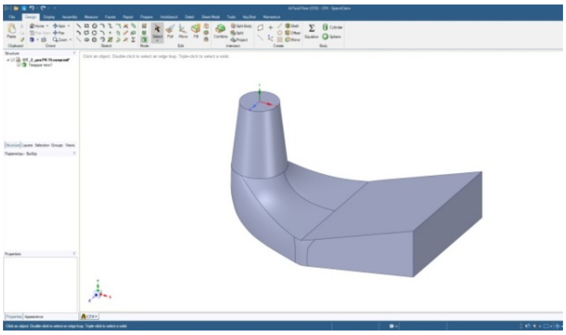

Fig. 2. Solid model of the draft tube of the hydroturbine, made in SpaceClaim

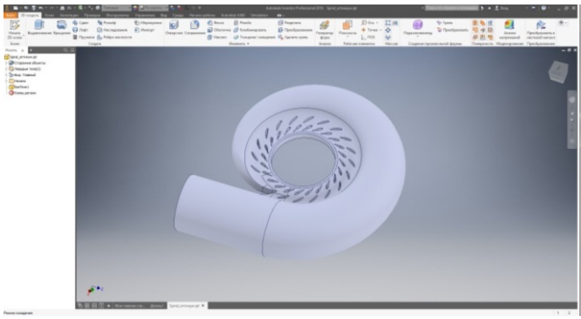

Fig. 3. Solid model of the inlet part of the hydroturbine, made in Autodesk Inventor

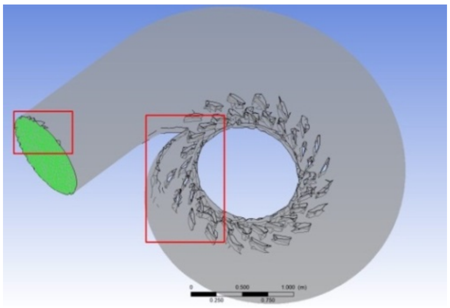

Fig. 4. Damaged wet end geometry

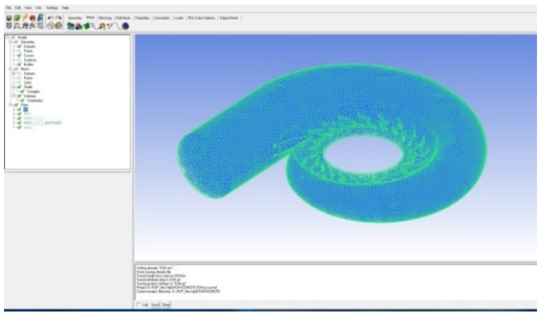

Fig. 5. The computational built in ICEM CFD

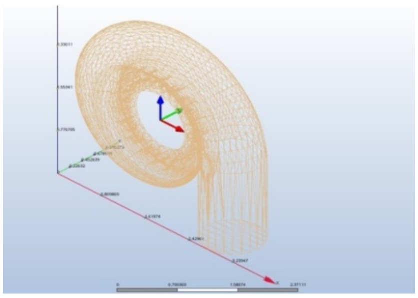

Fig. 6. The computational grid built in Autodesk CFD

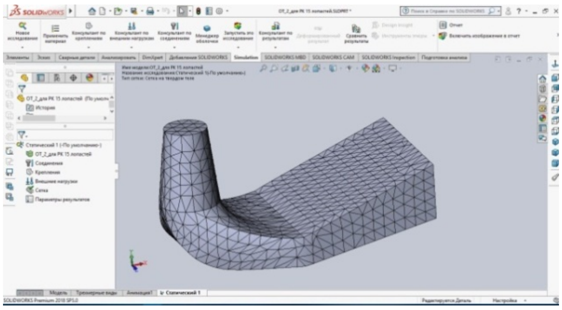

Fig. 7. The computational grid built in SolidWorks

Mathematical models used to calculate the spatial flow. The calculation of the flow of a liquid or gas in modern software products is performed by numerically solving a system of equations that describe the most general case of the movement of a liquid medium. This is the Navier-Stokes equation (1) and continuity (2):

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

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

Here, the abbreviated notation of equations i, j = 1–3 is used, summation over the same indices is assumed, x_1, x_2, x_3 are coordinate axes, t is time. The term fi expresses the action of body forces [8, 9].

In this system with four equations, the independent required parameters are the three velocity components u_1, u_2, u_3 and the pressure p. The density ρ of a liquid, as well as a gas at speeds up to 0,3 Mach, is considered a constant value.

The flows in the rotating working bodies of hydraulic machines are considered in a relative reference frame, while the fi term on the right side of the equations expresses the action of centrifugal and Coriolis forces:

$$\vec{f}_i=-\rho\left(2\vec{\omega}\times\vec{u}+\vec{\omega}\times\left(\vec{\omega}\times\vec{r}\right)\right), \quad (3)$$

where $\vec{\omega}$ is the angular velocity of rotation, $\vec{r}$ is the radius vector (the module of which is equal to the distance from the given point to the axis of rotation).

Flows in hydraulic machines are usually turbulent. Direct modeling of turbulent flows by numerically solving the Navier-Stokes equations written for instantaneous velocities is still extremely difficult, and in addition, as a rule, not instantaneous, but time-averaged values of velocities are of interest. Thus, to analyze turbulent flows, instead of equations (1) and (2), the Reynolds equation (4) is used:

$$\frac{\partial}{\partial t}\left(\rho\bar{u}_i\right)+\frac{\partial}{\partial x_j}\left(\rho\overline{u_i u_j}\right)+\frac{\partial}{\partial x_j}\left(\rho\overline{u_i' u_j'}\right)= =-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial\bar{u}_i}{\partial x_j}+\frac{\partial\bar{u}_j}{\partial x_i}\right)\right]+f_i, \quad (4)$$

where $\bar{u}_1$, $\bar{u}_2$, $\bar{u}_3$ are the time-averaged velocities;

$\overline{u_1'}$, $\overline{u_2'}$, $\overline{u_3'}$ are the pulsation components of the velocities.

Various turbulence models are used to close these equations. In the computational study of hydraulic machines, the following turbulence models have proven themselves well: "k-ε", "k-ω" and SST [1, 4].

The model "k-ε" uses two differential equations (5–6) to close the system of equations of motion of the Reynolds fluid, describing the transfer of the kinetic energy of turbulence k and the dissipation rate ε.

The disadvantages of the "k-ε" model are low accuracy when modeling flows with separation from smooth surfaces, as well as the need to use special techniques when calculating the flow near the walls.

$$\frac{\partial}{\partial t}\left(\rho k\right)+\frac{\partial}{\partial x_j}\left(\rho\bar{u}_j k\right)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial}{\partial x_j}\right)+P_k-\rho\varepsilon, \quad (5)$$

$$\frac{\partial}{\partial t}\left(\rho\varepsilon\right)+\frac{\partial}{\partial x_j}\left(\rho\bar{u}_j\varepsilon\right)= =\frac{\partial}{\partial x_j}\left(\Gamma_\varepsilon\frac{\partial\varepsilon}{\partial x_j}\right)+\frac{\varepsilon}{k}\left(C_{\varepsilon1}P_k-\rho C_{\varepsilon2}\varepsilon\right), \quad (6)$$

where $P_k=-\rho\overline{u_i' u_j'}\frac{\partial\bar{u}_i}{\partial x_j}$ is the term expressing the energy generation k, $\Gamma_k=-\mu+\frac{\mu_t}{\sigma_k}$, $\Gamma_\varepsilon=-\mu+\frac{\mu_t}{\sigma_\varepsilon}$.

The parameters ε and μ are defined as follows:

$$\varepsilon=\frac{\mu}{\rho}\overline{\left(\frac{\partial u_i'}{\partial x_j}\right)^2},\ \mu_t=\rho C_\mu\frac{k^2}{\varepsilon}.$$

Model k-ε constants: C_μ = 0,09; $C_{\varepsilon1}$ = 1,44; $C_{\varepsilon2}$ = 1,92; σ_k = 1,0; σ_ε = 1,3 [1, 4].

There is also the "k-ω" turbulence model developed by Wilcox. In this model, the differential equation for the dissipation rate ε is replaced by the differential equation for the frequency of turbulent fluctuations ω. This model has shown itself well in calculations near the walls.

The disadvantage of the "k-ω" model, in contrast to the "k-ε" model, is the excessively strong dependence of the calculation results on the specified values of ω in the inlet section [4].

A hybrid variant between "k-ε" and "k-ω" turbulence models was proposed by Menter − SST (Shear Stress Transport) model of shear stress transfer.

Through the use of a special switchable function, one or another turbulence model is activated. Switching is carried out depending on the distance of the mesh nodes from the wall.

Thus, the SST model combines the stability and accuracy of the standard "k-ω" model in near-wall regions and the "k-ε" model at a distance from the walls [4].

Specifying introductory boundary conditions. In the CFX-Pre (ANSYS) module, boundary conditions in the program tree [14].

The following parameters are set on the boundaries of the computational domains:

- at the inlet – mass flow rate;
- on the wall – no-slip condition (velocity is zero);
- at the outlet – static pressure.

Next, it is necessary to save the .def file, which contains the calculation area with all the input parameters, which can be changed if necessary.

We pass to the next stage, namely, to the numerical calculation itself. If we created a project in Workbench, then we need to go to the next Solution action in the list. Subsequently, the CFX-Solver Manager module should open with the file already selected for the calculation. Here you can set the number of processor cores that will be involved in the calculations.

At the start of the calculation, in the online mode, you can see the construction of graphs and the passage of iterations (Fig. 8).

After the end of the calculation, we get the .res file with the results.

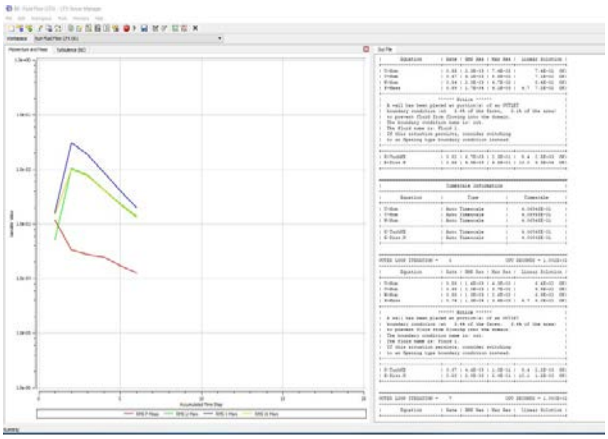

Fig. 8. Interface CFX Solver Manager

Boundary conditions are set similarly in Autodesk CFD and SolidWorks Flow Simulation. However, in these software systems, in contrast to ANSYS, in which there is a clear sequence of Geometry-Mesh-Setup-Solution-Results, the computational grid is specified along with the boundary conditions.

Results of a numerical study of the spatial flow in the flow part of hydraulic machines. To view the results of hydrodynamic calculation in the ANSYS software package, you need to download the file with the results [16, 19–20].

In the project tree, it is possible to set additional planes that display the necessary parameters, such as pressures, velocities, and temperatures. On the planes, the distribution of the given values will be displayed in accordance with the scale.

The type and division of the scale can be changed. It is also possible to visualize flow lines with animation function. All graphical results can be combined [20].

To obtain numerical data, it is recommended to use a table in which you need to register functions.

All these functions are also possessed by SolidWorks and Autodesk software packages. In addition, these programs contain various report templates that can be exported to Microsoft Word [15, 17].

Fig. 9–11 show examples of graphical visualization of the results of numerical studies performed in various CFD software packages.

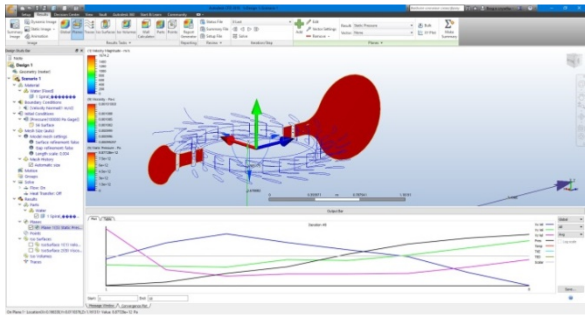

Fig. 10. Graphical representation of results in Autodesk CFD

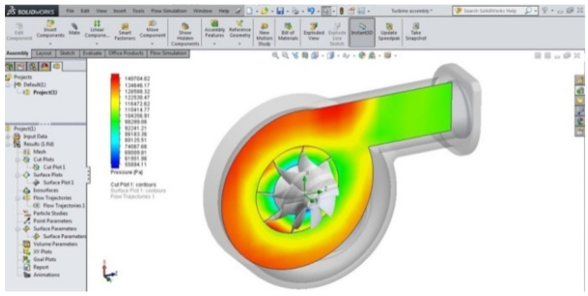

Fig. 11. Graphical representation of results in SolidWorks Flow Simulation

Conclusions. The paper presents an analytical review of the current state of software systems for the numerical study of flow in solid parts of hydraulic machines, such as SolidWorks, Autodesk CFD, Ansys.

The main functionality of CFD programs is described in terms of constructing the geometry of the object under study, creating a grid, subsequent calculation and visualization of the study results.

The use of modern CFD programs to study the hydrodynamic characteristics of the flow in hydraulic machines significantly reduces the time and material resources compared to physical simulation.

References

1. 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.
2. 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
3. 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.
4. Шевченко Н. Г., Шудрик А. Л., Радченко Л. Р. Особенности численного моделирования течения вязкой жидкости в каналах погружных лопастных насосов низкой и средней быстроходности. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units*. Kharkiv: NTU "KhPI". 2015. No. 45 (1154). P. 76–81.
5. 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.

6. Рєзва К. С., Дранковський В. Е., Тиньянова І. І. Дослідження високонапорних оборотних гідромашин. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units.* Kharkiv: NTU "KhPI". 2017. No. 42 (1264). P. 84–88.
7. 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.
8. 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.
9. 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.
10. Bychkov I. M. Verification of the OpenFOAM application package on aerodynamic profile flow problems. *XIX school-seminar "Aerodynamics of Aircraft"*. 2008.
11. Stefan D., Rudolf P. Proper Orthogonal Decomposition of Pressure Fields in a Draft Tube Cone of the Francis (Tokke) Turbine Model. *Journal of Physics: Conference Series*. 2015. Vol. 579.
12. *Что нового в SOLIDWORKS 2022. Особенности новой версии.* URL: https://www.syssoft.ru/softpower/chto-novogo-v-solidworks-2022 (дата звернення: 07.04.2022).
13. *ANSYS CFX: CFD Software.* URL: https://www.ansys.com/products/fluids/ansys-cfx (дата звернення: 02.06.2022).
14. *ANSYS ICEM CFD | CAE Expert.* URL: https://cae-expert.ru/product/ansys-icem-cfd (дата звернення: 02.06.2022).
15. *ANSYS ICEM CFD, Сеточный генератор – CADFEM*. URL: https://www.cadfem-cis.ru/products/ansys/geometry/icemcfd/ (дата звернення: 05.06.2022).
16. *ANSYS CFX, Моделирование течений, расчет турбомашин – CADFEM*. URL: https://www.cadfem-cis.ru/products/ansys/fluids/cfx/ (дата звернення: 10.06.2022).
17. *Autodesk CFD | CFD Software | Autodesk.* URL: https://www.autodesk.com/products/cfd/overview (дата звернення: 03.06.2022).
18. *Inventor Vs SolidWorks | Which is Better & Why?* URL: https://www.buildercentral.com/inventor-vs-solidworks (дата звернення: 04.06.2022).
19. *SolidWorks vs Autodesk Inventor | CAD Software Compared.* URL: https://www.scan2cad.com/blog/cad/solidworks-vs-autodesk-inventor/ (дата звернення: 03.06.2022).
20. *(186) Tutorial Ansys – How to Make Simulation Fluid Flow by CFX YouTube*. URL: https://www.youtube.com/watch?v=PfZ0opXcqAQ (дата звернення: 03.06.2022).

References (transliterated)

1. 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.
2. 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
3. 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.
4. Shevchenko N. G., Shudrik A. L., Radchenko L. R. Osobennosti chislennogo modelirovaniya techeniya vyazkoy zhidkosti v kanalah pogruzhnyih lopastnyih nasosov nizkoy i sredney byistrohodnosti [Features of numerical modeling flow of viscous liquid in channels of submersible bladed pumps of low and average rapidity]. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units*. Kharkiv, NTU "KhPI" Publ., 2015, no. 45 (1154), pp. 76–81.
5. 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.
6. Riezva K. S., Drankovskyi V. E., Tynianova I. I. Doslidzhennia vysokonapornykh oborotnykh hidromashyn [The investigation of the high-pressure reversible hydraulic machines]. *Bulletin of the National Technical University "KhPI". Series: Hydraulic machines and hydraulic units*. Kharkiv, NTU "KhPI" Publ., 2017, no. 42 (1264), pp. 84–88.
7. 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.
8. 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.
9. 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.
10. Bychkov I. M. Verification of the OpenFOAM application package on aerodynamic profile flow problems. *XIX school-seminar "Aerodynamics of Aircraft"*. 2008.
11. Stefan D., Rudolf P. Proper Orthogonal Decomposition of Pressure Fields in a Draft Tube Cone of the Francis (Tokke) Turbine Model. *Journal of Physics: Conference Series*. 2015, vol. 579.
12. *Chto novogo v SOLIDWORKS 2022. Osobennosti novoy versii* [What's New in SOLIDWORKS 2022. New Version Features]. Available at: https://www.syssoft.ru/softpower/chto-novogo-v-solidworks-2022 (accessed 07.04.2022).
13. *ANSYS CFX: CFD Software*. Available at: https://www.ansys.com/products/fluids/ansys-cfx (accessed 02.06.2022).
14. *ANSYS ICEM CFD | CAE Expert*. Available at: https://cae-expert.ru/product/ansys-icem-cfd (accessed 02.06.2022).
15. *ANSYS ICEM CFD, Setochnyiy generator – CADFEM* [ANSYS ICEM CFD, Grid generator – CADFEM]. Available at: https://www.cadfem-cis.ru/products/ansys/geometry/icemcfd/ (accessed 05.06.2022).
16. *ANSYS CFX, Modelirovanie techeniy, raschet turbomashin – CADFEM* [ANSYS CFX, Modeling of currents, calculation of turbomachines – CADFEM]. Available at: https://www.cadfem-cis.ru/products/ansys/fluids/cfx/ (accessed 10.06.2022).
17. *Autodesk CFD | CFD Software | Autodesk*. Available at: https://www.autodesk.com/products/cfd/overview (accessed 03.06.2022).
18. *Inventor Vs SolidWorks | Which is Better & Why?* Available at: https://www.buildercentral.com/inventor-vs-solidworks (accessed 04.06.2022).
19. *SolidWorks vs Autodesk Inventor | CAD Software Compared.* Available at: https://www.scan2cad.com/blog/cad/solidworks-vs-autodesk-inventor/ (accessed 03.06.2022).
20. *(186) Tutorial Ansys – How to Make Simulation Fluid Flow by CFX YouTube.* Available at: https://www.youtube.com/watch?v=PfZ0opXcqAQ (accessed 03.06.2022).

Received 15.08.2022

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 Євгенія Олександрівна (***Demchuk Yevheniia***) – Національний технічній університет «Харківський політехнічний інститут», студентка кафедри «Гідравлічні машини ім. Г. Ф. Проскури»; м. Харків, Україна; ORCID: https://orcid.org/0000-0003-0260-0467; e-mail: evgenia.kolesnichenko45@gmail.com