Journal of Power and Energy Engineering, 2022, 10, 27-37
https://www.scirp.org/journal/jpee
ISSN Online: 2327-5901
ISSN Print: 2327-588X

Improving the Energy Performance of a High-Pressure Hydraulic Turbine by Researching the Flow in the Flow Part

Konstantin Mironov[1], Yuliia Oleksenko[1], Aminjon Gulakhmadov[2,3,4*]

[1]Department of Hydraulic Machines named after Academician G.F. Proskura, National Technical University "Kharkiv Polytechnic Institute", Kharkiv, Ukraine
[2]Ministry of Energy and Water Resources of the Republic of Tajikistan, Dushanbe, Tajikistan
[3]Research Center for Ecology and Environment of Central Asia, Xinjiang Institute of Ecology and Geography, Chinese Academy of Sciences, Urumqi, China
[4]Institute of Water Problems, Hydropower and Ecology of the National Academy of Sciences of Tajikistan, Dushanbe, Tajikistan
Email: *aminjon@ms.xjb.ac.cn

How to cite this paper: Mironov, K., Oleksenko, Y. and Gulakhmadov, A. (2022) Improving the Energy Performance of a High-Pressure Hydraulic Turbine by Researching the Flow in the Flow Part. *Journal of Power and Energy Engineering*, **10**, 27-37.
https://doi.org/10.4236/jpee.2022.104002

Received: January 28, 2022
Accepted: April 21, 2022
Published: April 24, 2022

Abstract

In this study, the goal is to increase the efficiency of a high-pressure hydraulic turbine. The goal is achieved by numerical flow simulation using CFX-TASCflow. This approach reduces costs and time compared to the experimental approach and allows for improving the turbine productivity and its design. The analysis of energy losses in the flow part of the turbine Fr500, as well as the analysis of the influence of the opening of the guide vanes on changes in energy losses. The results showed that the greatest losses occur in the guide vane 3.02% based on the two-dimensional model and 2.5% based on the 3D model, which significantly affects the efficiency. The analysis was carried out using programs for calculating fluid flow in two-dimensional and three-dimensional formulations. With the help of the study, the main energy problem is solved—increasing efficiency.

Keywords

Francis Turbine, Efficiency, Spiral Case, Runner, Guide Vane, Draft Tube

1. 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 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 number of physical experiments, great attention has been given to numerical experiments. 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.

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] [4] [5] [6] [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 model [1] [8] [9].

These workflow models are used in solving problems of selection and optimization of the main parameters of hydro turbines [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. The objective of the study is to calculate energy losses in the flow part of the high-pressure hydraulic turbine (Francis Turbine) using methods of the two-dimensional model and the 3D model.

2. Materials and Methods

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 channels 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.

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 [10] [11]. 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] [12].

Creating a grid is a technique for sampling all flow areas into small elements. These elements consist of nodes that calculate unknown variables. Scale factor and element size must be specified to create a grid. The accuracy of the calculation is strongly influenced by the size of the elements. An unstructured grid is created using triangles for 2D surfaces and a tetrahedron for 3D flow.

All components of the turbine have complex geometry, so geometric modeling of the turbine is very difficult. In this simulation, the basic design parameters are calculated based on the raw construction and modeled on any specialized CAD software. The blade profile is an aerodynamic surface, and to model this profile requires the generation of section 3D coordinates along its length based on the theoretical design of the blade in the data file.

3. 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] and the model developed at the hydraulic machines department [9].

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]-[13]. 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 [1]. 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] [9] [12].

We write two equations for k and ε:

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

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

where, constants for models with two differential equations $\mu_t = C_\mu p\frac{k^2}{\varepsilon}$, $C_\mu = 0.09$, $C_{S1} = 1.44$, $C_{S2} = 1.92$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, P_k – takes into account the occurrence of turbulence due to viscous friction forces and is determined by:

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

In more detail, models based on two differential equations are given in [1].

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 (1) and (2) 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 [1] [2] [12] [13].

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

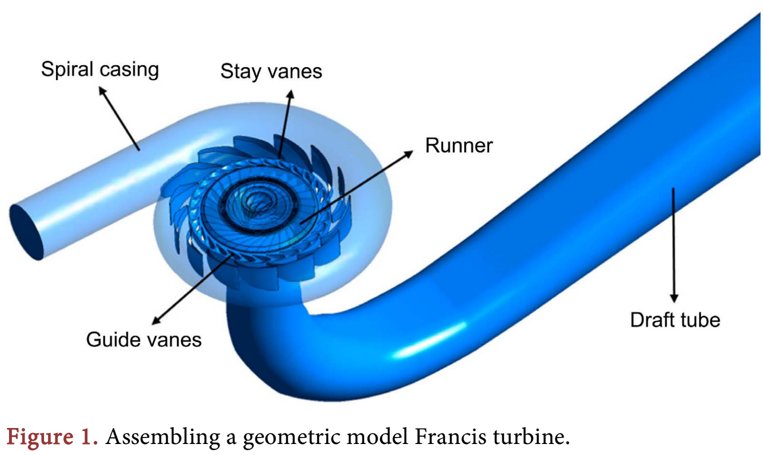


Figure 1. Assembling a geometric model Francis turbine.

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 (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 intervention 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 the 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 outputing the runner, as well as the trajectory of fluid particles in the draft tube.

The flow of fluid in the area of the stator columns, blades of the guide vane, and the runner is shown in **Figure 2**, and the distribution of total pressure is shown in **Figure 3**.

The data obtained (see **Figure 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 in **Figure 3**. The pressure becomes negative at the outlet from the runner due to the influence of the draft tube.

Figure 4 shows the trajectory of the movement of fluid particles in the draft tube (when the flow of fluid from the runner drops out) in the optimal mod, based on the calculation of the spatial flow.

The location of the current lines in the draft tube **Figure 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 with 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 number of losses in it [5] [8] [10] and on the optimal, from the point of view of minimizing inductive losses, the distribution pattern of the tangential velocity component an increase in its values in the peripheral region.

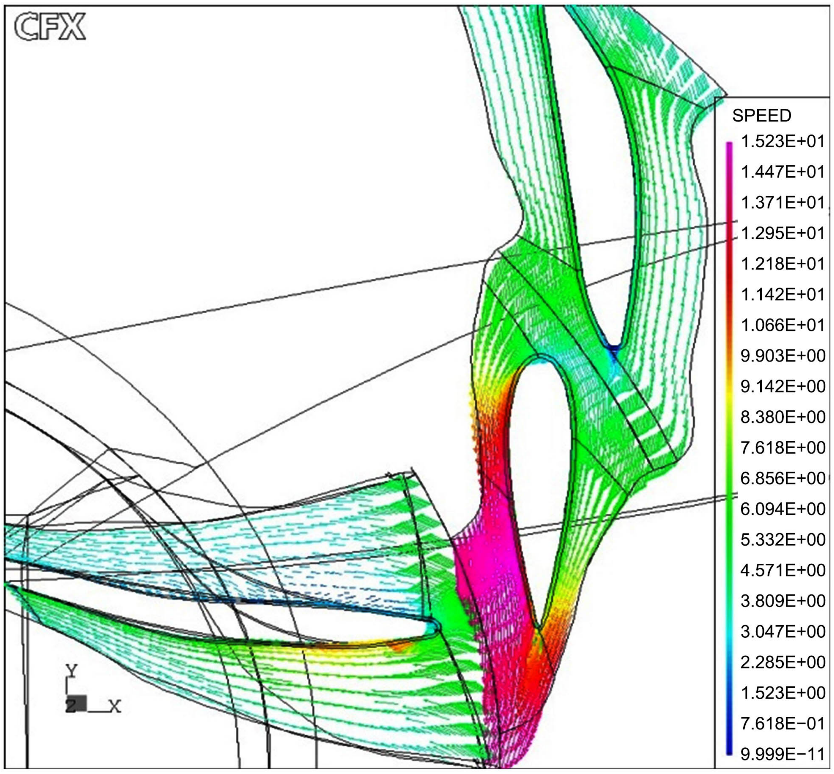


Figure 2. The field of the vectors of the velocity of the spatial flow of fluid in the region of the stator columns, the blade guide vanes and runner in the optimal mode.

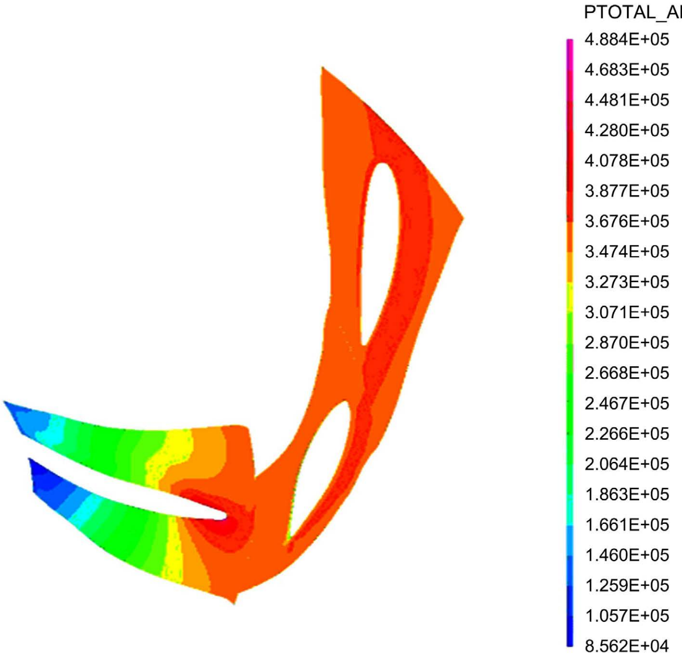
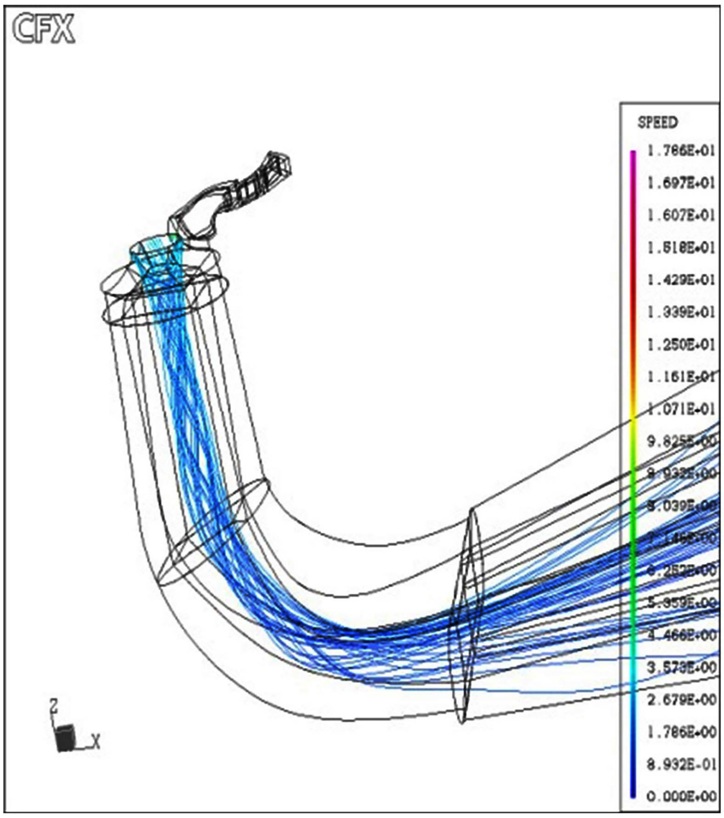
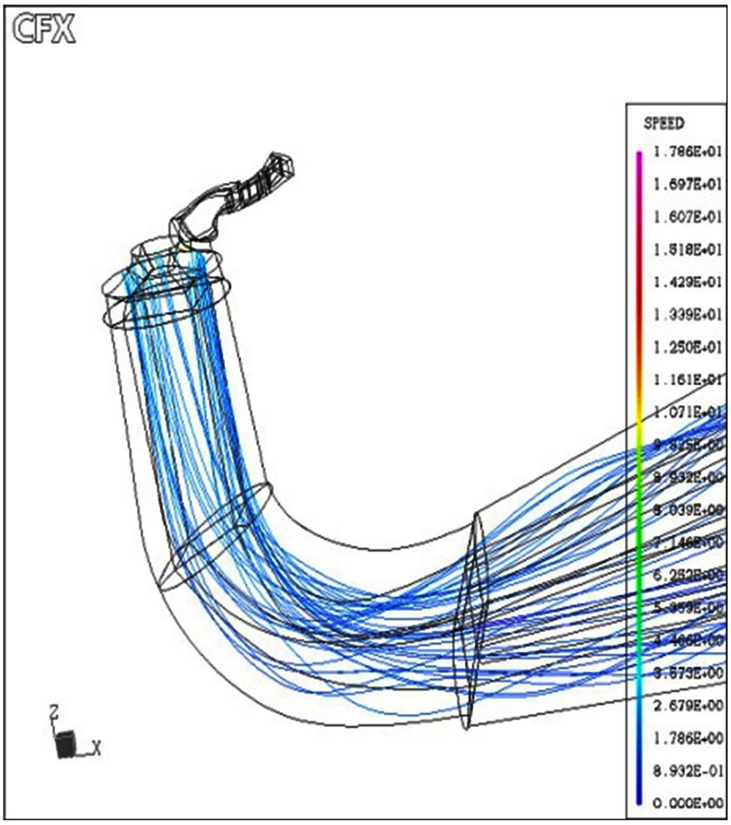


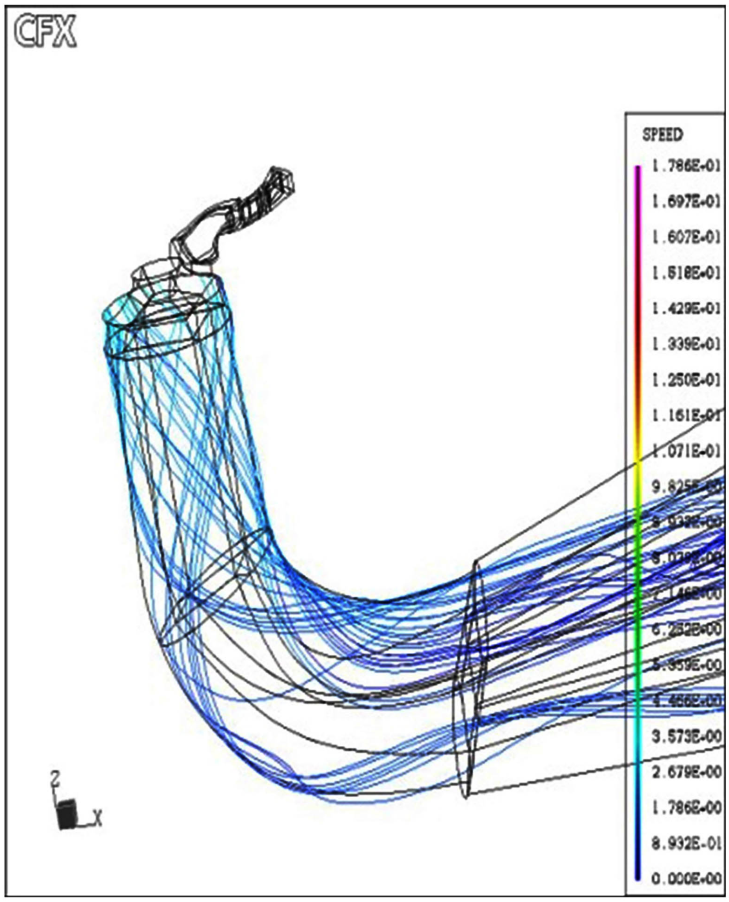
Figure 3. Isolines of total pressure in the blade systems.


CFX
SPEED
1.786E+01
1.697E+01
1.607E+01
1.518E+01
1.429E+01
1.339E+01
1.250E+01
1.161E+01
1.071E+01
3.573E+00
2.679E+00
1.786E+00
8.932E-01
0.000E+00
Z
X

(a)


CFX
SPEED
1.786E+01
1.697E+01
1.607E+01
1.518E+01
1.429E+01
1.339E+01
1.250E+01
1.161E+01
2.679E+00
1.786E+00
8.932E-01
0.000E+00
Z
X

(b)



(c)

Figure 4. The trajectories of the movement of fluid particles in the draft tube: (a) upper rim; (b) middle; (c) lower rim.

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 **Table 1**.

Today, scientists from all over the world in their research are trying to solve the problem of increasing the energy characteristics of high-pressure hydraulic turbines. This topic is relevant in the modern world and is described in many publications. Each approach has a right to exist and is suitable for specific flow and turbine parameters. Among the many options, it is necessary to choose the most suitable for a particular hydraulic turbine and the flow conditions in it. Of great importance are the boundary conditions that are specified in the calculation.

In the works of K. Ruchi [12], the flow part of a specific turbine with a certain number of blades of guide vane and runner blades is considered, specific boundary conditions are specified. The methodology of this scientist was applied using the available software. The data obtained by him are applicable to improve the turbine itself and are of interest for the evaluation and analysis of similar turbines.

The works of V. Prasad [13] were also considered, where the results of the calculation and analysis of another Francis turbine with their parameters are presented.

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

The search for new solutions, methods, study and analysis of existing developments, and most importantly the exchange of experience is the main task of a modern scientist.

Based on the existing experiments, we can compare our results and evaluate their significance. As mentioned earlier, each calculation is unique and makes it possible to improve the energy performance of the turbine. The contribution of each scientist will achieve successful results and increase the efficiency of existing and new hydraulic turbines.

In connection with the development of software systems and their modernization, new possibilities for numerical modeling are presented. The considered works of scientists can be made more accurate by using the latest versions and more powerful computers.

In turn, we will continue to research and improve the flow part of the high-pressure turbine. We will study in more detail the effect of specific elements of the flow part on losses and we will look for methods to reduce losses and increase efficiency.

This section may be divided into subheadings. It should provide a concise and precise description of the experimental results, their interpretation as well as the experimental conclusions that can be drawn.

4. Conclusions

- To reduce the number of physical experiments, it is necessary to pay more attention to numerical experiments. This will reduce the time and cost of design work.
- Considered in detail the nature of the movement of fluid in the flow part of a high-pressure hydro turbine.
- The results of the calculation optimal mode of the hydro turbine using two-dimensional and three-dimensional flow models are given, and the obtained data are in good agreement with each other.
- As can be seen from the obtained analysis results, the greatest losses occur in the guide vane, which significantly affects the efficiency. To improve the energy performance of the 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. Using this method, calculate the suitable blade shape of the guide vane.

- It is necessary to continue the study of increasing the energy characteristics of the Francis turbine and strive to improve its performance.

Funding

The study was supported by the CAS PIFI Fellowship (Grant No. 2021PC0002), Research Fund for International Scientists of National Natural Science Foundation of China (Grant No. 42150410393), K.C. Wong Education Foundation (Grant No. GJTD-2020-14), and Xinjiang Tianchi Hundred Talents Program (Grant No. Y848041).

Conflicts of Interest

The authors declare no conflicts of interest regarding the publication of this paper.

References

[1] Kolychev, V.A. (1995) Kinematic Characteristics of Flow in Blade Hydraulic Machines. ISIO, Kiev.

[2] Kolychev, V.A., Mironov, K.A. and Tyn'janova, I.I. (2010) Modeling the Energy Characteristics of Hydro Turbines at the Initial Design Stage. *Eastern-European Journal of Enterprise Technologies*, **43**, 27-38.

[3] Myronov, K.A. and Oleksenko, Y.Y. (2019) CFD Approach for Analyzing the Flow Characteristics of a High Pressure Francis Turbine. *Bulletin of the National Technical University "KhPI", Series: Hydraulic Machines and Hydraulic*, **2**, 106-111. https://doi.org/10.20998/2411-3441.2019.2.13

[4] Chernyiy, S.G., Chirkov, D.V., Lapin, V.N., *et al.* (2006) Numerical Simulation of Currents in Turbomachines. Nauka, Novosibirsk.

[5] Chung, T.J. (2002) Computational Fluid Dynamics. Cambridge University Press, Cambridge. https://doi.org/10.1017/CBO9780511606205

[6] Minkowycz, W.J., Sparrow, E.M. and Murthy, J.Y. (2006) Handbook of Numerical Heat Transfer. 2nd Edition, Wiley, Hoboken, NJ.

[7] Tucker, P.G. (2001) Computation of Unsteady Internal Flows. Springer, Berlin. https://doi.org/10.1007/978-1-4615-1439-8

[8] Myronov, K.A. and Oleksenko, Y.Y. (2018) Using CFD to Calculate the Spiral Case and Stator Columns of the High-Pressure Francis Turbine. *Bulletin of National Technical University "KhPI", Series: Hydraulic Machines and Hydrounits*, **17**, 50-53.

[9] Barlit, V.V., Mironov, K.A., Vlasenko, A.V. and Jakovleva, L.K. (2008) Calculation and Design of the Flow Parts of Jet Turbines Based on Numerical Simulation of the Workflow. NTU "KhPI", Kharkiv.

[10] Mironov, K. and Oleksenko, Y. (2019) Research of Fluid Flow in Two-Dimensional and Three-Dimensional Formulation in the Flow Part of a High-Pressure Francis Turbine. *Bulletin of the National Technical University "KhPI", Series: Hydraulic Machines and Hydraulic*, **1**, 72-76. https://doi.org/10.20998/2411-3441.2019.1.11

[11] ANSYS (2015) Ansys 16.0 Release Documentation, Theory and Modelling Guide. ANSYS Inc., Canonsburg, PA.

[12] Khare, R., Prasad, D.R.V. and Kumar, D.R.S. (2010) CFD Approach for Flow Characteristics of Hydraulic Francis Turbine. *International Journal of Engineering*

Science and Technology, **2**, 3824-3831.

[13] Prasad, V., Gahlot, V.K. and Krishnamachar, P. (2009) CFD Approach for Design Optimization and Validation for Axial Flow Hydraulic Turbine. *Indian Journal of Engineering and Material Sciences*, **16**, 229-236.