USING MODERN MECHANICAL DESIGN METHODS FOR DETERMINING THE MAIN CHARACTERISTICS OF A CRYOGENIC CENTRIFUGAL PUMP

Andrii Rogovyi[1], Oleksandr Shudryk[2], Alona Tulska[3], Yevheniia Basova[4], Kseniya Rezvaya[1], Vadym Makarov[5], Olga Lazaryeva[6], Katarzyna Antosz[7], José Machado[8]

[1]Department of Hydraulic Machines, Institute of Education and Science in Mechanical Engineering and Transport, National Technical University "Kharkiv Polytechnic Institute", 2, Kyrpychova St., 61002 Kharkiv, Ukraine

[2]Region LLC, Blvd. Tarasa Shevchenko, 33, 01001 Kyiv, Ukraine

[3]Department of Oil, gas and solid fuel refining technologies, Institute of Education and Science in Chemical Technologies and Engineering, National Technical University "Kharkiv Polytechnic Institute", 2, Kyrpychova St., 61002 Kharkiv, Ukraine

[4]Department of Mechanical Engineering Technology and Metal-Cutting Machines, Institute of Education and Science in Mechanical Engineering and Transport, National Technical University "Kharkiv Polytechnic Institute", 2, Kyrpychova St., 61002 Kharkiv, Ukraine

[5]Department of Industrial and Biomedical Electronics, Institute of Education and Science in Power Engineering, Electronics and Electromechanics, National Technical University "Kharkiv Polytechnic Institute", 2, Kyrpychova St., 61002 Kharkiv, Ukraine

[6]Department of foreign languages, Educational and Scientific Institute of International Education, National Technical University "Kharkiv Polytechnic Institute", 2, Kyrpychova St., 61002 Kharkiv, Ukraine

[7]Rzeszow University of Technology, Faculty of Mechanical Engineering and Aeronautics, Powstańców Warszawy 8, 35-959 Rzeszów, Poland

[8]MEtRICs Research Center, Campus of Azurém, University of Minho, 4800-058 Guimarães, Portugal

Emails: rezvayaks@gmail.com, Yevheniia.Basova@khpi.edu.ua

Abstract - The paper is devoted to the study of the flow structure of liquid nitrogen in centrifugal pumps and determination of the energy and kinematic characteristics of the flow in the impeller of the support cryogenic pump of the mobile nitrogen units. An analysis of modern technical solutions in this field was made identifying their advantages and disadvantages. The work examines the processes and specifics of the flow of liquid nitrogen in the impeller flow part. Integral characteristics, such as differential pressure (head) and hydraulic efficiency, of the impeller during pumping liquid nitrogen and water were obtained using modern modelling tools. An increase in hydraulic losses in the impeller during pumping of liquid nitrogen in the area containing curved channels and rotation was detected. This contradicts the tendency of loss reduction in a fixed pipe, where differential pressure is much smaller than during pumping water. A study of different configurations of booster pumps was made. According to mechanical approaches to the hydraulic equipment design, it was justified that reduced number of impeller blades (4 blades) ensures the operation of the unit in a required efficiency range. As a result of spatial modelling of the flow using the free open source CFD software OpenFOAM, distributions of the main kinematic characteristics of the flow were obtained. It was found that the physical properties of liquid nitrogen do not have a significant effect on the main parameters of centrifugal pumps, such as efficiency and pressure (the hydraulic efficiency during pumping of liquid nitrogen is only 2-5% lower than during pumping water).

Keywords: Mechanical design, Booster pump, Liquid nitrogen, Impeller, Turbulence, Differential pressure, Velocity profile, Viscosity.

1. Introduction

Currently, many fields in many oil-producing countries are at the final stage of development. The main methods of increasing production and extracting the final hydrocarbon reserves are underground capital, current repairs and intensification of production wells. Often, when developing wells and carrying out various processing of formations, mobile nitrogen units are used. This equipment transforms liquid nitrogen with a temperature of -196°C into gaseous harmless, inert, non-corrosive gas. Such gas does not harm the environment and underground equipment. The advantage of these mobile nitrogen units over compressors and mobile nitrogen compressor stations is a wide range of gaseous nitrogen supplies and the creation of pressures of 70 MPa and even more. Liquid nitrogen, which turns into gas, is pumped further into the well and is used to pump fluids during well flushing or as a working medium in jet pumps or ejectors.

The need to install support centrifugal pumps is one of the key factors for uninterrupted and high-quality operation during the development and intensification of wells. The design of pumps for pumping cryogenic media differs from classical methods.

Two types of cryogenic pumps are installed in any nitrogen units: single-stage centrifugal and plunger.

Liquid nitrogen from the cryogenic tank, under its hydrostatic pressure and the overpressure, which is created by the vaporizer, first enters the suction line into the centrifugal pump, which creates additional pressure in front of the plungers and prevents the failure of the entire unit [1-3].

An actual problem of oil industry is to increase the efficiency of the cryogenic centrifugal pumps use due to their ability to operation in the area of low flow rate and high efficiency, namely no more than 50% of the optimal flow rate. The minimum possible flow rate of nitrogen plants operated in many countries is 8-10 m^3/min of gaseous nitrogen, which corresponds to 560-700 kg/h of liquid one.

2. Literature Review

The main manufacturers of booster centrifugal cryogenic pumps (booster pump) for the operation of nitrogen units are such specialized companies as NOV, Hydraulics International, NiGen. However, they all design and produce their products based on practical experience.

The works [4-8] are devoted to research the flow of liquid nitrogen or other cryogenic media, as well as to the study of the processes occurring in cryogenic superchargers. However, they do not fully reveal the problems of designing and improving the efficiency of this type of pump. This subsequently leads to the operation of cryogenic support centrifugal pumps at modes that deviate significantly from the optimum characteristics, and the total efficiency of the entire unit reduces.

In [4] J. Zhang designed an impeller of a two-stage LNG (liquefied natural gas) submerged pump using the quasi-3D hydraulic design method. In the design procedure, the finite element method (FEM) with a quadrilateral nine-node element was adopted for the stream surfaces calculation, and the quasi-orthogonal method was used for the average stream surface calculation. For analyzing the hydraulic and cavitation performance, the entire flow passage of the two-stage LNG submerged pump was numerically simulated.

Jun-Won Suh et al. [5] optimized model based on the genetic algorithm of MATLAB was applied with different parameters of impeller that were taken as optimization variables. A numerical simulation on the Reynolds-averaged Navier-Stokes combined with the k-ε realizable turbulence model accounts for three-dimensional unsteady flows was applied for getting the performances. The research results adopted in this paper provide a theoretical basis for the design and optimization of the cryogenic fluid pump.

X. Shao [6] investigated a low-pressure LNG pump for an LNG fuel supply system with a general specific speed. Author designed the main components based on the previous studies and CFturbo. In this work the numerical analysis method was established and the performance prediction was conducted using commercial CFD packages. The goal of study was to identify problems with existing model features and find improvement directions.

E. Karakas [7] studied the unique design of the cryogenic pumps for Liquefied Natural Gas (LNG) with respect other industrial pump applications. As the result of research was investigation of a cavitation behaviour of cryogenic submerged pump in terms of Net Positive Suction Head (NPSH). Comparison of different cavitation models and assessment of their applicability to predict cavitation performance of a pump was discussed in detail in this work.

T. Hayashi et al. [8] determined analysed main kinematic and cavitation characteristics of impellers of centrifugal pumps that are used to pumping cryogenic liquid flow. The purpose of his work is to suggest an adequate design method for unshrouded impellers by comparing various shapes of shrouded and unshrouded impellers with the same meridional plane, that have various blade angle distributions, and splitter blade shapes, using a computational fluid dynamics (CFD) approach.

Works [9-13] are devoted to study fluid flow in different types of hydraulic blade machines (centrifugal pumps and turbines) using modern mechanical methods, i.e. simulation 3D flow based on optimization or artificial intelligent algorithm.

M. Chelabi et al. [9] studied the effect of the ratio of the average inlet diameter and the average exducer inlet diameter on the performance of a mixed inlet turbine and concluded that the output work and the total static isentropic efficiency were directly proportional to the ratio between the blade diameters.

W. Zhang, An Lili et al. [10] described the operation of rotational part of a centrifugal pump in order to define factors that reduce energy losses and develop an optimization algorithm for centrifugal pumps. C.-N Wang, F.-C. Yang et al [11] developed the idea of [9] and studied the efficiency of centrifugal pumps by involving AI algorithms, which assessed pump efficiency by casing section area, the interference of the impeller, the volute tongue length, and the volute tongue angle.

V.E. Drankovskiy et al. [12] also concentrated on studying the affect of geometrical parameters of the impeller such as meridional projection on the efficiency of a pump-turbine.

K. Rezvaya et al. [13] developed a mathematic model, which provides an effective assessment of the affect of hydrodynamic characteristics of elementary lattices on the runner of a reversible hydraulic machine. The regularities of changes in the hydrodynamic characteristics of elementary lattices allow to assess the loss coefficient of the runner in the operation optimal mode and to evaluate the consistency of the runner elementary lattices with each other.

In general, almost all literature covers classical methods of calculation and design of centrifugal pumps for pumping Newtonian, non-Newtonian liquids and even gas-liquid mixtures [14-19].

S. S. Antonenko [14] presents the results of three-factor influence of design and operating conditions upon the performance curves of the centrifugal pump units of the ECP (Electric driven Centrifugal Pumps) dimension-type series. The physical model of high-viscous fluid flow inside low-sized channels of pump stages was developed. Analytical expressions for recalculation factors of main performance parameters of the pump were obtained. The semi-empirical technique for prediction of performance curves of low-sized centrifugal pump stages treating high-viscous fluid with high rotational speed was created.

V. S. Boyko [15] in article designing methods of oil well exploitation by means of electric centrifugal pumps with the use of wellbore pressure distribution curves and pressure consumption characteristic of the pump used in our country are pro-pounded. Changes in properties of oil, associated gas and deposit water which happen depending on pressure and temperature have been taken into account as well as changes in parameters of flow in the area, where the pump is located (contents of gas, gas separation, viscosity of the environment) and inside the pump (consumption and density of gas-fluid mixture, viscosity). Pressure consumption characteristic of the pump has been adjusted to the oil production capacity. Adjustment of pump parameters as for its working-out and rating has been carried out, head and rate has been correlated with productive capacity of the well.

The method has been developed in accordance with modular approach, which makes it possible do substitute separate modules with more advanced ones as they arise or to substitute Poettmann-Carpenter method used in the work with another one.

B. Kim et al. [16] in this study presented the effect of impeller geometry on pumping fluid viscosity through impeller design optimization. Here, pump operation is simulated numerically by solving the Reynolds-averaged Navier-Stokes (RANS) equations at different flowrates. Experimental testing is also performed using the same oils, for numerical validation. Artificial neural-network-assisted multiobjective optimization was performed with two independent design parameters; wrap angle and splitter blade length of impeller, with head and input power as objective functions. Wrap angle and splitter blade length, both significantly affect pump performance while pumping viscous oils; as the oil viscosity increases, increasing splitter length and decreasing wrap angle improve the head significantly.

Zhengchuan Zhang et al. [17] used a hybrid Reynolds-averaged Navier–Stokes/large eddy simulations method based on the von Karman scale. The complex flow characteristics of the vertical single-stage marine centrifugal pumps have been studied in this work, and the relationship between the energy distribution characteristics of the internal flow structure and hydraulic performance has been explored under different operating conditions.

Y. Li et al. [18] in paper is studying the internal fluid flow and its influence on the dynamic characteristics of the pump impeller and to explore the causes of vibration during the transient start-up process. The geometry of the flow channel inside the centrifugal pump is established using Creo 4.0 software (American PTC company). The internal fluid flow computer simulation is carried out using Flomaster V9 software (UK Flowmaster company) to obtain the variation law of speed and flow during the start-up of the centrifugal pump, which is loaded into the simulation calculation of the centrifugal pump. The variation of speed and flow during the start-up process was further processed using the fluid-structure coupling method.

H. Hou et al. [19] in article is studied based on the local entropy production theory. Four geometrical parameters are selected to establish orthogonal design schemes including blade outlet setting angle, wrapping angle volute inlet width, and throat area. Subsequently, a 3D steady flow with Reynolds stress turbulent model and energy equation model is numerically conducted and the entropy production is calculated by a user-defined function code. The range analysis is made to identify the optimal scheme indicating that the combination of local entropy production and orthogonal design is feasible on pump optimization.

3. Goal of the Work

The main goal of the work is to study the flow structure of liquid nitrogen, to determine the energy characteristics and kinematic parameters of the flow in the impeller of the centrifugal cryogenic booster pump of the nitrogen unit for further improving the total efficiency of the entire unit.

4. Materials and Methods

Liquid nitrogen differs in its properties from water, on the parameters of which all centrifugal pumps are designed (density 808 kg/m^3 and dynamic viscosity 0.00016 Pa·s against 1000 kg/m^3 and 0.001 Pa·s, respectively).

In this work modeling of media pumping along the pipeline that connects the tank with the booster pump was carried out in order to compare the nature of water (H_2O) and liquid nitrogen (N_2) flows, and pressure losses along the line length. The simulations were performed in the free open source CFD software OpenFOAM using the solver for homogeneous incompressible medium called simpleFoam. The studied area is a straight cylindrical section with a length of 1 m.

The OpenFOAM software package uses the weighted residuals method for discretion the mathematical model. To check the convergence of the iterative process, it is necessary to set the value of the root-mean-square (RMS) residuals – lower than 10^{-4}.

A system of Reynolds averaged Navier-Stokes equations and equation of continuity was used for 3D modeling of the flow in the inlet line and the centrifugal pump flow part [20]:

$$\begin{cases} \dfrac{\partial u_i}{\partial x_j} = 0; \\ \dfrac{\partial u_i}{\partial t} + u_j \dfrac{\partial}{\partial x_j} = F_i - \dfrac{1}{\rho}\dfrac{\partial p}{\partial x_i} + \dfrac{\partial}{\partial x_j}\left[\left(\nu + \nu_T \right) \dfrac{\partial u_i}{\partial x_j} \right], \end{cases} \quad (1)$$

where $i, j = 1..3$; F_i – coordinate axis; u_i – projections of the velocity vector on the coordinate axis; p – hydrodynamic pressure, ν_T – turbulent kinematic viscosity of liquid, ρ – liquid density.

The flow in rotating working bodies is considered in the relative coordinate system, while the term F_i on the right side of equation (1) expresses the effect of centrifugal and Coriolis forces: $\overrightarrow{F_i} = -\rho\left(2\overrightarrow{\omega_a} \times \vec{u} + \overrightarrow{\omega_a} \times \left(\vec{\omega}_a \times \vec{r}\right)\right)$, where ω_a – rotational angular velocity; $\vec{r}$ – radius vector (the module of which is equal to the distance from a given point to the rotation axis).

Different turbulence models are used to close the system of equations (1) [21]. The Menter SST turbulence model is the most appropriate in the cases considered in the work [21-23].

The Menter SST model is written by superposition of models $k-\varepsilon$ and $k-\omega$, based on the fact that models of the type $k-\varepsilon$ better describe the properties of free shear flows, the model $k-\omega$ have an advantage in modeling near-wall flows. A smooth transition from the $k-\omega$ model near the wall area to the model $k-\varepsilon$ at a distance from solid walls is provided by the entering of a weighted empirical function F_1.

A second important aspect of the model is the modification of the standard relationship between k, ω and turbulent viscosity. The modification of this relationship consists in introducing a transition to Bradshaw's formula in the near-wall area. According to Bradshaw's proposal, the shear stress in the boundary layer is proportional to the energy of turbulent pulsations.

To determine the initial values of the turbulence parameters: turbulence kinetic energy, dissipation rate and specific dissipation rate.

To determine the kinetic energy of turbulence k:

$$k = \frac{3}{2}(UI)^2, \quad (2)$$

where U – input velocity; I – turbulence intensity (when calculating 3D flows in hydraulic machines, it is usually taken 0.05).

Turbulence dissipation rate ω:

$$\omega = \frac{\rho k}{\mu}\left(\frac{\mu_t}{\mu}\right)^{-1}, \quad (3)$$

where μ_t/μ – the ratio of turbulent viscosity to dynamic one. It is worth noting that unlike molecular viscosity μ, turbulent viscosity μ_t is not a property of the liquid, but depends on the flow itself and can vary from point to point for a given flow.

The following equations are used to describe the Menter SST turbulence model.

For describing turbulence kinetic energy:

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k\omega + \frac{\partial}{\partial x_j}\left[\left(\nu + \sigma_k \nu_T \right) \frac{\partial k}{\partial x_j} \right].$$

For Specific Dissipation Rate:

$$\frac{\partial \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\nu + \sigma_\omega \nu_T \right) \frac{\partial \omega}{\partial x_j} \right] + \\ + 2(1 - F_1)\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i},$$

where F_1 is blending function and can be fined as:

$$F_1 = \tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right), \frac{4\sigma_{\omega 2} k}{CD_{k\omega} y^2}\right]\right\}^4\right\},$$

$F_1 = 1$ inside the boundary layer and 0 in the free stream, $CD_{k\omega} = \max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i}, 10^{-10}\right)$,

$$S = \sqrt{2S_{ij}S_{ij}},\ S = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right).$$

For kinematic eddy viscosity:

$$\nu_T = \frac{\alpha_1 k}{\max(\alpha_1\omega, SF_2)},$$

where F_2 is the second blending function, $F_2 = \tanh\left[\left[\max\left(\frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right)\right]^2\right]$.

For Production limiter:

$$P_k = \min\left(\tau_{ij}\frac{\partial u_i}{\partial x_j}, 10\beta^* k\omega\right).$$

A set of constants for the wall layer of the SST model:

$$\sigma_{k1} = 0,85,\ \sigma_{\omega 1} = 0,5,\ \beta_1 = 0,075.$$

A set of constants for free shear layers:

$$\sigma_{k2} = 1,0,\ \sigma_{\omega 2} = 0,856,\ \beta_2 = 0,0828.$$

Other constants used in the model:

$$\beta^* = 0,09,\ k = 0,41,\ a_1 = 0,31.$$

5. Results and Discussion

As a result of the modeling, the differences in total pressures at the inlet and outlet of the line (hydraulic losses) (Table 1) and the cross-sectional velocity profile (Figure 1) were obtained.

Table 1. Comparison of hydraulic losses along the studied section of the inlet line

Q, m^3/sec	ΔP, Pa (N_2)	ΔP, Pa (H_2O)
0.00024	15	29
0.00036	31	57
0.00048	51	93

Based on the results of modeling in the suction line in three modes for liquid nitrogen and water, it can be concluded that the viscosity of the pumped primarily affects the amount of hydraulic losses. This is also evidenced by the velocity profiles of the calculation area in the cross section in the middle of the studied suction line Figure 1.

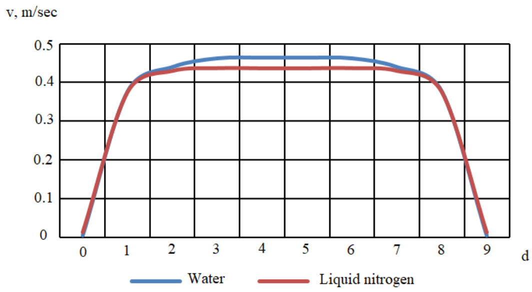


Figure 1: Profiles of fluid flow velocities in the inlet line

In the work the main research object is a booster pump that is a part of liquid nitrogen unit (Figure 2).

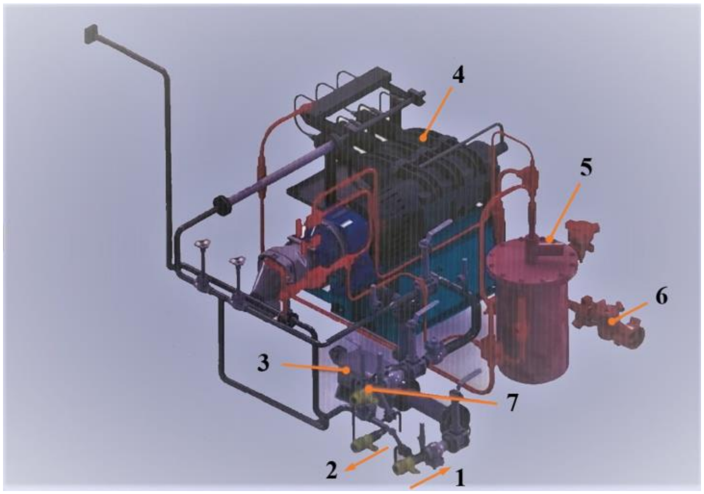


Figure 2: Structural scheme of the nitrogen circuit

This unit is called nitrogen circuit that consists of native nitrogen inflow connecting lines (pos. 1), liquid nitrogen supply connecting lines (pos. 2), booster (centrifugal) pump (pos. 3), three-plunge pump (pos. 4), evaporator (pos. 5), output interface gaseous nitrogen (pos. 6) and connecting lines for the supply of liquid nitrogen to bypass the booster pump (bypass) (pos. 7). In addition, the nitrogen circuit includes a low-pressure area (grey lines) and a high-pressure area (red lines).

To simulate the flow in the flow part of the cryogenic centrifugal pump, the impeller of the following configuration was taken as the investigated area (Figure 3).

The difference from pumps of similar speed is the reduction of the number of blades to 4, the inter-blade channel has an almost constant area from the inlet to the outlet.

In order to simplify calculations and to reduce its time, one sector was modeled, i.e. 1/4 (4 blades in the studied area). As in the previous calculation, liquid nitrogen and water were considered as working medium.

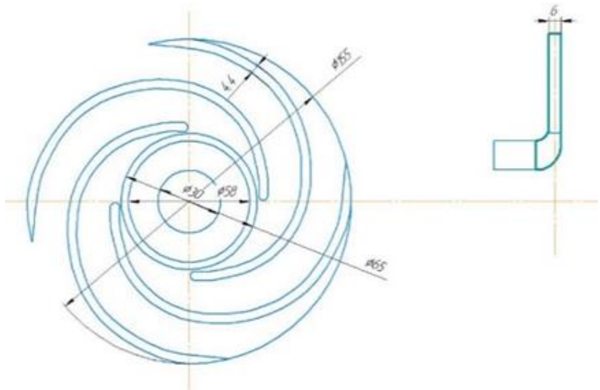


Figure 3: Meridional cross-section and profile of the investigated impeller blade

Setting boundary conditions is one of the key factors in obtaining correct output results. First, the liquid nitrogen flow was modeled at a mass flow rate of 5600 kg/h and the following data were obtained (Table 2).

Table 2. Comparison of hydraulic losses along the studied section of the inlet line

Inlet	Outlet	ΔP, atm	η_h, %
Static pressure	Mass flow rate	2.78	100
Total pressure	Mass flow rate	2.79	100
Mass flow rate	Outlet 5 atm	3.07	100
Mass flow rate	The open border is 5 atm	2.54	82.8

The torque created by the impeller blades gives a significant error in the calculation when determining the hydraulic efficiency.

Analyzing the data from the Table 2, it can be concluded that in order to obtain adequate real simulation results, it is necessary to set the mass flow rate of the pumped medium at the input, and at the output to set an open boundary, which allows the liquid to both enter the calculation domain and leave it without complicating the calculation.

Therefore, the boundary conditions for the numerical experiment were set as follows:

The mass flow rate of liquid nitrogen was set at the inlet (700, 1050, 1400, 2800, 5600 kg/h according to the practical operating experience of nitrogen units or similar flow rates when studying the water flow in terms of a different density).

An open border with a pressure of 5 atm was set at the outlet.

The sticking condition was set on the walls, i.e., the velocity is equal to 0.

In the calculation, an annular section 20 mm long was added in front of the impeller inlet. This is done in order to ensure obtaining more correct results, so there was ability to get an already formed boundary layer at the inlet of the impeller.

A hydraulic motor is used as the drive of the booster centrifugal pump. The rotation frequency of motor output shaft is regulated by the pressure of the hydraulic system. However, from practical experience, the impeller rotation frequency is taken as 3000 rpm for the calculation.

The total number of cells per sector of the calculated mesh was 2.7 million, which includes 20 prismatic layers to describe the wall layer.

The sensitivity analysis was carried out by comparing the results of calculations on three meshes of different sizes. If the number of cells becomes more than 2.5 million, the results do not differ and do not depend on the number of elements.

The cavitation model and heat exchange were not taken into account in the simulation.

Figure 4 shows the distribution of the boundary layer parameter y+ (y+ is less than 3), which indicates the correctness of the construction of the calculation mesh for the selected turbulence model.

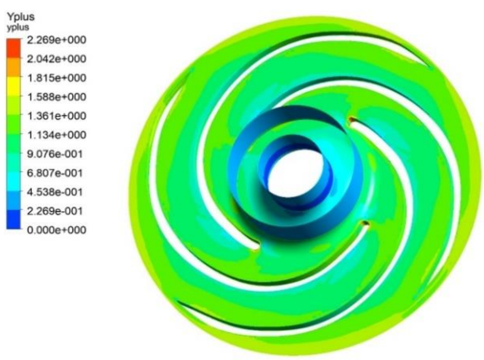


Figure 4: Distribution of parameter y^+ on the surface of the calculation area

The pressure created by the impeller is equal to the difference of full pressures (the sum of static and dynamic components) between the outlet and the inlet:

$$\Delta P = P_{\text{outlet}} - P_{\text{inlet}}, \quad (4)$$

where P_{outlet} – total pressure in the outlet of the impeller; P_{inlet} – total pressure in the inlet of the impeller.

As a result of the 3D calculation, the obtained output data allow to determine the hydraulic efficiency, since modeling does not take into account mechanical and volumetric losses as it occurs in real conditions:

$$\eta_h = \frac{\Delta P \cdot Q}{\omega_c \cdot M_r}, \quad (5)$$

where Q – pump flow rate, m³/sec; ω_c – angular speed, rad/sec; M_r – the torque created by the impeller blades.

As the results of the simulation of the impeller sector when pumping liquid nitrogen and water (Figure 5) Reynolds numbers are determined for each calculated mode. Analysis of the pump operation by Reynolds's numbers proves that flows are developed turbulent and operating modes are self-similar by Reynolds.

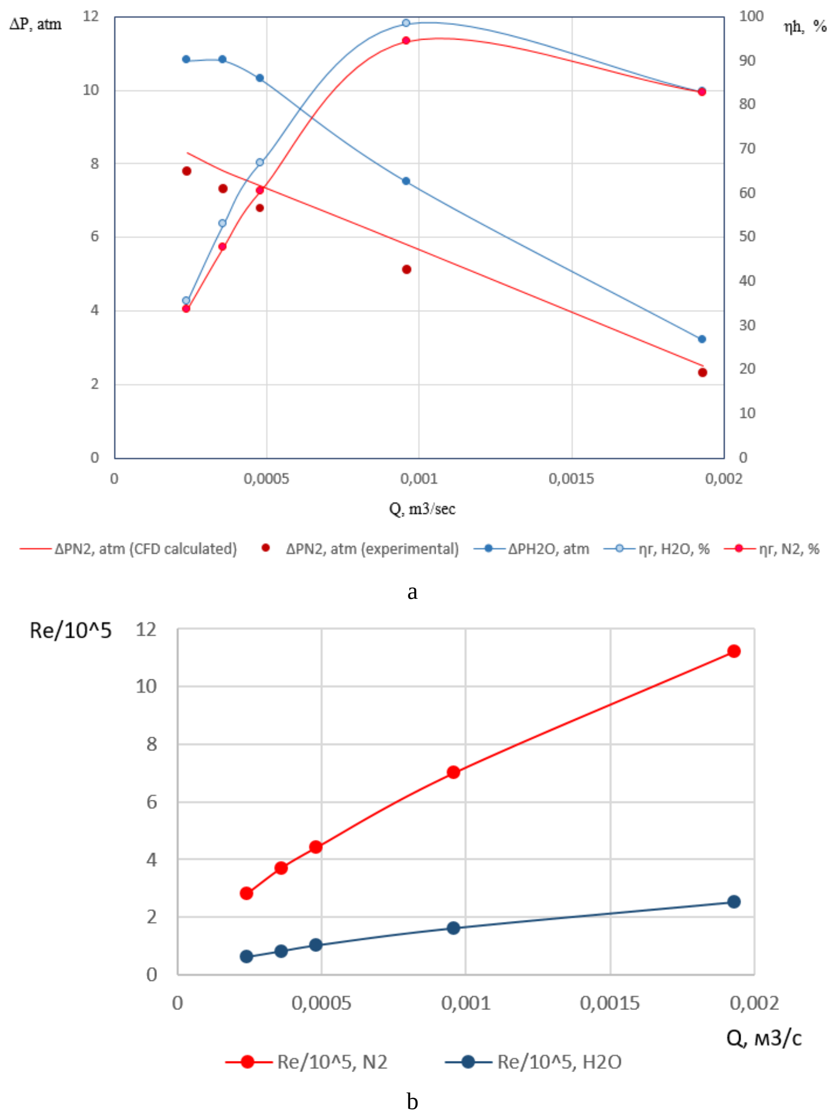


Figure 5: Results of flow simulation in the impeller of the support centrifugal pump: a – dependence of the hydraulic efficiency and differential pressure on the pump operation mode, b – dependence of the Reynolds number on the pump operation mode

It should be noted that the calculated differential pressure between outlet and inlet of the impeller corresponds to practical values when operating the pump.

The results of the calculation are adequate, with the maximum error in determining the values ΔP_{N2} is 12 %.

Hydraulic efficiency in pumping both fluids (liquid nitrogen and water) was 90-95%, which exceeds the generally accepted values for the pumps of a given speed [24].

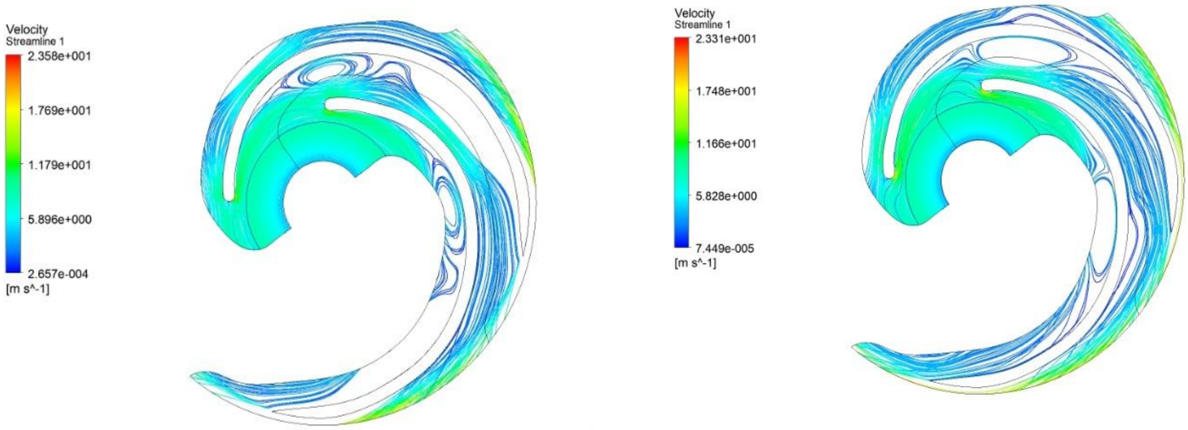
Despite the much lower viscosity of liquid nitrogen compared to water, the hydraulic efficiency of the impeller was smaller. This factor indicates that there were greater hydraulic losses passing impeller channels. All of this contradicts the previous calculations of the flow in the suction line, where the losses were less in the study of liquid nitrogen flow.

Figure 6 and Figure 7 show the distribution of velocities and pressures along the flow midline in the impeller when pumping liquid nitrogen and water for the mode of 0.00036 m^3/sec. It should be noted that the nature of the studied medium flows are very similar, but viscosity and other physical parameters of liquid nitrogen do not have a positive effect on the efficiency of the pump as a whole.

Analysis of the relative velocity distribution and flow lines (Figure 7) in pumping liquid nitrogen indicates the presence of significant vortex phenomena and the flow twist to the opposite direction from the outlet on both the vacuum side and pressure side of the blades.

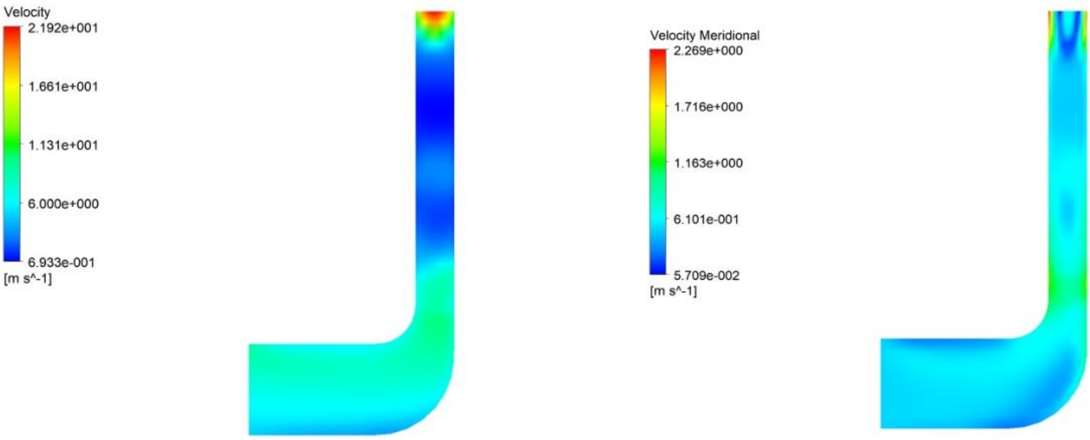
There are significant flow separations, which causes an increase of hydraulic losses in the inter-blade impeller channel when pumping liquid nitrogen than when pumping water, where the flow is smoother.

If analyze the pressure distribution fields in the impeller, there are almost no differences between two media. The flow is characteristic of centrifugal pumps, the pressure increases smoothly from the impeller inlet to its outlet without significant differences along the inter-blade channel (Figure 8).



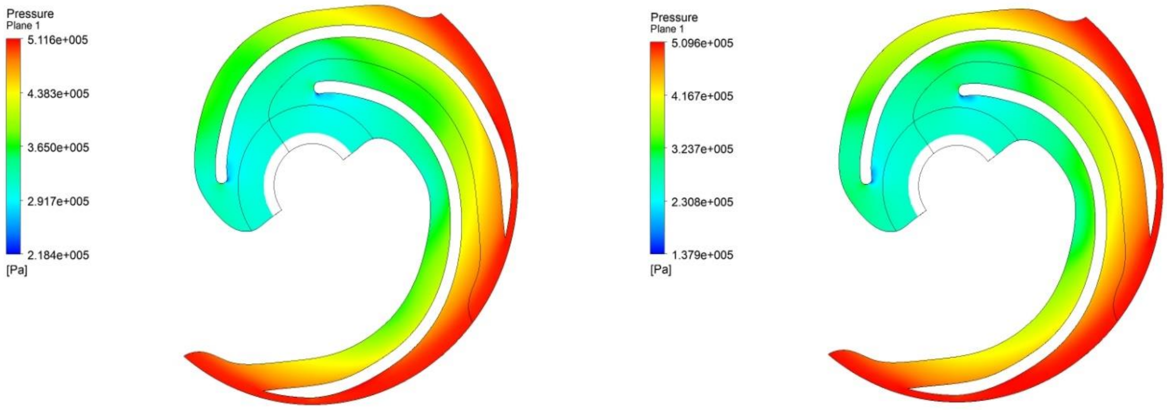
a *b*

Figure 6: Distribution of relative velocity along flow lines in the channel of the impeller during pumping liquid nitrogen (a) and water (b)



a *b*

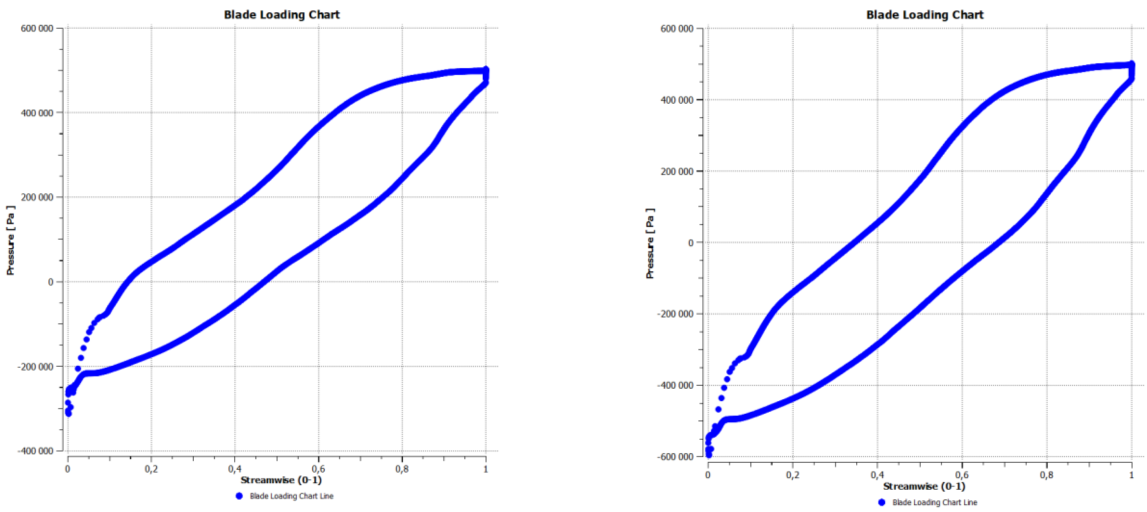
Figure 7: Distribution of meridional velocities averaged over the area during pumping of liquid nitrogen (a) and water (b)



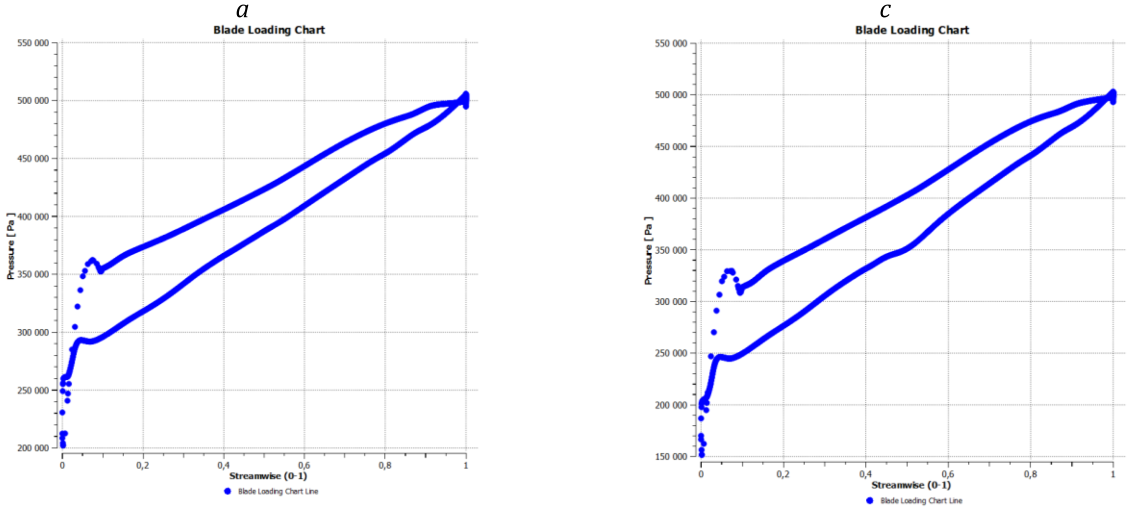
a *b*

Figure 8: Distribution of static pressure in the inter-blade channel during pumping of liquid nitrogen (a) and water (b)

The pressure distribution at the axial center plane of the impeller under different working conditions is shown in Figure 9. The average streamline along the blade was chosen as the line along which the pressure distribution was determined. The operation of the pump was considered when pumping liquid nitrogen (Figure 9 (a, b)) and water (Figure 9 (c, d)) while the flow rate was chosen to be 15 m³/min Figure 9 (a, c)) and 80 m³/min Figure 9 (b, d)).



a *c*



b *d*

Figure 9: The pressure distribution at the axial center plane of the impeller during pumping liquid nitrogen (a, b) and water (c, d) at different conditions

The curves have a residually smooth character, but in some areas, there are uneven pressure distributions. The pressure distribution in each impeller channel is not symmetrical due to the complex flow pattern in the impeller.

The graphs nature of the considered flows of liquid nitrogen and water is confirmed by the results of calculations, namely: during pumping a more viscous medium (which is water), the pressure drop between the inlet and outlet edges is greater than during pumping liquid nitrogen.

Considering the operation mode at $Q = 80\ m^3/min$, the flow rate deviation from the optimal value is clearly noted.

There are flow separations on the inlet edge, which indicates the difference between the vector of the relative velocity of the pumped medium and the angle of the inlet edge.

6. Conclusions

The flow structure of liquid nitrogen in blade system of the support cryogenic pump of the mobile nitrogen units was studied.

As a result of spatial modelling of the flow using the free open source CFD software OpenFOAM, distributions of the main energy and kinematic characteristics of the flow, such as differential pressure, hydraulic efficiency, velocity components, were obtained.

Calculated differential pressure between outlet and inlet of the impeller corresponds to practical values when operating the pump. Obtained values are adequate (maximum error of ΔP_{N2} is 12 % in comparison with the experimental data).

Analysis of the relative velocity distribution and flow lines in pumping liquid nitrogen indicates the presence of significant vortex phenomena and the flow twist to the opposite direction from the outlet on both the vacuum side and pressure side of the blades.

Analysis of the pressure distribution fields in the impeller, there are almost no differences between two media.

Analysis of the pressure distribution at the axial center plane of the impeller along the average streamline proves the complexity of the flow in the impeller. The curves have a residually smooth character, but in some areas there are uneven pressure distributions.

The calculations were carried out for two media and the hydraulic efficiency during pumping of liquid nitrogen is only 2-5 % lower than during pumping water.

New configuration of impeller was proposed. According to mechanical approaches to the hydraulic equipment design, it was justified that reduced number of impeller blades (4 blades) ensures the operation of the unit in a required efficiency range.

Based on the conducted researches, there is a need to improve the methodology and concept of calculation and design of hydraulic machines of this type for pumping cryogenic media in order to maximize efficiency.

There is a need to study the working process of a centrifugal pump consider the cavitation model, which takes into account the multiphase and compressibility of the media.

Acknowledgements

The general approach has been partially developed within the programs for introducing digital services in training specialists in 131 Applied Mechanics, 133 Industrial Engineering, 161 Chemical technology and engineering, 171 Electronics, and 185 Oil and Gas Engineering and Technology at the National Technical University "Kharkiv Polytechnic Institute".

References

[1] Mohhamed, I. I. Coil tubing unit for oil production and remedial measures, River Publishers: New York, USA, 2021; 250 p.
[2] Katerynchuk, P. O., Rymchuk, D. V., Tsybulko, S. V., Shudryk, O. L. Osvoiennia, intensyfikatsiia ta remont sverdlovyn (Development, intensification and repair of wells), Prom-art: Kharkiv, Ukraine, 2018; 602 p.
[3] Global Energy Services. Available online: http://www.nov.com (accessed on 31 December 2022).
[4] Zhang, J., Xu, C., Zhang, Y., Zhou, X. Quasi-3D hydraulic design in the application of an LNG cryogenic submerged pump. Journal of Natural Gas Science and Engineering 2016, Vol. 29, p. 89-100. https://doi.org/10.1016/j.jngse.2015.12.022.
[5] Suh, J.-W., Yang, H.-M., Lee, K.-Y., Hong, J.-H. and Choi, Y.-S. A numerical investigation of cryogenic cavitation flow in LNG submerged pump. Journal of Physics: Conference Series 2022, Vol. 2217, [012051]. https://doi.org/10.1088/1742-6596/2217/1/012051.
[6] Shao, X. Optimal a Partial Emission Circulation Pump in Cryogenic Systems Based on Reducing Hydraulic Loss and Improving Cavitation., Social Science Research Network, Elsevier. 2023. http://dx.doi.org/10.2139/ssrn.4390479.
[7] Karakas, E. Computational Investigation of Cavitation Performance and Heat Transfer in Cryogenic Centrifugal Pumps with Helical Inducers. Doctor of Philosophy in Mechanical Engineering, University of Nevada, Reno, 2019. 148 p.

[8] Hayashi, T., Nakamura, Y., & Miyagawa, K. Design optimization of a low specific speed centrifugal pump with an unshrouded impeller for cryogenic liquid flow. IOP Conference Series: Earth and Environmental Science. 2019, 240(3), [032046]. https://doi.org/10.1088/1755-1315/240/3/032046.

[9] Chelabi, M.A.; Dobrotvorskiy, S.; Basova, Y.; Aleksenko, B.A.; Edl, M.; Zdebor, J.; Machado, J. Influence of the Main Geometrical Parameters on the Design and Performance of Mixed Inflow Turbines. Appl. Sci. 2022, 12, 12165. https://doi.org/10.3390/app122312165.

[10] Wei Zhang, Lili An, Xiaojun Li, Feng Chen, Lidong Sun, Xunming Wang, Jie Cai. Adjustment method and energy consumption of centrifugal pump based on intelligent optimization algorithm. Energy Reports. 2022, Volume 8, p. 12272-12281, https://doi.org/10.1016/j.egyr.2022.09.031.

[11] Wang, C.-N.; Yang, F.-C.; Nguyen, V.T.T.; Vo, N.T.M. CFD Analysis and Optimum Design for a Centrifugal Pump Using an Effectively Artificial Intelligent Algorithm. Micromachines 2022, 13, 1208. https://doi.org/10.3390/mi13081208

[12] Drankovskiy V. E., Khavrenko, M. Yu., Shudryk, A. L. K raschetu protochnykh chastey vysokonapornykh radyalno-osevykh obratymykh hydromashyn (To the calculation of the water passages of high-head radial-axial reversible hydraulic machines). Bulletin of the National Technical University "KhPI". Series: Mathematical modeling in engineering and technologies 2016, Vol. 16, p. 31–36.

[13] Rezvaya, K., Tynyanova, I., Drankovskiy, V., Makarov, V., Basova, Y., Chyzhykov, I., Ruzmetov, A. Study of the characteristics of the runner blade system of a hydraulic machine. International Journal of Mechatronics and Applied Mechanics 2022, Issue 12, p. 74-80.

[14] Antonenko, S. S. Zmina napirnoi ta enerhetychnoi kharakterystyk vysokoobertovoho dynamichnoho nasosa pid vplyvom viazkosti seredovyshcha, shcho perekachuietsia (Change in pressure and energy characteristics of a high-speed dynamic pump under the influence of the viscosity of the pumped medium). PhD, Sumy State University, Sumy, Ukraine, 2004.

[15] Boyko, V. S. Metodyka proektuvannia elektrovidtsentrovonososnoi ekspluatatsii naftovykh sverdlovyn (Methodology for the design of electric centrifugal pumps for the operation of oil wells). Exploration and exploration of oil and gas deposits 2013, Vol. 2(47), p. 106–116.

[16] Kim, B.; Siddique, M.H.; Samad, A.; Hu, G.; Lee, D.-E. Optimization of Centrifugal Pump Impeller for Pumping Viscous Fluids Using Direct Design Optimization Technique. Machines 2022, 10, 774. https://doi.org/10.3390/machines10090774

[17] Zhang, Z., Chen, H., Yin, J., Ma, Z., Gu, Q., Lu, J. and Liu, H. Unsteady flow characteristics in centrifugal pump based on proper orthogonal decomposition method. Physics of Fluids 2021, Vol. 33 [075122]. https://doi.org/10.1063/5.0058553.

[18] Li, Y.; Su, H.; Wang, Y.; Jiang, W.; Zhu, Q. Dynamic Characteristic Analysis of Centrifugal Pump Impeller Based on Fluid-Solid Coupling. J. Mar. Sci. Eng. 2022, 10, 880. https://doi.org/10.3390/jmse10070880.

[19] Hou H, Zhang Y, Zhou X, Zuo Z, Chen H. Optimal hydraulic design of an ultra-low specific speed centrifugal pump based on the local entropy production theory. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy. 2019, 233(6), p. 715-726. https://doi.org/10.1177/0957650918825408.

[20] Garbaruk, A. V., Strelets, M. H., Shur, M. L. Modelirovanie turbulentnosti v raschetah slozhnyih techeniy (Modeling of turbulence in the calculations of complex flows). NTU «KhPI»: Kharkiv, Ukraine, 2012; 88 p.

[21] Smirnov, P. E., Menter, F. R. Sensitization of the SST turbulence model to rotation and curvature by applying the Spalart-Shur correction term. Journal of Turbomachinery 2009, Vol. 131, issue 4, p. 1–8. https://doi.org/10.1115/1.3070573.

[22] Cozzi, F., Rohit, S., Solero, G. Analysis of coherent structures in the near-field region of an isothermal free swirling jet after vortex breakdown. Experimental Thermal and Fluid Science 2019, Vol. 109. https://doi.org/10.1016/j.expthermflusci.2019.109860.

[23] Besagni, G., Inzoli, F. Computational fluid-dynamics modeling of supersonic ejectors: Screening of turbulence modeling approaches. Applied Thermal Engineering 2016, Vol. 117, p. 122–144.

[24] Zaychenko, E. T. Hydravlika y hidropryvody (Hydraulics and hydrodrives), NTU «KhPI»: Kharkiv, Ukraine, 2007; 132 p.