

FEATURES OF MATHEMATICAL MODELING OF AIR FLOW IN THE PARAXIAL ZONE OF A VORTEX-CHAMBER EJECTOR WITH A CLOSED OUTLET CHANNEL

Andrii Rogovyi¹, Oleksandr Shudryk¹, Kseniya Rezvaya¹, Danut Iulian Stanciu²,
Maryna Petruniak³, Viktor Bovkun³, Andrew Ruzmetov¹

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

²National Institute of Research and Development in Mechatronics and Measurement Technique – INCDMTM, Șos. Pantelimon, Nr. 6-8, Sector 2, 21358 Bucharest, Romania

³Department of Petroleum Engineering and Technology, National University "Yuri Kondratyuk Poltava Polytechnic", Pershotravnevyi ave, 24, 36011 Poltava, Ukraine
E-mail: rezvayaks@gmail.com

Abstract - This article is devoted to the scope of application and the possibility of expanding the use of vortex chamber pumps in various industries. Vortex chamber pumps are well-known as effective devices for pumping liquids and gases with the ability to pump solid particles, which makes them relevant in such areas as oil and gas, the chemical industry, water supply, industrial processing, and energy. Modeling of 3D airflow in vortex chamber pumps using numerical simulation methods in the open OpenFOAM package was carried out to gain a better understanding of their working process. In the calculation, the SST DDES turbulence model. The possibility of using this model appeared only in OpenFOAM version 3.0, with the implementation of an adjustment for the streamline curvature that equals 3. The results of the numerical analysis were compared with experimental data, which confirmed the model's adequacy. The obtained results show that calculated and experimental vacuum, created in the chamber of a pump in its operating mode with a closed outlet, values have excellent coincidence. This indicates the high accuracy of the selected model. The study also proves the prospects for using vortex chamber pumps in new industries and the possibilities for their further enhancement.

Keywords Vortex-chamber pump, Numerical study, Vacuum, Efficiency, Temperature field, Dynamic viscosity.

1. Introduction

aulic machines for pumping multiphase media play a key role in various industries, including oil and gas, mining, and chemical production. These machines must be designed taking into account the features of multiphase flows, such as the presence of gases, liquids, and solid particles.

There are types of hydraulic machines whose design features allow them to work effectively with multiphase media.

Vortex pumps use centrifugal force to pump liquids with suspended solids or gas bubbles. The main advantages of vortex pumps are the ability to pump highly abrasive and corrosive media, as well as the ability to work with suspensions and foam solutions. Vortex pumps are widely used in oil and gas, chemical, construction, and other industries [1].

Piston pumps are often used to pump liquids with high solids or gas bubble content. They provide

high pressure levels and precise capacity, making them ideal for processes that require strict flow. The disadvantages of piston pumps are the complexity and high cost of maintenance, especially when pumping abrasive materials [2].

Screw pumps are positive displacement pumps that use a screw rotor that rotates inside a stator. These pumps are capable of pumping multiphase media with high solids content and highly viscous liquids. They are often used in the oil and gas industry for oil and gas production and the food industry for pumping thick products [3].

Centrifugal pumps are widely used for pumping multiphase media due to their simple design and high efficiency. However, they are less suitable for handling highly viscous or abrasive media. Special designs, such as multi-channel impellers, can be used to improve performance with multiphase flows [4].

Diffuser pumps use a system of blades and guides to increase pressure and control the liquid

flow. They are effective in handling fluids containing gaseous particles. These pumps are often used in cases when it is required a high degree of flow and pressure control [5].

2. Literature Review

In the modern world, the creation of new hydraulic machines for pumping multiphase, often abrasive mixtures, and improving the efficiency and reliability of existing ones, occupies one of the priority positions in the field of hydraulic and power engineering [6-8].

Such pumps are widely used due to their ability to provide significant volumes of pumped liquid, gas-liquid mixtures and mixtures including mechanical impurities at relatively low energy costs. Scientific works of Rogovoy A.S., Semin D.A. [9-11] significantly contributed to an in-depth understanding of the processes occurring in vortex chamber pumps. These studies cover the analysis of the dynamics and aerodynamic characteristics of vortex chamber pumps, which allows for a better operation understanding of the vortex chamber pump and their impact on efficiency. These works include both experimental methods and numerical modeling, demonstrating that optimization of geometric characteristics and the choice of design solutions can significantly increase the efficiency and service life of these pumps.

The paper [12] presents the results of calculations and modeling of a 3D air flow in a vortex chamber pump at the closed outlet mode. When compared with the experiment, the authors found that under the given conditions, the most suitable model is an incompressible fluid (the SST turbulence model with a curvature correction, the value of which is equal to 1). As a result of the studies, the error in the integral parameters was 10%.

3. Goal of the Work

The main goal of this work is to select a mathematical model that allows, using numerical modeling of a 3D flow of a viscous medium, to accurately predict the characteristics and magnitude of the vacuum formed in the chamber of vortex chamber pumps, as well as flow parameters.

4. Materials and Methods

The computational domain is a solid model of the internal volume of a vortex chamber pump, which was divided into an unstructured tetrahedral mesh with 3.5 million cells. The maximum cell size is 1.5 mm, the thickness of the first prismatic layer at the wall is 0.001 mm (to ensure the optimal value of the dimensionless parameter y^+) [13]. In total, there

are 18 prismatic layers to ensure a high-quality description of the boundary layer during modeling (fig. 1).

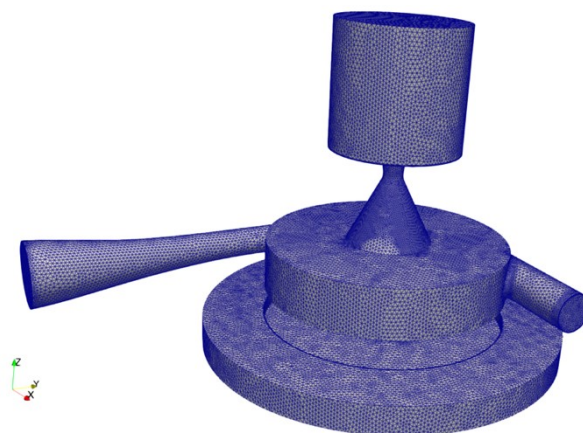


Figure 1: Calculated unstructured mesh

To resolve complex hydrodynamic processes and correctly assess the vacuum in the chamber of the vortex chamber pump, the mesh was thickened along the axis of the vortex chamber with cells 0.5 mm in size (Fig. 2).

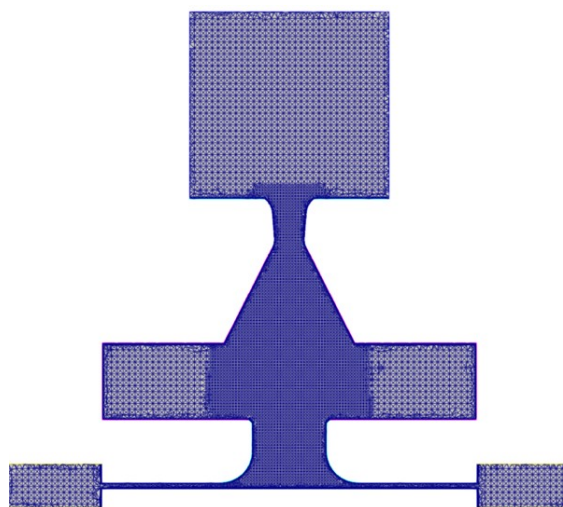


Figure 2: Mesh refinement in the center of the vortex chamber of the computational domain

At the inlet and outlet sections, the mesh is thickened to the cell size of 0.5-0.7 mm. (Fig. 3).

This is necessary to make the surface area of the cells of the studied geometry similar as circular form (Fig. 4). This allows the liquid or gas flow rates to be determined most accurately.

The practice of constructing mesh and modeling has shown that calculation of a flow, which is not always stationary, and study of a compressible medium, exclude dividing the calculation domain into separate sections to simplify the mesh construction (in particular, the structured hexa-mesh), and then stitching them together. In this case, all parameters are averaged at the boundary of one

domain and transferred to the boundary of the next domain, as a result of which the calculated values are distorted. In some cases, the distortion can completely change the flow character with the violation of all physical parameters.

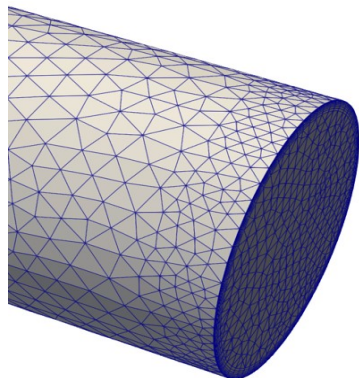


Figure 3: Mesh refinement at the inlet of the sections of the computational domain

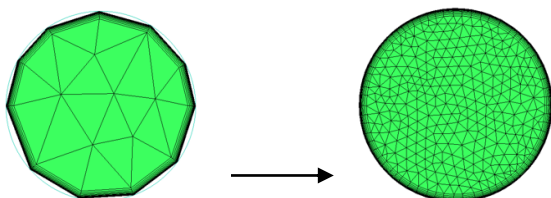


Figure 4: Example of mesh densification on a surface

In addition to averaging, another reason for the incorrect transfer of averaged parameters from domain to domain is the discrepancy between cells and nodes on adjacent surfaces (Fig. 5). Creating mesh with a 1:1 combination of nodes and cells is a very long, labor-intensive, and sometimes impossible process.

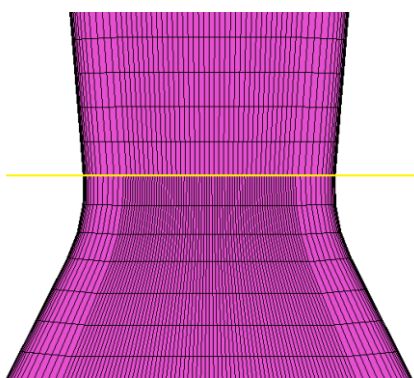


Figure 5: Discrepancy in the location of cells and nodes of two domains of the computational area

The simulation was carried out using the open source package OpenFOAM solver rhoSimpleFoam [11].

The rhoSimpleFoam is a stationary solver included in the OpenFOAM package. This solver is designed to simulate compressible flows. This solver

is based on the SIMPLE method (Semi-Implicit Method for Pressure-Linked Equation).

Main features of the solver rhoSimpleFoam are:

1. It can simulate flow at both low and high Mach numbers, taking into account the compressibility of the working medium.

2. The solver is designed to solve stationary tasks, where it is assumed that the flow properties do not change over time.

3. The SIMPLE method is used to decompose and solve the Navier-Stokes equations, which makes it possible to efficiently calculate velocity and pressure fields.

4. rhoSimpleFoam solver (since the OpenFOAM package is provided with open source code) can be extended and modified to suit specific tasks requirements, including the addition of extra turbulence models, heat transfer and other physical processes.

The rhoSimpleFoam solves a system of equations describing the behaviour of compressible fluids. The main equations solved by this solver include [14, 15]:

The mass conservation (continuity) equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0, \quad (1)$$

where ρ – density, kg/m³;
 \vec{u} – velocity, m/sec.

Momentum equation (Navier-Stokes):

$$\frac{\partial (\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla p + \nabla \cdot \tau + \rho g, \quad (2)$$

where p – pressure, Pa;
 τ – stress tensor;
 g – acceleration vector of gravity, m/sec².

State equation:

This equation establishes the connections between pressure, density and temperature. Depending on the used model, it can be expressed in different ways. For example, for an ideal gas:

$$p = \rho RT, \quad (3)$$

where R – universal gas constant, 8.31 J/(mol·K);
 T – temperature, K.

Energy equipment:

$$\frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho e \vec{u}) = -\nabla \cdot q + \dot{Q}, \quad (4)$$

where e – internal energy;
 q – heat flow;
 \dot{Q} – heat source.

In the experimental data [9, 12] it is indicated that air was considered as the studied medium.

The medium (air) parameters were changed depending on the pressure and temperature. They were following:

1. Density (according to the thermobaric conditions).

2. Dynamic viscosity of air (according to the Sutherland model) [16].

The Sutherland viscosity model is used to describe the dependence of the dynamic viscosity of a gas on temperature. This model is used to correct gas viscosity at temperature changes. It is especially important during modeling processes when the temperature changes. This model was selected due to the pressure change in the area where the flow and, as a consequence, the air temperature are considered. The advantage of this approach is that this model was obtained based on experimental data and allows more accurately and efficiently describe gas flows with subsequent determination of pressure at the desired points.

Dynamic viscosity according to the Sutherland law is determined by the following expression:

$$\mu = \frac{A_s \cdot T^{0.5}}{1 + \frac{T_s}{T}} \quad (5)$$

where $A_s=1.584 \cdot 10^{-6}$, $T_s = 110.33$ - coefficient and temperature of the Sutherland model for air.

3. Specific heat capacity (value according to the coefficients from the JANAF (Joint Army-Navy-Air Force) model tables) [17]. In OpenFOAM, it is used to set the thermodynamic properties of gases and other substances depending on temperature. It presents data on heat capacity, enthalpy, entropy and Gibbs free energy for various substances based on polynomials.

Formula for calculating specific heat capacity is following:

$$C_p = R \cdot \sum_{i=1}^n a_i \cdot T_i, \quad (6)$$

where a_i - JANAF coefficients for the selected medium.

The coefficients of the JANAF model for air are given in Table 1.

3. Temperature of the environment -293.15 K (corresponds to 20°C).

Gravity acceleration - along the Z axis = -9.81 m/s².

Boundary conditions:

- INTER_VKN_SUPPLY - input, total pressure 108189.7 Pa. This corresponds to a pressure of 700 mm H₂O + 1 atm. In OpenFOAM, during modeling compressible media flow, pressure of 0 Pa corresponds to a complete vacuum (absolute zero pressure). When pressure equals 0 Pa, the package cannot count, since the calculated pressure will fall below 0 (complete vacuum). Therefore, all pressures must be set as P+Patm=P+101325 Pa.

- IN1, IN2 - InletOutlet type boundary with total pressure of 101325 Pa (1 atm).

- INTER_VKN_EXIT - closed boundary (wall), sticking condition (no slip).

- WALL_BIG, WALL_MID, WALL_SMALL - closed boundary (wall), sticking condition (no slip).

Table 1. The coefficients of the JANAF model for air

Low	High
3.4	3.001
$4.832 \cdot 10^{-5}$	$1.487 \cdot 10^{-3}$
$-1.063 \cdot 10^{-6}$	$-6.082 \cdot 10^{-7}$
$2.424 \cdot 10^{-9}$	$1.237 \cdot 10^{-10}$
$-1.25 \cdot 10^{-12}$	$-9.886 \cdot 10^{-15}$
-1029.939	-957.586
3.889	5.869

In this work, the SST-DDES turbulence model was chosen [18-21]. It is a modification of the model SST-DES.

SST-DES (Detached Eddy Simulation) and SST-DDES (Delayed Detached Eddy Simulation) models are turbulent models that are used to simulate high Reynolds number flows and transient modes between laminar and turbulent flows.

1. SST-DES is a hybrid model that combines the RANS (Reynolds-Averaged Navier-Stokes) and LES (Large Eddy Simulation) models. Near the walls, the RANS model is used to describe turbulence; away from the walls, the LES model is used. The transition between RANS and LES occurs based on a comparison of the local dimensional scale with the mesh scale.

2. SST-DDES is an improved version of the SST-DES model. SST-DDES introduces an additional delay term, which prevents premature transition to the LES model in areas with insufficient mesh resolution. This criterion ensures more stable and predictable behaviour of the model in areas where the mesh resolution is insufficient for correct use of LES, which improves the accuracy of modeling in transient modes.

The main advantage of SST-DDES over SST-DES is its more reliable control of the transition between the RANS and LES modes, which makes this model more suitable for complex tasks that require detailed solution of turbulent structures.

The main equations of the SST-DDES model are the following:

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho u k) = \nabla \cdot [(\mu + \sigma_k \mu_t) \nabla k] + P_k - \rho \frac{\sqrt{k}^3}{l_{DDES}} \quad (7)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \nabla \cdot (\rho u \omega) = \nabla \cdot [(\mu + \sigma_\omega \mu_t) \nabla \omega] + 2(1 - F_1) \rho \sigma_{\omega 2} \frac{\nabla k \cdot \nabla \omega}{\omega} + \alpha \frac{\rho}{\mu_t} P_k - \beta \rho \omega^2, \quad (8)$$

$$\mu_t = \rho \frac{a_1 \cdot k}{\max(a_1 \cdot \omega, F_2 \cdot S)} \quad (9)$$

In Equations 7-9, F_1 and F_2 denote the mixing functions and are as follows:

$$F_1 = \tanh(\arg_1^4), \quad (10)$$

$$\arg_1 = \min\left(\max\left(\frac{\sqrt{k}}{C_\mu \omega d_w}, \frac{500\nu}{d_w^2 \omega}\right), \frac{4\rho\sigma_{\omega 2} k}{CD_{k\omega} d_w^2}\right), \quad (11)$$

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega 2} \frac{\nabla k \cdot \nabla \omega}{\omega}, 10^{-10}\right), \quad (12)$$

$$F_2 = \tanh(\arg_2^2), \quad (13)$$

$$\arg_2 = \max\left(\frac{2\sqrt{k}}{C_\mu \omega d_w}, \frac{500\nu}{d_w^2 \omega}\right), \quad (14)$$

where d_w – distance to closest wall.

P_k is calculated as follows:

$$P_k = \min(\mu_\tau S^2, 10 \cdot C_\mu \rho k \omega). \quad (15)$$

DDES length scale is as follows:

$$l_{DDES} = l_{RANS} - f_d \max(0, l_{RANS} - l_{LES}), \quad (16)$$

$$l_{LES} = C_{DES} h_{max}, \quad (17)$$

$$l_{RANS} = \frac{\sqrt{k}}{C_\mu \omega}, \quad (18)$$

$$C_{DES} = C_{DES1} \cdot F_1 + C_{DES2} \cdot (1 - F_1). \quad (19)$$

where h_{max} – maximum length of cell edge.

The empirical mixing function f_d in equation 16 is defined as follows:

$$f_d = 1 - \tanh[(C_{d1} r_d)^{C_{d2}}], \quad (20)$$

$$r_d = \frac{\nu_t + \nu}{\kappa^2 d_w^2 \sqrt{0.5 \cdot (S^2 + \Omega^2)}}, \quad (21)$$

where S – magnitude of the strain rate tensor;

Ω – magnitude of the vortex-formation tensor.

Model constants: $C_\mu = 0.09$; $\kappa = 0.41$; $\alpha_1 = 0.31$; $C_{DES1} = 0.78$; $C_{DES2} = 0.61$; $C_{d1} = 20$; $C_{d2} = 3$; $\alpha_2 = 0.555$; $\beta_1 = 0.075$; $\sigma_{k1} = 0.85$; $\sigma_{\omega 1} = 0.5$; $\alpha_2 = 0.44$; $\beta_2 = 0.0282$; $\sigma_{k2} = 1$; $\sigma_{\omega 2} = 0.856$.

The value α in equation 8 is determined as:

$$\alpha = \alpha_1 \cdot F_1 + \alpha_2 \cdot (1 - F_1). \quad (22)$$

Carrying out a 3D calculation of airflow in a vortex chamber pump with a closed outlet channel, the Curvature Correction coefficient was set equal to 3 [22-23].

The streamline curvature correction is used when modeling with a high degree of flow curvature inside hydraulic and turbomachines.

This parameter is used in turbulence models to account for the flow curvature, which can affect the generation and destruction of turbulent kinetic energy. Such effects can be significant in places with curved trajectories, where centrifugal forces can change the turbulence characteristics.

Correction for streamline curvature is set through the modification of the coefficient β . Special functions that depend on the curvature parameters is used for this.

The modified coefficient takes the form:

$$\beta_c = \beta(1 + C_{curv} f_{curv}), \quad (23)$$

where β_c – adjusted coefficient value β ;

β – coefficient from equation 8;

C_{curv} – coefficient that specifies the correction level;

f_{curv} – function depending on local curvature:

$$f_{curv} = \frac{|\nabla \times u|}{\omega}, \quad (24)$$

where $|\nabla \times u|$ – module of the vortex velocity field.

The coefficient of the cubeRootVol function is taken with a value of 0.5 [24].

The cubeRootVol is a function in OpenFOAM used to calculate the cube root of the cell volume. It is used in calculations to scale or reduce values to a unit size. In the SST-DDES model, this function is used to scale various variables or coefficients in calculations and to more accurately describe turbulent flows. The cubeRootVol can be used to normalize or scale terms such as turbulence diffusion or other parameters that depend on the cell volume. This helps to ensure numerical stability and improve the calculation accuracy in complex turbulent flows.

The calculation of three-dimensional flow in the OpenFOAM package was carried out using the main parameters up to the values of the root-mean-square residuals 10^{-4} :

- velocity components U_x, U_y, U_z ;
- pressure p ;
- internal energy e ;
- kinetic energy of turbulence k ;
- specific dissipation rate ω .

To stabilize the calculation, the turbulence model was disabled for the first 35 iterations – the parameters k and ω are not calculated.

5. Results of Modeling

A preliminary comparison of two variants of calculation meshes was carried out:

1. The computational area is divided into three parts, after which the computational mesh was stitched into one area.

2. The computational domain is represented by one body.

Calculations showed that when dividing the domain and subsequently stitching the meshes, the flow character does not correspond to reality and contradicts the co-author previous calculations [12].

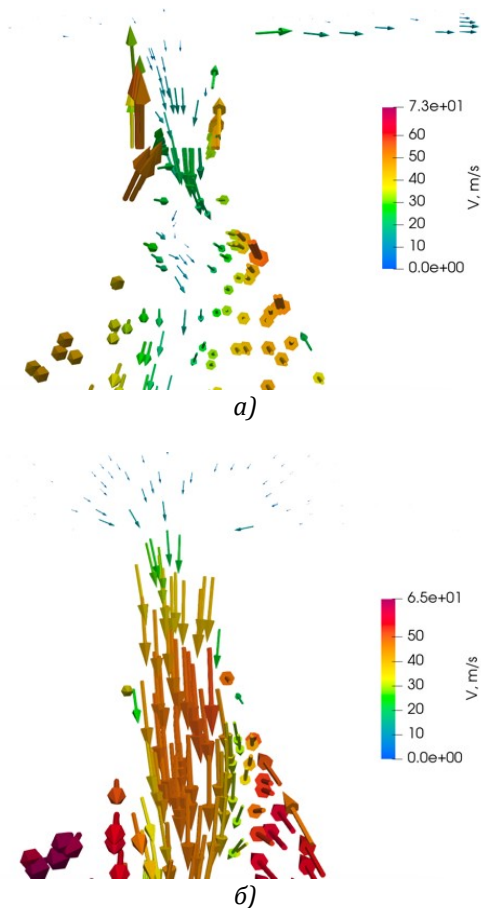


Figure 6: Velocity vectors at the upper axial inlet of the studied vortex chamber pump with mesh division (a) and without division (b)

Fig. 6 shows that when dividing the area into sections, the calculation gives overestimated values of velocities and the presence of air counter flow at the wall to the outlet. In turn, this affects the correct determination of the vacuum value that is formed in the chamber of the vortex chamber pump. In this case, the maximum vacuum value is 1347 Pa.

Verification and determination of the adequacy of the selected mathematical models for describing the properties of the working medium and turbulence were carried out by comparing the pressure parameters in the inlet and the outlet plane, as well as the flow components at the inlet of the vortex chamber pump and at the upper axial inlet (Table 2). The dimensionless parameter y^+ does not exceed 0.22 (Fig. 7), which indicates the correct construction of the calculation mesh, in particular about the prismatic layers and the correctness of the description of the boundary layer.

Table 2. Modeling results and comparison with experimental data

Parameter	Experiment	CFD calculation	Error, %
P_s , мм вод. ст.	700	699,3437	0,09
Q_s , М ³ /с	0,0018	0,001819	1,07
P_e , мм вод. ст.	625	631,0718	0,97
Q_{in} , М ³ /с	0,000843	0,000856	1,54

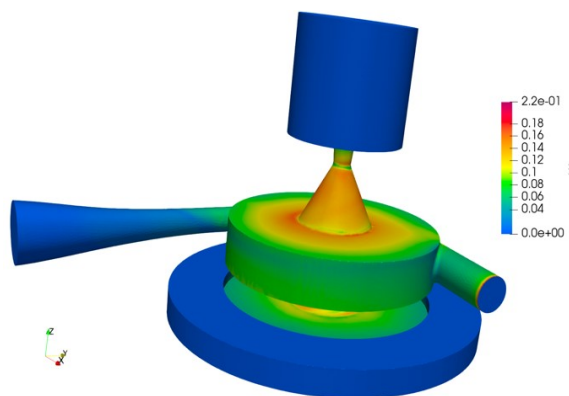


Figure 7: Dimensionless parameter y^+

Numerical flow modeling made it possible to obtain the pressure distribution along the radius of the vortex chamber. Comparison with experimental data showed that the error in determining the minimum vacuum value was 1.58% (Fig. 8). This ensures the most accurate and minimal error determination of the integral parameters of vortex chamber pumps.

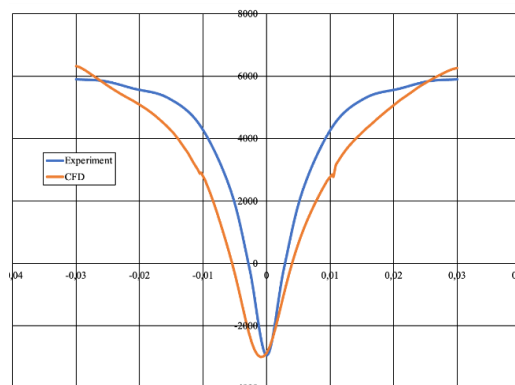


Figure 8: Pressure distribution along the radius of the vortex chamber (comparison with experimental data)

Fig. 9 shows the visualization of the pressure distribution in the cross-section of the chamber of the studied vortex chamber pump. It is worth noting the presence of a vacuum zone in the chamber center, the maximum value of which is -2996.5 Pa with the actually measured value during the experiment - 2950 Pa.

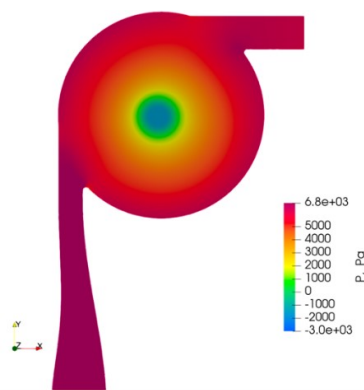


Figure 9: Pressure distribution in the cross-section of the pump chamber

Fig. 10 shows the uneven distribution of the air dynamic viscosity in the chamber of a vortex chamber pump. Its change is insignificant and the difference between the maximum and minimum values is no more than 1%.

The character of viscosity change corresponds to real physical processes during a compressed medium. At a higher temperature, gas molecules move faster and collide with each other more often, which leads to an increase in internal friction – the main aspect of viscosity.

For given boundary conditions, the temperature variation during flow through the calculation area ranges from 18 °C to 24 °C (Fig. 11). The maximum temperature is observed in the outlet of the slot diffuser, where the cross-sectional area of the calculation area is minimal.

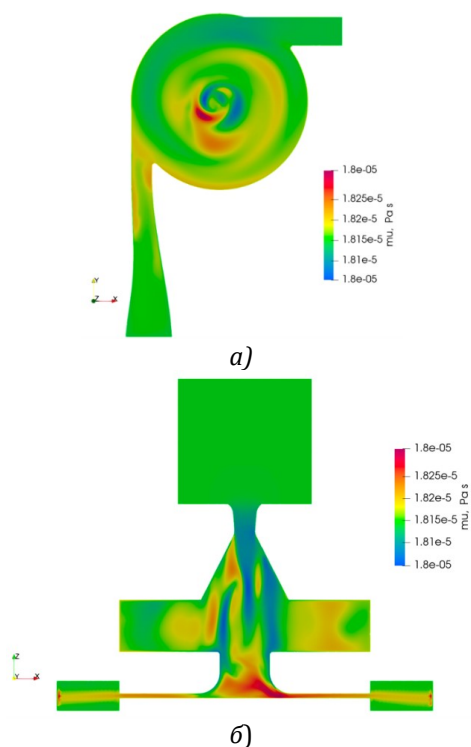


Figure 10: Distribution of air viscosity in the transverse (a) and longitudinal (b) sections of the studied vortex chamber pump

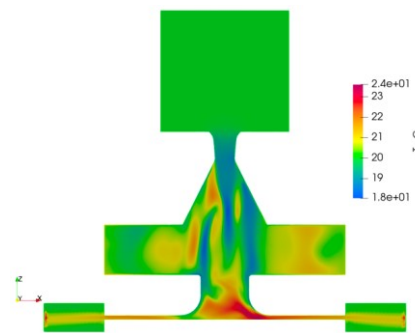


Figure 11: Temperature distribution in the longitudinal section of the studied vortex chamber pump

6. Conclusions

The article considers the main types of hydraulic machines for pumping multiphase media.

The concept of the computational mesh creation, setting the parameters of the working medium, which are based on experimental data and studies, are presented.

Taking into account the high degree of medium turbulence in the vortex chamber pump, SST-DDES turbulence model was used with the adding the streamline curvature correction that equals 3.

In order to maximize the approximation of the working medium properties during modeling to the experimental ones, the Sutherland and JANAF models were selected. These models determine viscosity and specific heat of air or another considered medium, based on the dependencies and coefficients obtained experimentally.

The results of modeling the 3D flow, where air was taken as a compressible medium and its properties were determined by thermobaric conditions, showed very good alignment with experimental data. This allows confirming the adequacy of using these models in further calculations of hydraulic machines of this type.

For the first time, the distribution of viscosity and temperature fields in a vortex chamber pump was determined.

The effect of temperature on the viscosity of the working medium and the characteristics of the pump as a whole is shown.

Using Sutherland and JANAF models in CFD calculations allows more accurate modeling of heat transfer processes and flow dynamics in gases at changing temperatures. This approach improves the accuracy of calculations, especially in thermodynamically complex systems. As a result, a detailed analysis of viscosity and its dependence on temperature is a key element for achieving high-quality results in numerical simulations.

Acknowledgements (Not Mandatory)

The general approach has been partially developed within the programs for introducing digital services in training specialists in 131 Applied Mechanics and 133 Industrial Engineering at the National Technical University "Kharkiv Polytechnic Institute".

References

- [1] Tham, K. W., & Heng, P. A. "Progress in Vortex Pumping Technology: A Review on Multiphase Flow Applications". *Journal of Mechanical Engineering Science*, 2020.
- [2] Chaban, N., & Barnaul, V. "Dynamic Behavior of Piston Pumps in Multiphase Systems: A Theoretical Approach". *Energy*, 2022.
- [3] Singh, P., & Patel, R. "Operational Performance of Screw Pumps in Multiphase Flow Systems: A Comprehensive Review". *International Journal of Multiphase Flow*, 2019.
- [4] Aydin, M. & Altintas, O. "Numerical Simulation of a Centrifugal Pump Handling Multiphase Flow". *Journal of Hydraulic Engineering*, 2021.
- [5] Meyer, J. & O'Brien, P. "Performance of Diffuser Pumps in Multiphase Flow Systems: A Comparative Study". *Journal of Fluids Engineering*, 2022.
- [6] 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>
- [7] Chelabi, M.A.; Saga, M.; Kuric, I.; Basova, Y.; Dobrotvorskiy, S.; Ivanov, V.; Pavlenko, I. The Effect of Blade Angle Deviation on Mixed Inflow Turbine Performances. *Appl. Sci.* 2022, 12, 3781. <https://doi.org/10.3390/app12083781>
- [8] Basova, Y., Dobrotvorskiy, S., Balog, M. at all. INCREASING SME SUPPLY CHAIN RESILIENCE IN THE FACE OF RAPIDLY CHANGING DEMAND WITH 3D MODEL VISUALISATION. *International Journal of Mechatronics and Applied Mechanics*. 2023, 14, pp. 35–47. [dx.doi.org/10.17683/ijomam/issue14.5](https://doi.org/10.17683/ijomam/issue14.5)
- [9] Somin D.O. Pidvishchennya yefektivnosti peremishchennya vantazhiv truboprovidnim transportom zasobami struminnoi armature [Increasing the efficiency of moving cargo by pipeline transport by jet fittings]: Dis...d-ra tekhn. nauk: 05.22.12 / Skhidnoukraïnskii natsionalnii un-tim. Volodimira Dalya. — Lugansk, 2004. — 381 p.
- [10] Rogovii A.S. Rozrakhunok vitrati perekachuvanogo seredovishcha u vikhrekamernikh nagnitachakh. [Calculation of the flow rate of medium pumping in the vortex chamber supercharger] /A.S. Rogovii, I.D. Goncharov// Naukovi pratsi Mizhnarodnoi naukovo-praktichnoi konferentsii prisvyachenoï 85-richchyu zasnuvannya KhNADU, 85-richchyu zasnuvannya avtomobilnogo fakultetu ta z nagodi Dnya avtomobilista ta dorozhnika: «Novitni tekhnologii v avtomobilebudivnitstvi ta transporti», 15-16 zhovtnya 2015 r., m. Kharkiv. – Kh: «FORT», 2015 r. – pp. 295-298.
- [11] Somin D.O. Obgruntuvannya mozhlivostei stvorenniya bagatostupinchastikh vikhrekamernikh nagnitachiv [Justification of the possibilities of creating multi-stage vortex chamber superchargers] /D.O. Somin, A.S. Rogovii, Ya.I. Maltsev // Mizhnarodna naukovo-tekhnichna konferentsiya «Gidroaeromekhanika v inzhenernii praktitsi». Cherkasi, 17-20 kvitnya 2012 r.: materialy konferentsii – Cherkasi: 2012. – pp.86-87.
- [12] Rogovii A.S. Rozrobka teorii ta metodiv rozrakhunku vikhrokamernikh nagnitachiv [Development of the theory and methods of calculating vortex chamber superchargers]: Dis...d-ra tekhn. nauk: 05.05.17 / KhNADU. — Kharkiv, 2017. — 364 p.
- [13] Krause, R. et al. "Assessment of turbulence models for turbulent flow around a circular cylinder". *Journal of Wind Engineering and Industrial Aerodynamics*, 2022.
- [14] Kozhushko, A. Hydrodynamics analysis on partially filled agricultural tanks by driving cycle of transportation, *International Conference on Reliable Systems Engineering (ICoRSE)*, 762, 2023. https://doi.org/10.1007/978-3-031-40628-7_21
- [15] Kozhushko, A., Pelypenko, Y., Kravchenko, S., Danylenko, V. Improving the Procedure for Modeling Low Frequency Oscillations of the Free Surface Liquid in a Tractor Tank. *Eastern-European Journal of Enterprise Technologies*, 122 (7), 2023, pp. 61-68. <https://doi.org/10.15587/1729-4061.2023.277254>
- [16] Krishnamurthy, A., & Anand, K. "Numerical simulation of compressible turbulent flows using rhoSimpleFoam in OpenFOAM". *Journal of Computational Physics*, 2021.
- [17] Miller, R. K. "Gas Viscosity and the Sutherland Equation" in *Journal of Physical and Chemical Reference Data*, 2018.
- [18] Chase, M. W. et al. "JANAF Thermochemical Tables". *Journal of Physical and Chemical Reference Data*, 1985.
- [19] Reddy, M. K., & Reddy, M. S. "Numerical simulation of 3D turbulent flow using the SST DDES model". *Computers & Fluids*, 2021.
- [20] Kaur, P., et al. "Evaluation of SST DDES model for predicting the aerodynamic performance of wind turbine blades". *Renewable Energy*, 2022.
- [21] Singh, S., & Purohit, R. "Application of SST DDES for predicting flow over complex geometries". *AIAA Journal*, 2020.
- [22] Menter, F. R., & Kuntz, M. "Adaptation of the SST Turbulence Model for High-Speed Flows". *AIAA Journal*, 2016.
- [23] Fiorini, K., et al. "Turbulence Modelling of Curved Flows Using an Enhanced SST Model". *International Journal of Heat and Fluid Flow*, 2020.
- [24] Saleh, M. S. et al. "Investigation of turbulent flow structures using OpenFOAM". *Journal of Computational Physics*, 2021.