TECHNICALUNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 68, Issue Special I, May, 2025

ANALYSIS OF CAVITATION PHENOMENON AS A CRITERION FOR OPTIMIZATION OF CENTRIFUGAL PUMP DESIGN

Viorel BOSTAN, Andrei PETCO

Abstract: *In the given paper, the problems of describing the cavitation process by means of CFD methods are presented. The role of cavitation process study as an optimization criterion of centrifugal pump construction is also described. At the same time the theoretical part related to modeling the cavitation process in centrifugal pumps is discussed. At the end of the paper, the process of determining the NSPH value by means of CFD simulations and comparison of the results with the results of physical testing of a serial pump produced by CRIS Hermetic Pumps is presented.*

Keywords: *Cavitation phenomenon, Centrifugal pumps, CFD, Impeller, Optimization*

1. INTRODUCTION

Centrifugal pumps are an integral part of most fluid handling systems, where they provide efficient transportation of liquids under a variety of operating conditions. Their widespread use is because of their simple design, reliability and ability to pump a big variety of liquids under different conditions and fluid flow regimes [1]. However, under modern conditions of rising energy costs and environmental safety requirements, the need to improve the energy efficiency of pumping equipment is increasing. Increasing energy efficiency is also necessary to meet the commitments undertaken by the European Union under the Paris Agreement on climate change [2]. One of the key directions for improving the efficiency of centrifugal pumps is the optimization of the geometry of its working parts, as this directly affects the hydraulic and energy performance of the pumping unit.

The application of the optimization process starts with setting the optimization criteria and parameters. The choice of optimization criteria for a pump is determined by a complex of factors, including its purpose, operating conditions, design characteristics and economic requirements. Proper application of the optimization process, in turn, results in a pump that performs its function efficiently, with high reliability and cost-effectiveness.

At the same time, it should be noted that geometrical parameters of the pump are often chosen as optimization criteria. The main geometrical parameters include [3]:

- impeller inlet and outlet diameter;
- blade angle (angles at leading and trailing edges)
- number of blades;
- the thickness and shape of the blades;
- impeller flow channel width;
- geometrical parameters of the volute etc.

The following criteria can be used as criteria for optimizing the geometry of working parts [4]:

- the pump’s energy efficiency increase, including increasing their efficiency at nominal flowrate Q_{nom}, both at and within the working range, as well as extending the working range - the range between minimum flowrate Q_{min} and maximum flowrate Q_{max};
- the Net Positive Suction Head required i.e. NPSHr reduction, also one of the main energy indicators;
- reduced hydraulic wear caused by the effects of cavitation and abrasive wear caused by impurities in the pumped liquid;

- passing solid inclusions through the rotor flow area to prevent blockages.
- Reduced vibration and noise, which factor in design improvements for increased comfort and durability;
- balancing rotor axial forces;
- increased reliability and service life, etc.

Selection of an optimization parameters, criterion and constraints plays a vital role [5-7] in the optimization process. Maximum efficiency and minimum NPSH are the most commonly selected optimization criteria [8-10]. However, increasing the efficiency is the main optimization criterion because it reduces the energy required to pump the liquid. But it should also be noted that this criterion is easier to formalize due to its clear structure (eq.1).

$$\eta_h = \frac{\rho g H Q}{\omega N} \quad (1)$$

where: H - pump's head, Q - pump's flow rate, ρ - the density, ω - impeller's angular velocity and N – impeller's torque.

Reducing the cavitation reserve, the second most important optimization criterion, is also related to energy efficiency, allowing to increase the pump's permissible operating range, increase its reliability, preventing the destruction of its elements due to cavitation phenomenon. But the NSPH level is also much more difficult to measure. In this paper some aspects of this problem are presented.

2. DESCRIPTION OF THE CAVITATION PROCESS IN PUMPS

Cavitation in pumps is the physical process of formation and subsequent collapse of vapor bubbles in the liquid in the pump. It should be noted that cavitation is a complex physical process that includes a phase transition with heat and mass transfer and can be thought of as a loss of continuity of liquid flow in a zone of reduced pressure [11].

This effect occurs when the liquid pressure in certain areas of the pump, usually at the impeller inlet and at the volute tongue, drops below saturation pressure, causing the liquid to instantly boil and vapor bubbles to form.

When these bubbles move into the high-pressure zone at the outlet of the pump impeller, they collapse, causing a number of negative effects and, if the system pressure is lower than the cavitation reserve (NPSHR), significantly affecting the performance of the pump [1].

Due to the fact that the cavitation process produced in centrifugal pumps is difficult to describe, modeling it is a challenging task. The cavitation process is modeled using a model based on the Rayleigh-Plesset equation [12].

The Rayleigh-Plesset equation (2) originates from the Navier-Stokes equations. It is a differential equation which describes the dynamics of a spherical bubble in an incompressible fluid.

This equation is used to study various phenomenon, including the formation and collapse of bubbles, the initiation of the cavitation process, bubble oscillations, and the interaction of bubbles with pressure waves or solid walls [13].

$$R\frac{d^2R}{\partial t^2} + \frac{3}{2}\left(\frac{dR}{dt}\right)^2 + \frac{4\nu}{R}\frac{dR}{dt} + \frac{2\gamma}{\rho_L R} + \frac{\Delta P(t)}{\rho_L} = 0 \quad (2)$$

where: R – bubble's radius, ρ_L - liquid density, ν - kinematic viscosity, γ - surface tension (between the bubble and the liquid),

$$\Delta P(t) = P_\infty(t) - P_B(t),$$

where: $P_\infty(t)$ - liquid pressure and $P_B(t)$ is the vapor pressure in the bubble.

Of course, in today's environment, this type of optimization can only be achieved based on Computational Fluid Dynamics (CFD) methods, which open new possibilities for a deep and comprehensive pump geometry optimization, taking into account complex operating conditions.

The integration of modeling and design in Computer-aided Design (CAD) systems and computational experimentation in Computer-aided engineering (CAE) systems allows extensive testing to be reduced significantly, leading to cost and time savings [14].

The cavitation model used in ANSYS CFX relies on the Rayleigh-Plesset equation, the Zwart, Gerber, and Belamri model, described in the paper [15]:

$$\dot{S}_{lv} = \begin{cases} Fvap\dfrac{3r_{nuc}(1-r_v)\rho_v}{R_B}\sqrt{\dfrac{2}{3}\dfrac{P_v-P}{\rho_l}}, & P < P_v \\ Fcond\dfrac{3r_v\rho_v}{R_B}\sqrt{\dfrac{2}{3}\dfrac{P-P_v}{\rho_l}}, & P > P_v \end{cases}$$

The model presented above is based on the multiphase mass transfer rate per unit volume $\dot{S}_{lv}$ calculation, where r_{nuc} presents the nucleation volume fraction.

The Zwart-Gerber-Belamri model is compatible with all turbulence models provided in ANSYS CFX, and liquid and gas phases can be compressible or incompressible.

At the same time, it should be noted that all software used for the analysis of cavitation in pumps is based on models that greatly simplify the complex interaction between the liquid and vapor phase. Assumptions adopted in computational models of mass transfer flow (cavitation) are: the system under study must contain a liquid and a gas phase, mass transfer is achieved both at bubble formation (vapor formation) and at bubble collapse (condensation), and fluid properties can be constant, temperature dependent or user-defined [4].

Generally speaking, according to the level of development of the cavitation process, cavitation can be divided into [13]:

- onset of cavitation, limiting flow regime between non-cavitated and cavitated flow; vapor bubbles affect incipiently cavitated flow very little.
- developed cavitation, which implies a change in flow parameters, when the magnitude of cavitation leads to significant decrease in pump performance.

Namely this developed cavitation state and we need to capture to set the NSPH3 value (critical value). Unfortunately, setting the critical value of NSPH, is an iterative process [1, 16], which directly influences the time to perform the optimization process.

3. ANALYSIS OF THE CAVITATION PROCESS BY CFD METHODS

Although NSPH value determination is time-consuming, we can apply this procedure to a small number of geometric models requiring analysis [16, 17] or to the final phase of the optimized process [4].

3.1 Establishing the geometric model

In this study, the calculation procedure of the NSPH value for the CH 12,5/50-4-2 (Fig. 1) type pump, produced by CRIS Hermetic Pumps Company, Chisinau, Republic of Moldova, is presented.

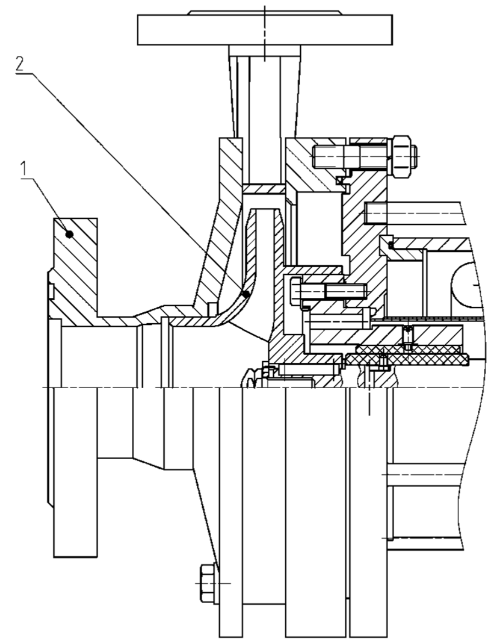

Fig. 1. CH 12.5/50-4-2 pump drawing (pumping part).

To obtain the geometric model, the ANSYS DesignModeler module was applied. The impeller's model was extracted from the geometric model of the pump impeller, so as to be compatible with the TurboGrid mash generator, ANSYS BladeEditor tools were used to perform the export.

3.2 Geometric model discretization

Impeller discretization was performed in ANSYS TurboGrid (Fig. 2), a tool generally created for turbomachinery applications. ANSYS TurboGrid was used, such as it provides automated discretization algorithms. Figure 2 presents a grid created in ANSYS TurboGrid. The discretization grid was created according to the desired parameter y+, equal to 1, at the accepted Reynolds number of fluid flow in the pump's impeller equals to $6 \cdot 10^6$.

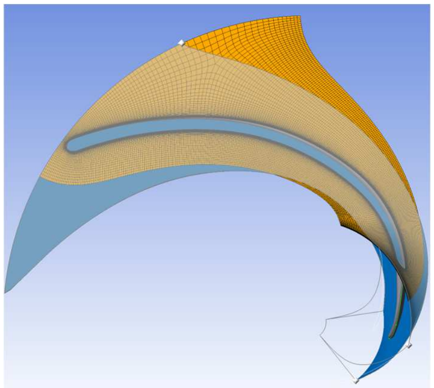

Fig. 2. Discretization grid obtained in TurboGrid.

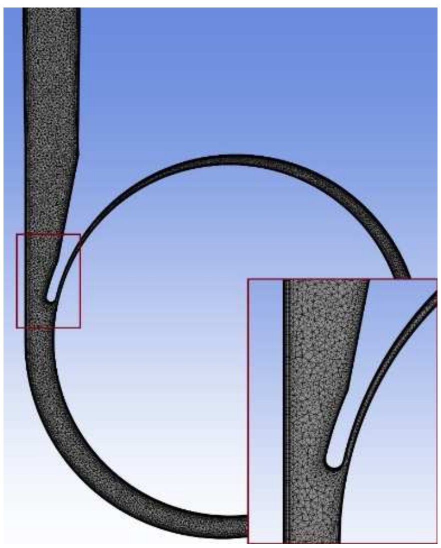

Fig. 3. Pumps volute discretization network obtained in ANSYS.

A discretization grid contains ca. $10.35 \cdot 10^6$ finite volumes.

ANSYS Masher (Fig. 3) has been applied for the discretization network of the suction pipe and volute.

For discretization of the suction pipe the following settings were used maximal size of finite element: ΔS = 2 mm. The inflation layers of 20 finite volumes in depth and 3 mm layer thickness were applied on the suction pipe's surface. The discretization grid with cca. $7.29 \cdot 10^5$ finite volumes was obtained.

For the discretization of the volute the following settings were used maximum finite element size of ΔS = 1mm. Thickening layers of 23 finite volume in depth and 2 mm layer thickness were applied on the volute surfaces. The discretization grid with $4.47 \cdot 10^6$ finite volumes was obtained

3.3 Initial and boundary conditions settings

In the case of this study, the testing process of a centrifugal pump is described in ISO 9906:2012.

In the research, the cavitation characteristics pump test was modeled after a closed scheme. The fluid flow rate was used as a constant, and the reservoir pressure, in the case of CFD simulation of inlet pressure, was used as an independent variable. Acceptance tests of hydraulic performance [18], i.e., by gradually reducing the inlet pressure until the pumping head decreases by 3%, was simulated.

The initial and boundary conditions applied are shown in Figure 4. At the inlet the flow rate was shown to be entirely liquid water and static pressure P_{inlet} equal to NPSH = 17879.6 Pa, also calculations were performed for the value P_{inlet} = NPSH - 0.2 m(H2O) and the value P_{inlet} = NPSH + 0.2 m(H2O). At the outlet, the flow rate is put at a nominal flow rate Q_{out} = 3.47 kg/s. Flow characteristics associated with turbulence and cavitation in the initial state are computed by the solver.

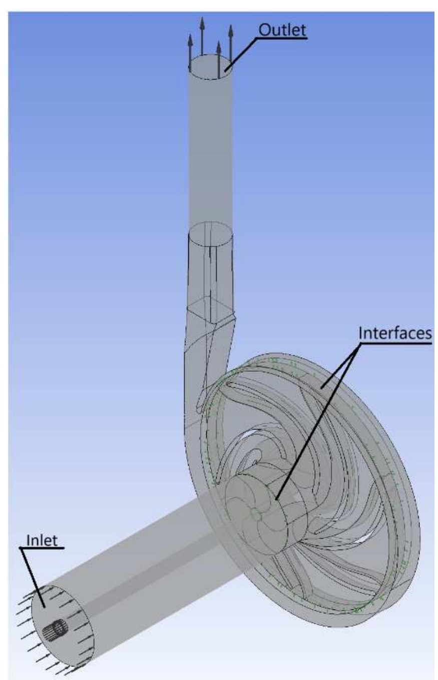

Fig. 4. Initial and boundary conditions applied.

The reference pressure is zero. A rotational speed of 2950 min^{-1} is applied to the impeller's domain. The Wall boundary conditions are used to walls without slip specification. The model disregards the roughness of contact surfaces and gravitational forces. An isothermal model at a fluid temperature of 25°C was also selected.

Initially, the fluid in the computational area consists entirely of liquid water.

The Frozen rotor was chosen as the interface between domains (interface model), which models the interaction between different domains more precise. Contrasted to the Stage model (mixing plane), that only mediates the velocities on the interface surface it displays better interaction.

The Physical Timeline option was selected as the timing management model. The Physical Timeline parameter was selected equal to $1/\omega$ = 0.0032, being optimal for the pump impeller, here: $\omega=2\pi n$ is the angular velocity, according to data described in the paper [19].

The steady-state Reynolds-averaged Navier-Stokes (RANS) approach, namely, the k-omega SST (Menter's Shear Stress Transport) model [20] has been used as a model to describe turbulent flows. The Menter's SST model is the most widely used model for modeling turbulence in turbomachinery [14].

The cavitation model used in ANSYS CFX founded on the Rayleigh-Plesset equation, the Zwart, Gerber and Belamri model. The chosen bubble's radius R_B is $1\cdot10^{-6}$ m, with a vapor saturation pressure of $P_v = 3170$ Pa.

3.4 Analysis of the obtained results

Minimum requirements were chosen to optimize computational sources, which simultaneously ensures a numerical convergence.

A number of 500 computational iterations was selected. The computation is also finished when the tolerance of 10^{-6} for the root mean square residual error is achieved. For the purpose of controlling the convergence, the following indicators were chosen: unbalance, the pump's head and the torque relative to the Z-axis of rotation.

Analyzing the results of the calculation, we can see that the y^+ parameter's value is in the range of units (Fig. 5).

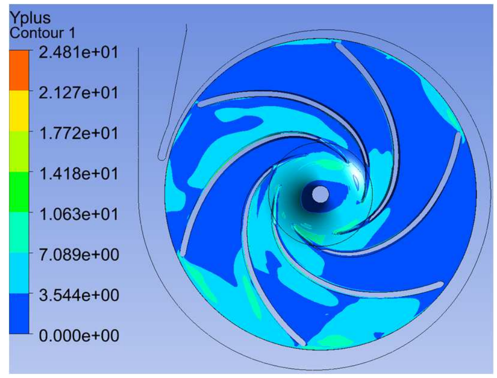

Fig. 5. Value of parameter y^+.

At the same time, it should be noted that cavitation cavities were also obtained. The gray areas in Figures 6-8 represent areas where the volume of liquid vapor exceeds that of the liquid itself, i.e., the volume of liquid vapor exceeds 50%.

Figure 6 shows the cavity obtained at the inlet pressure Pinlet = NPSH + 0.2 m(H2O). The pumping head of the pump does not deviate much from the nominal H = 50 m(H2O).

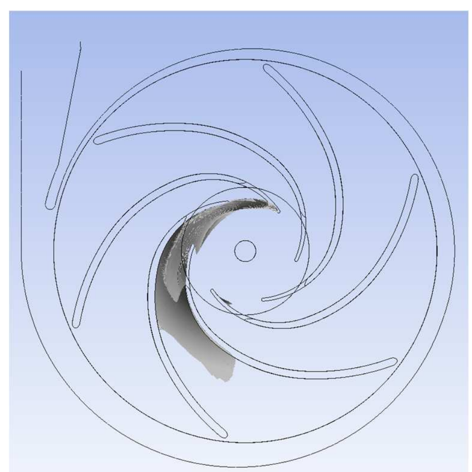

Fig. 6. Cavitation cavity obtained at P_{inlet} = NPSH + 0.2 m(H2O).

Figure 7 shows the cavity obtained at inlet pressure Pinlet = NPSH. The pumping head decreases approximately 1-1.5 m(H2O).

Figure 8 shows the cavity obtained at inlet pressure Pinlet = NPSH - 0.2 m(H2O). In this case we can observe the sudden decrease of the pumping head. We can observe massive increase of cavitation cavity. At the same time due to the pronounced cavitation phenomenon problems with calculation convergence are growing.

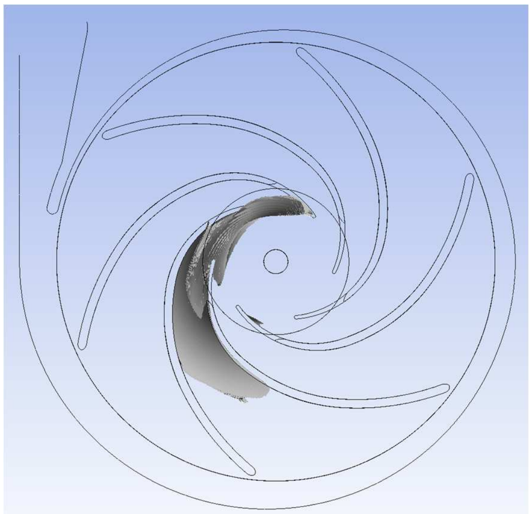

Fig. 7. Cavitation cavity obtained at P_{inlet} = NPSH.

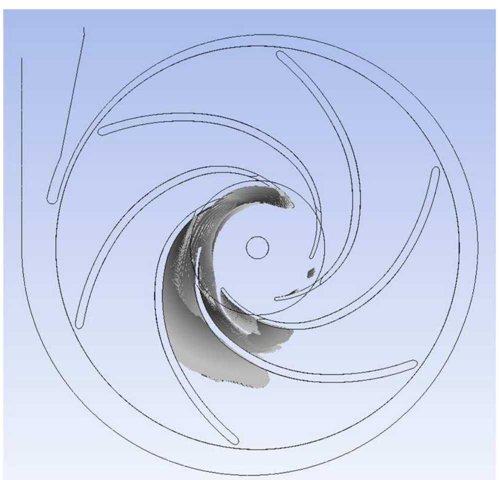

Fig. 8. Cavitation cavity obtained at P_{inlet} = NPSH - 0.2 m(H2O)

It should also be noted that the asymmetric location of the cavitation spot obtained during the simulation is explained by the structure of the pressure distribution field at pressure P_{inlet} = 17879.6 Pa. In Figure 9, one can notice a significant shift of the underpressurized zone towards the lower part of the wheel, filling in turn almost completely one of the inter-blade channels. This distribution appears to be due to the asymmetrical design of the pump, namely the tangential positioning of the discharge connection of the pump casing.

4. FURTHER RESEARCH

As a continuation of the study, it is proposed to switch the calculation from Steady-State to Transient model and also to compare the obtained data with NSPH determination models based on the analysis of the blade's surface pressure values.

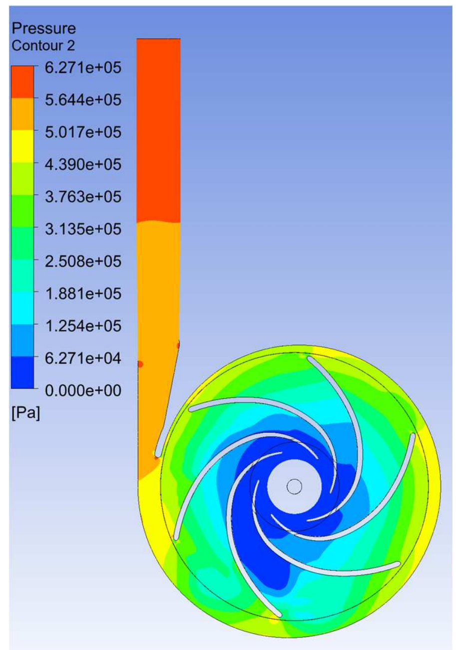

Fig. 9. Pressure distribution field at the inlet pressure P_{inlet} = 17879.6 Pa

5. CONCLUSION

The problem of optimizing centrifugal pump impellers, as well as other centrifugal pump working parts, remains relevant for the pump industry. At the same time, it should be noted that the study of pump cavitation characteristics is becoming a difficult task for specialists in this field. It should be noted that:

1. The application of CFD technology allows visualization of the shape of the cavity, which in turn provides more information for optimizing pump impellers.
2. Despite the absence of a fast and stable method to determine the NSPH, by

calibrating the calculation model to the test results of existing pumps, we can determine the NSPH level based on CFD modelling.
3. The tuned model allows us to determine, with an approximation of 0.1-0.2 mH2O, the value of the critical NSPH3.

6. ACKNOWLEDGMENT

The authors gratefully acknowledge the support of the Institutional research program of the Technical University of Moldova: 020403. Performance development of machine drive mechanisms based on precessional transmissions, mechanical systems and magnetic transmissions. Program head: Acad. Ion Bostan. Period: 2024-2027**.**

The authors are also grateful to the staff of FabLab Chișinău, for research assistance and support.

The authors also expresses sincere thanks to CRIS Hermetic Pumps Company.

7. REFERENCES

[1] Gülich, J.F., *Centrifugal Pumps*. Springer International Publishing, ISBN: 978-3-030-14787-7, Cham, 2020.

[2] European Commission, *Reflection paper. towards a sustainable Europe by 2030.* https://commission.europa.eu/publications/sustainable-europe-2030.

[3] Bostan, V., Petco, A., *Minimizing blade-fluid energy energy losses in centrifugal hydraulic pump impellers*, ACTA Tech. Napoc. Appl. Math. Mech. Eng. vol. 67, no. 2S, Oct. 2024

[4] Petco, A., *Increasing the energy efficiency of centrifugal pumps through mathematical modeling and numerical computation of fluid flow*, Technical University of Moldova, Chisinau, https://anacec.md/files/Petco-teza.pdf, 2024.

[5] Moisă, I. G., Susan-Resiga, R., Muntean, S., *Pump inducer optimization based on cavitation criterion*, Proceedings of the Romanian academy, series A, Volume 14, Number 4/2013, pp. 317–325, The Publishing House Proceedings of The Romanian Academy, 2013.

[6] Parikh, T., Mansour, M., Thévenin, D., *Maximizing the performance of pump inducers using CFD-based multi-objective optimization*, Struct. Multidiscip. Optim. vol. 65, no. 1, p. 9, Dec. 2021.

[7] Zhang, Y., Hu, S., Wu, J., Zhang, Y., Chen, L., *Multi-objective optimization of double suction centrifugal pump using Kriging metamodels*, Adv. Eng. Softw. vol. 74, pp. 16-26, Aug. 2014.

[8] Abdolahnejad, E., Mahdi, M., Shahram, D., *Optimization of Centrifugal Slurry Pump Through the Splitter Blades Position*, 2021.

[9] 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 10 (9) 774, 2022.

[10] Mălăel, I., Gherman, G.B., *Numerical Investigation of a New LH2 Centrifugal Pump Concept Used in Space Propulsion*. INCAS BULLETIN 10 (June), pp. 65–74, 2018.

[11] Li, S.C., *Cavitation of Hydraulic Machinery*, vol. 1. in Series on Hydraulic Machinery, Imperial College Press, ISBN 1860942571, London, 2000.

[12] Franc, J.-P., *The Rayleigh-Plesset equation: a simple and powerful tool to understand various aspects of cavitation*, in *Fluid Dynamics of Cavitation and Cavitating Turbopumps Springer*, pp. 1-41, ISBN 978-3-211-76668-2 Cham, 2007.

[13] Franc, J.-P., Michel, J.-M., *Fundamentals of Cavitation*, Springer Netherlands, ISBN 978-1-4020-2232-6, Dordrecht, 2005.

[14] Bostan, V., *Mathematical models in engineering: contact problems: numerical modeling and simulation in aero-hydrodynamics.*, S.N., ISBN 978-9975-80-831-6, Chisinau, 2014.

[15] Zwart, P., Gerber, A. G., Belamri, T., *A two-phase flow model for predicting cavitation dynamics*, Fifth Int. Conf. Multiph. Flow, ICMF 2004 International Conference on Multiphase FlowYokohama, Japan, May 30-June 3, 2004.

[16] Xie, S., Wang, Y., Liu, Z., Zhu, Z., Ning, C., Zhao, L., *Optimization of centrifugal pump cavitation performance based on CFD*, International Symposium of Cavitation and Multiphase Flow, ISCM 2014, IOP Conf. Series: Materials Science and Engineering 72, IOP Publishing, 2015

[17] Qiu, N., Wang, L., Kong, F., Wu, D., *Research on cavitation characteristic of inducer*, IOP Conf. Ser. Mater. Sci. Eng, vol. 52, Nov. 2013.

[18] International Organization for Standardization, *ISO 9906:2012 Rotodynamic pumps - Hydraulic performance acceptance tests - Grades 1, 2 and 3,* iso.org/standard/41202.html

[19] Bostan, V., Petco, A., *Determining optimal simulation settings for the centrifugal pump parts optimization process.*, *J. Eng. Sci.* no. 30 (2), pp. 8-22, 2023.

[20] Menter, F. R., *Two-equation eddy-viscosity turbulence models for engineering applications*, AIAA J., vol. 32, no. 8, pp. 1598-1605, 1994.

Analiza fenomenului de cavitație ca criteriu de optimizare a construcției pompelor centrifuge

În articolul dat sunt prezentate problemele de descriere a procesului de cavitație prin intermediul metodelor CFD. De asemenea este prezentată și rolul studiului cavitației în calitate de criteriu de optimizare ale construcției pompelor centrifuge. Totodată este prezentată partea teoretică ce ține de modelarea procesului de cavitație în pompele centrifuge. La finele lucrării este prezentat procesul de stabilire a valorii NSPH prin intermediul simulărilor CFD și compararea rezultatelor cu rezultatele testării fizice ale unei pompe de serie produse de CRIS Hermetic Pumps.

Viorel BOSTAN, PhD, Professor, Technical University of Moldova, Faculty of Computers, Informatics and Microelectronics, viorel.bostan@adm.utm.md, Rep. Moldova, Chisinau, 168 Stefan cel Mare Avenue, Rectorate.

Andrei PETCO, PhD, Lecturer, Technical University of Moldova, Faculty of Mechanical and Industrial Engineering and Transport, Design Engineer, S.R.L. "CRIS", andrei.petco@tcm.utm.md, Rep. Moldova, city Chisinau, 9/8 Studentilor str., off. 6-214.