UDC 621.454.2
Doi: 10.31772/2587-6066-2020-21-3-417-422

For citation: Torgashin A. S., Zhujkov D. A., Nazarov V. P., Begishev A. M., Vlasenko A. V. CFD methods for cavitation modeling in centrifugal and axial pumps of LRE. *Siberian Journal of Science and Technology.* 2020, Vol. 21, No. 3, P. 417–422. Doi: 10.31772/2587-6066-2020-21-3-417-422

Для цитирования: Методы CFD моделирования кавитации в центробежных и осевых насосах жидкостных ракетных двигателей / А. С. Торгашин, Д. А. Жуйков, В. П. Назаров, А. М. Бегишев, А. В. Власенко // Сибирский журнал науки и технологий. 2020. Т. 21, № 3. С. 417–422. Doi: 10.31772/2587-6066-2020-21-3-417-422

CFD METHODS FOR CAVITATION MODELING IN CENTRIFUGAL AND AXIAL PUMPS OF LRE

A. S. Torgashin*, D. A. Zhujkov, V. P. Nazarov, A. M. Begishev, A. V. Vlasenko

Reshetnev Siberian State University of Science and Technology
31, Krasnoyarskii rabochii prospekt, Krasnoyarsk, 660037, Russian Federation
*E-mail: ttarg23@yandex.ru

Currently, design and manufacture of liquid-propellant rocket engines (LRE) are imposed with ever greater reliability requirements. Accordingly, the standards for the design and manufacture of rocket engine units are raising. One of these units is a turbopump unit (TNA), which provides continuous supply of liquid components from combustion reaction to the combustion chamber of a rocket engine to create traction or other engine units. TNA is also the main source of pressure increase for these liquid components in front of the LRE combustion chamber. Important requirements are imposed on a turbopump unit (TNA): ensuring work performance and basic parameters for a given resource with the necessary possible pauses of a specified duration; providing all engine operating modes, supplying the fuel components of the required flow rate and pressure, guarantying a high degree of reliability with acceptable entire unit efficiency; providing high anti-cavitation characteristics of the pump in all modes. In the article, the authors summarize the latest results of the study on cavitation in turbopump units of liquid propellant rocket engines alongside with the relevant research in the field of hydraulics. The problems of cavitation in cryogenic liquids, simulation of stall characteristics, and usability of various models to simulate cavitation flow are observed. A solution to the problems of flow modeling was considered with respect to applicability to the following structural elements of LRE units: interscapular space of the screw centrifugal main and booster pumps, axial pre-pump. Particular attention is paid to the implementation of various numerical methods based on the use of various cavitation models, computational fluid dynamics in various CFD packages, and also comparison of results with the model. In summary, the authors draw conclusions about the possibility of applying these methods to solve the problems of the cavitation phenomenon research in LRE.

Keywords: Cavitation, TNA, LRE, CFD modeling.

МЕТОДЫ CFD МОДЕЛИРОВАНИЯ КАВИТАЦИИ В ЦЕНТРОБЕЖНЫХ И ОСЕВЫХ НАСОСАХ ЖИДКОСТНЫХ РАКЕТНЫХ ДВИГАТЕЛЕЙ

А. С. Торгашин*, Д. А. Жуйков, В. П. Назаров, А. М. Бегишев, А. В. Власенко

Сибирский государственный университет науки и технологии имени академика М. Ф. Решетнева
Российская Федерация, 660037, г. Красноярск, просп. им. газ. «Красноярский рабочий», 31
*E-mail: ttarg23@yandex.ru

В настоящее время к проектированию и производству жидкостных ракетных двигателей (ЖРД) предъявляются все большие требования по обеспечению надежности. В соответствии с этим повышаются требования по проектированию и изготовлению агрегатов ЖРД. Одним из таких агрегатов является турбонасосный агрегат (ТНА), обеспечивающий непрерывную подачу жидких компонентов реакции горения в камеру сгорания ракетного двигателя для создания тяги или в другие агрегаты двигателя. Также ТНА является основным источником повышения давления данных жидких компонентов перед камерой сгорания ЖРД. К ТНА предъявляются важные требования по обеспечению работоспособности основных параметров при заданном ресурсе с необходимыми возможными паузами установленной продолжительности; подачи компонентов топлива требуемого расхода и давления на всех режимах работы двигателя; высокой степени надежности с приемлемым КПД всего агрегата; высоким антикавитационным характеристикам насоса на всех режимах. В данной статье авторы обобщают последние результаты исследования кавитации в турбонасосных агрегатах ЖРД, а также применимые к ним исследования в области гидравлики. Рассмотрены проблемы кавитации в криогенных жидкостях, моделирование срывной характеристики, применение различных моделей к модели-

рованию кавитационного потока. Решение данных проблем моделирования течения рассматривалось относительно применяемости к следующим элементам конструкции агрегатов ЖРД: межлопаточного пространства шнекоцентробежного основного и бустерных насосов, осевого преднасоса. Особое внимание уделено реализации различных численных методов, основанных на использовании различных моделей кавитации, вычислительной гидрогазодинамики в различных CFD пакетах, а также сравнении результатов с модельными. Авторы делают выводы о возможности применения данных методов к решению вопросов исследования явления кавитации в ЖРД.

Ключевые слова: кавитация, ТНА, ЖРД, CFD моделирование.

Introduction. Cavitation in hydrodynamics is a special case of liquid boiling (phase transition of a liquid into a gas inside a liquid at a certain temperature and pressure), which occurs in moving liquid due to local pressure reductions to the level of saturated steam pressure. In hydrodynamics the phenomenon of cavitation plays a negative role as it causes violation of medium homogeneity. As a result, there may be bubbles in the flow that break when in contact with the blades, which can lead to a hydraulic shock that destroys the blades. It is important to take into account that the cavities connecting the units, movable parts of pumps and turbines, supply pipelines are elements of a complex spatial structure, geometry of which also affects the flow. In the working parts of the pump, the pressure reduction inside the flow part is connected with the streamlining of the blade profiles, where, as with the streamlining of any profile, a cavity of reduced pressure is formed at the inlet from the rear (non-working) side. This region of minimum pressure is the region of cavitation origin. The higher velocity of the flow, flowing around the blade is, the greater will be the discharge on the blade. Therefore, the most distant point from the axis of rotation of the blade leading edge may be the generation center of cavitation. Cavitation during TNA LRE operation can lead to three main negative consequences:

a) failure of TNA operation modes, i.e. sharp decrease of main output parameters – head, flow rate and, as a result, efficiency;

b) to the collapse of steam cavitation bubbles in the area of the blades, accompanied by strong blows, helps to destruct the blades of the machine impeller – erosion destruction. This phenomenon is usually manifested during long-term operation in cavitation mode, during the operation of the TNA LRE;

c) to occurrence of low-frequency self-oscillations due to possible unstable operation of TNA LRE [1].

To determine cavitation conditions in theoretical and experimental studies, the following non-dimensional parameter is used:

$$C_a = \frac{p - p_v}{\frac{1}{2}\rho U^2}, \tag{1}$$

where p is flow pressure (for example, the input pressure), p_v is the saturated vapor pressure for the liquid, denominator is the dynamic pressure. This coefficient represents the pressure difference at a point of the body and in an undisturbed fluid at a certain distance from it and is proportional to the square of the velocity of the relative motion. The definition of the main condition for reducing the pressure to the minimum at which cavitation begins is also derived from it.

In 1917 Lord Rayleigh published the article “On the pressure, developing in a liquid when a spherical cavity” [2]. Rayleigh used the formulation of the problem on an empty cavity in a homogeneous liquid at constant pressure at infinity, proposed by Besant in 1859: “An infinitely large mass of a forced homogeneous incompressible fluid is at rest. The liquid inside a certain spherical surface instantly disappears. It is required to find the instantaneous change in pressure at any point of the liquid and the filling time of the cavity, assuming that the pressure at infinity is constant”. The solution to this problem is based on the complete transformation of the work done by the mass when the cavity collapses into kinetic energy. The resulting equation:

$$\rho(RR_{TT} + \frac{3}{2}R_T^2) = -p, \tag{2}$$

where p is the pressure at infinity, R is the bubble radius, ρ is the density of the fluid around the cavity, R_T and R_{TT} are the derivatives of the radius with respect to time. There is also a version of the Rayleigh equation for a gas-filled bubble, in which the condition is accepted that the gas does not exchange heat with the liquid, which means that its state is described by the Poisson pV^γ = const:

$$\rho(RR_{TT} + \frac{3}{2}R_T^2) = P_0\left(\frac{R_0}{R}\right)^{3k} - p, \tag{3}$$

where κ is the polytropic index, P_0 is the atmospheric gas pressure in the bubble, and R_0 is the atmospheric radius of the bubble. Note that equations (1) and (2) do not take into account the influence of the gas cavity contents, surface tension, viscosity, and compressibility. Also, for these equations, it is assumed that the pressure at the distance from the bubble is also constant. At the moment, various generalizations of the Rayleigh equation are used to solve hydrodynamic problems.

Currently, significant work has been carried out on the study of the Rayleigh-Plesset model applicability for solving the problems of cavitation flow modeling in the channels of the LRE pumps. The Rayleigh-Plesset equation provides the basis for the flow equation that controls steam formation and condensation. The Rayleigh-Plesset equation describing the growth of a gas bubble in liquid is derived from the equations of moments:

$$R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2}\left(\frac{dR_B}{dt}\right)^2 + \frac{2\sigma}{\rho_f R_B} = \frac{p_v - p}{\rho_f}, \tag{4}$$

where R_B represents the bubble radius, p_v is the pressure in the bubble (it is assumed that this is the vapor pressure at the liquid temperature), p is the pressure in the liquid surrounding the bubble, ρ_f is the liquid density, and σ is the coefficient of surface tension between the liquid and the vapor. In the practices of cavitation flows modeling with various software packages simplified modifications of the Rayleigh-Plesset equation are usually used. For example, in the ANSYS CFX package, the second-order terms (which are suitable for low vibration frequencies) and surface tension are neglected [3].

In addition to numerical studies of cavitation flows in the pump, general solutions of the Rayleigh and Rayleigh-Plesset equations were obtained in [4] and [5], respectively.

Modern approaches to modeling the cavitation flow. Let us consider the approaches used for modeling the cavitation flow in pumps applied in various areas of hydraulic engineering. The main attention will be paid to a representative comparison with real tests with reference to the used cavitation models. Over the past several decades due to the growth of computational capacities many tests have been carried out the results of which allow to select the most suitable cavitation model for modeling the flow in the pump TNA LRE.

In [6] the authors consider the possibility of applying the Rayleigh-Plesset model to modeling the flow of cryogenic components. Since this work was carried out directly in application to the TNA LRE unit, it is also necessary to highlight the following comments and assumptions given by the authors:

– lack of consideration of thermal effects affecting the development of the cavity, development of which according to [6] is currently underway;

– insufficient amount of empirical data on cryogenic fluids;

– calculation used a model with three interscapular channels.

The work was carried out in the ANSYS CFX environment. Importantly, in addition to the cavitation equation discussed above, the ANSYS CFX model also includes a k-ω turbulence model, which also affects the numerical simulation results. Comparison in the article is based on the total volume of cavitation cavities. In contrast to the results discussed further, the text only mentions the results of full-scale tests of the screw, but no numerical data or percentage comparison is given. However, the authors note that despite the fact that heat effects are not taken into account in the Rayleigh – Plesset formula, good convergence of the data was obtained from numerical modeling. Also, the authors carried out a numerical simulation of the cavitation flow with an improved screw.

In [7] a technique for modeling a stalling cavitation flow in a booster turbopump unit is considered. In this article construction of a cavitation model is also based on the application of the Rayleigh-Plesset model. The work, as well as [4], was carried out in the ANSYS environment. The authors considered the method of setting the parameters for constructing computational grid, setting initial conditions, and also compared the results obtained for models with and without a gap, as well as with field tests. The discrepancy with the model test for the model with a gap in the case of calculating the cavitation headroom was about 15 %, without a gap – 10 %. However, the clearance model showed better head convergence with the model tests. Also, the model without a gap does not take into account the vortex component of the flow. The authors conclude that the model with a gap is more applicable to modeling cavitation flow than without it.

Articles [8–12] were also considered in [7], where the authors compared the results obtained with the results of the authors of [8–12], however, it also seems necessary to consider cavitation models applicable in them and the authors' conclusions about the possible reasons for the discrepancies in the modeling results and model tests.

In the article [8], the authors consider development and application of numerical methods for cavitation modeling. The work was carried out in the CFX TASC flow 2.12 software environment. Currently the developers of this software environment are part of ANSYS. The constructed model was compared with the results obtained in model tests of the National Graduate School of Arts and Crafts. In the preface, the authors note that for flows with large accelerations, it is necessary to use a solution method based on solving the conservation equations for each of the phases and used in the development of non-equilibrium conditions for the exchange of heat, mass, momentum between phases. Moreover, the models will contain certain assumptions. The model used by the authors in [8] is based on the use of a non-equilibrium approximation in order to reduce the number of equations to be solved. According to the simulation data, in comparison with the real experiment, the results of the numerical model converge agree with the indicators of the ratio of expenses (current to nominal) from 0.91 to 1.09. In this work, a simplified Rayleigh – Plesset equation was used, which is similar to the equation used in the ANSYS CFX software environment.

The article [9] also discusses the problem of cavitation numerical simulation. The authors emphasize that at the time of 2003 the problem of flow optimization by reducing cavitation was rarely used and mainly suggested optimizing the flow angle at the pump input. This article discusses several software packages CFX-Tascflow, FLUENT and STAR-CD. FLUENT, like CFX-Tascflow, is currently part of the ANSYS package and allows the use of Zwart-Gerber-Belamri, Schnerr and Sauer and Singhal et al. The STAR-CD package is also in production today and uses the Rayleigh-Plesset cavitation model. At the time of this article writing, none of them take into account three-dimensional turbulent flow and viscosity effect. Based on the simulation results, the authors note that of the three selected programs, CFX-Tascflow shows the most accurate results in comparison with the model test. Also, the most accurate results obtained in CFX-Tascflow were compared with a simplified method based on the use of the initial shape of the cavitation cavity. This model is described in detail in [9]. Both methods are good at predicting the starting level and the percentage of head drop. Based on the comparison results, the authors conclude that none of the proposed software packages at the time of 2003 justifies the time spent on the calculation, as compared to the achieved level of accuracy. However, CFX-Tascflow simulation data remains relatively accurate.

Article [10] addresses the issues of cavitation modeling in a diagonal centrifugal pump. In the article the authors employ the commercial package ANSYS CFX and clarify that they use the ANSYS CFX model without changes, that is, a model based on the VOF (volume of fluid) method. The aim of the study was to test the applicability of this commercial package as part of a standard engineering study. In terms of building a spatial model and a method of breaking it up into elements, the authors note that the hexahedral mesh is preferable to the tetrahedral one when modeling flows in turbopump units. Also in the model moving and stationary parts of the mesh are connected using the frozen-rotor interface. As in the work of previous authors, the studies also consider cavitation at higher, equal or lower than the nominal pump flow rate. In comparison with real tests the authors note that cavitation in the model begins with a lower number of cavitations and pressure drop is steeper. They attribute this to the lack of accounting for instability in the numerical model used. Under overload conditions at a flow rate of 1.25 of the nominal flow rate, the model showed the results that are closest to the real experiment. Analyzing the data obtained, the authors confirm that the model built in ANSYS CFX shows accurate results in terms of displaying the position, size of the cavitation cavity, and also emphasize the importance of using meshes based on hexahedral elements.

In the article [11], the authors consider numerical simulation of cavitating flow performed according to the standard model built into ANSYS CFX. The authors compared the results of numerical simulation with the standard model used in ANSYS CFX, as well as with the model with a modified k-ω turbulence model, which takes into account the cavity compressibility in the cavitation flow. Also, the Schnerr-Sauer cavitation model is used for the modified k-ω turbulence model. The comparison was also carried out with real model tests and showed that, despite discrepancies between the real cavitation curve and the one built on the basis of numerical simulation, the modified k-ω model gives results that are closer to real data. However, at a higher flow rate the pressure drop curves obtained using the two models have practically no differences. Apart from that, in comparison with model tests cavitation occurs with a lower number of cavitations and the total head is higher in both numerical models. The high head is explained by the imperfection of the model and the lack of accounting for flow losses that inevitably arise during the real pump operating. Based on the comparison results, the authors concluded that the use of the Schnerr-Sauer model can improve the accuracy of cavitation flow modeling.

In the article [12], the authors analyze the effect of cavitation flows on screws with different blade geometry. A number of studies have been carried out for each of the three geometry options. In their work, the authors used the Fluent software package. In the mass transfer model, the cavitation rate is also based on the simplified Rayleigh-Plesset model. Before working with pumps, the authors did preliminary work in terms of studying cavitation, first in Venturi tubes, then in two-blade cascades and axial screws. The type of emerging cavitation and the development of cavitation during the test were determined for all of the above geometries. In terms of a more accurate behavior of the flow characteristics, considering the comparison of the stable flow of model tests and numerical modeling: similar to previous studies, the numerical model shows the onset of cavitation at large cavitation numbers for all three pumps tested by the authors. The authors also note that the results are in good agreement. The factors influencing the discrepancies between model tests and numerical modeling, according to the authors' conclusions, are: modeling of only one interscapular channel and, as a consequence, the lack of consideration of interactions between the channels, in comparison with the real geometry of the model is ideal without defects and deviations, as well as a numerical model does not take into account the radial clearance.

The article [13] presents the development and numerical simulation of cavitation flow based on a set of open source tools in OpenFOAMR, allowing the use of various cavitation models. The authors carried out work considering applicability of various cavitation models to the problem of cavitation flow modeling in the pump. In the article, the authors reviewed 4 models: Kunz et al, Merkle et al., Schnerr-Sauer-Yuan and Zwart et al. As in all previous studies, a 3 % head drop due to cavitation was modeled. Of the above models, the Zwart model first showed the best results on the airfoil and was then used to simulate cavitation flow. Comparison with model tests also showed high head values, but low values of the cavitation volume for the numerical model compared to the experimental one. The authors explain this discrepancy by the fact that the material used to create the impeller has a low stiffness and could deform during a real experiment, which cannot be taken into account in the numerical model; pump operation in a numerical model.

In article [14] the authors consider unsteady flow in the pump, as well as structural calculation of strength based on hydraulic calculation. Despite the fact that the main topic of this article is the analysis of the possibility of realizing the combination of interrelated strength and hydraulic calculations in the software package, the authors also carried out research in the field of cavitation modeling. When carrying out hydraulic calculations, a comparison of model tests and a numerical model was also carried out. The authors used the Zwart – Gerber – Belamri model, also based on the simplified Rayleigh – Plesset model. This model was also used by the authors of the article [13]. The results presented by the authors show good agreement with model tests. The numerical model of cavitation shows the onset of stalling at large cavitation numbers, in comparison with model tests, at a pump flow rate equal to the nominal (1.0 Q_d). Note that in this study, the values of the head during the real test and the results of the numerical model are close to each other.

The article [15] considers pump impeller optimization in order to increase productivity. The work was carried out using the CFturbo 9.0 software. Despite the fact that the main topic of this article is the pump efficiency increase, the authors also carried out tests in terms of determining the cavitation characteristics. Prior to presenting test results with already optimized geometry, the authors also compared numerical and model tests results. Provided that a two-phase vapor-liquid model is used with a k-ε turbulence model and a constant mass fraction of gas, based on the already Rayleigh-Plesset equation, the numerical simulation data give almost identical results in comparison with model tests.

Conclusion. Having considered all the above works along with conclusions made by the authors, it is possible to define a number of factors that affect the accuracy of cavitation modeling in software packages, which must be taken into account:

– to increase the simulation accuracy it is necessary to take into account the working fluid leaks during the operation of the TNA pump;

– it is important to take into account the clearance between the blades and the casing wall in geometric models;

– when choosing a numerical model, it is necessary to consider both the model of cavitation and turbulence.

In summary, the results of [13] and [14] show good convergence of the numerical model and real experiment, which allows to conclude that the Zwart – Gerber – Belamri model, also based on the Rayleigh – Plesset model, is more applicable for cavitation flow simulating in a pump. This line of research is relevant, but insufficiently developed for application in engineering calculation methods and design of TNA LRE, especially for obtaining numerous options for more advanced designs at the initial stages of new models of rocket engines development.

References

1. Gahun G. G., Baulin. V. I., Volodin V. A. et al. *Konstrukciya i proektirovanie zhidkostnyh raketnyh dvigateley* [Construction and design of liquid rocket engines]. Moscow, Mashinostroenie Publ., 1989, 424 p.

2. Lord R. On the pressure developed in a liquid during the collapse of a spherical cavity. *Phil. Mag.* 1917, No. 34 (200), P. 94–98.

3. ANSYS CFX Tutorial Guide. Chapter 28: Drop Curve for Cavitating flow in a Pump. ANSYS Inc. Release 15.0.

4. Kudryashov N. A., Sinelshchikov D. I. Analytical solutions of the Rayleigh equation for empty and gas-filled bubble. *J. Phys. A: Math. Theor.* 2014. No. 47. P. 405202.5.

5. Mancas Stefan C., Rosu Haret C. Cavitation of spherical bubbles: closed-form, parametric, and numerical solutions. Physics of Fluids. 2016.

6. Afanasyev A. A., Demyanenko Yu. V., Popkov A. N. [Application of the Rayleigh-Plesset cavitation model for studying the flow of cryogenic liquid in the paths of a screw-center-run pump]. *Vestnik voronezhskogo gos. tekhn. un-ta.* 2017, Vol. 12, No. 2, P. 44–49 (In Russ.).

7. Kazyonnov I. S., Kanalin Yu. I., Poletaev N. P., Chernysheva I. A. [Modeling of the breakdown cavitation characteristics of a booster turbopump unit and comparison of experimental and numerical results]. *Vestnik samarskogo gos. aerokosmich. un-ta im. ak. S. P. Koroleva (Nats. issled. un-ta).* 2014, No. 5-1 (47), P. 188–198 (In Russ.).

8. Bakir F., Rey R., Gerber A. G., Belamri T., Hutchinson B. Numerical and Experimental Investigations of the Cavitating Behavior of an Inducer. *International Journal of Rotating Machinery*. 2004, No. 10, P. 15–25.

9. Dupont P., Okamura T. Cavitating Flow Calculations in Industry. *International Journal of Rotating Machinery*. 2003, No. 9 (3), P. 163–170.

10. Pierrat D., Gros L., Couzinet A., Pintrand G. On the Leading Edge Cavitation In a Helico-centifugal Pump: Experimental and Numerical Investigations. *3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems*. October 14–16, 2009, Brno, Czech Republic.

11. Application of modified k-ω model to predicting cavitating flow in centrifugal pump / Hou-lin Liu*, Dong-xi Liu, Yong Wang, Xian-fang WU, Jian Wang // Water Science and Engineering. 2013, No. 6(3), P. 331–339.

12. Rafael Campos-Amezcua, Sofiane Khelladi, Zdzislaw Mazur-Czerwiec, Farid Bakir, Alfonso Campos-Amezcua and Robert Rey (2011). Numerical and Experimental Study of Mass Transfer Through Cavitation in Turbomachinery, Mass Transfer – Advanced Aspects, Dr. Hironori Nakajima (Ed.), InTech.

13. Hanimann L., Mangani L., Casartelli E., Widmer M. Cavitation modeling for steady-state CFD simulations. *IOP Conf. Ser.: Earth Environ.* 2016.

14. Denghao Wu, Yun Ren, Jiegang Mou, Yunqing Gu, Lanfang Jiang. Unsteady Flow and Structural Behaviors of Centrifugal Pump under Cavitation Conditions. Chinese. *Journal of Mechanical Engineering*. 2019.

15. Pei Ji1, Yin Tingyun1, Yuan Shouqi1, Wang Wenjie1, Wang Jiabin. Cavitation Optimization for a Centrifugal Pump Impeller by Using Orthogonal Design of Experiment. *Chinese Journal of Mechanical Engineering*. 2017, Vol. 30, No. 1, P. 103.

Библиографические ссылки

1. Конструкция и проектирование жидкостных ракетных двигателей / Г. Г. Гахун, В. И. Баулин, В. А. Володин и др. ; под общ. ред. Г. Г. Гахуна. М. : Машиностроение, 1989. 424 с.

2. Lord R. On the pressure developed in a liquid during the collapse of a spherical cavity // Phil. Mag. 1917. No. 34 (200). P. 94–98.

3. ANSYS CFX Tutorial Guide. Chapter 28: Drop Curve for Cavitating flow in a Pump. ANSYS Inc. Release 15.0.

4. Kudryashov N. A., Sinelshchikov D. I. Analytical solutions of the Rayleigh equation for empty and gas-filled bubble // J. Phys. A: Math. Theor. 2014. No. 47. P. 405202.

5. Mancas Stefan C., Rosu Haret C. Cavitation of spherical bubbles: closed-form, parametric, and numerical solutions. Physics of Fluids. 2016.

6. Афанасьев А. А., Демьяненко Ю. В., Попков А. Н. Применение кавитационной модели Релея – Плессета для исследования течения криогенной жидкости

в трактах шнекоцентробежного насоса // Вестник воронежского гос. техн. ун-та. 2017. Т. 12, № 2. С. 44–49.

7. Моделирование срывной кавитационной характеристики бустерного турбонасосного агрегата и сравнение экспериментальных и численных результатов / И. С. Казённов, Ю. И. Каналин, Н. П. Полетаев, И. А. Чернышева // Вестник самарского гос. аэрокосмич. ун-та им. ак. С. П. Королёва (Нац. исслед. ун-та). 2014. № 5-1 (47). С. 188–198.

8. Numerical and Experimental Investigations of the Cavitating Behavior of an Inducer / F. Bakir, R. Rey, A. G. Gerber, T. Belamri, B. Hutchinson // International Journal of Rotating Machinery. 2004. No. 10. P. 15–25.

9. Dupont P., Okamura T. Cavitating Flow Calculations in Industry // International Journal of Rotating Machinery. 2003. No. 9 (3). P. 163–170.

10. On the Leading Edge Cavitation In a Helicocentifugal Pump: Experimental and Numerical Investigations / D. Pierrat, L. Gros, A. Couzinet, G. Pintrand // 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, October 14–16, 2009, Brno, Czech Republic.

11. Application of modified k-ω model to predicting cavitating flow in centrifugal pump / Hou-lin Liu*, Dong-xi Liu, Yong Wang, Xian-fang WU, Jian Wang // Water Science and Engineering. 2013. No. 6(3). P. 331–339.

12. Rafael Campos-Amezcua, Sofiane Khelladi, Zdzislaw Mazur-Czerwiec, Farid Bakir, Alfonso Campos-Amezcua and Robert Rey (2011). Numerical and Experimental Study of Mass Transfer Through Cavitation in Turbomachinery, Mass Transfer – Advanced Aspects, Dr. Hironori Nakajima (Ed.), InTech.

13. Cavitation modeling for steady-state CFD simulations / L. Hanimann, L. Mangani, E. Casartelli, M. Widmer // IOP Conf. Ser.: Earth Environ. 2016.

14. Unsteady Flow and Structural Behaviors of Centrifugal Pump under Cavitation Conditions. Chinese / Denghao Wu, Yun Ren, Jiegang Mou, Yunqing Gu, Lanfang Jiang // Journal of Mechanical Engineering. 2019.

15. Cavitation Optimization for a Centrifugal Pump Impeller by Using Orthogonal Design of Experiment / Pei Ji1, Yin Tingyun1, Yuan Shouqi1, Wang Wenjie1, Wang Jiabin // Chinese Journal of Mechanical Engineering. a2017. Vol. 30,aNo. 1. P. 103.

© Torgashin A. S., Zhujkov D. A., Nazarov V. P., Begishev A. M., Vlasenko A. V., 2020

Begishev Aleksey Mikhaylovich – graduate student; Reshetnev Siberian State University of Science and Technology. E-mail: alex-beg95@mail.ru.

Zhujkov Dmitrij Aleksandrovich – Cand. Sc., Docent of a department of Aircraft Engines; Reshetnev Siberian State University of Science and Technology. E-mail:d_zhuikov@sibsau.ru.

Na**zarov Vladimir Pavlovich** – Cand. Sc., Professor, Head of the Department of Aircraft Engines; Reshetnev Siberian State University of Science and Technology. E-mail:nazarov@sibsau.ru.

Torgashin Anatoliy Sergeevich – graduate student; Reshetnev Siberian State University of Science and Technology. E-mail: ttarg23@gmail.com.

Vlasenko Alesksey Vladimirovich – graduate student; Reshetnev Siberian State University of Science and Technology. E-mail: lesha.vlasenko.94@mail.ru.

Бегишев Алексей Михайлович – аспирант; Сибирский государственный университет науки и технологий имени академика М. Ф. Решетнева. E-mail: alex-beg95@mail.ru.

Жуйков Дмитрий Александрович – кандидат технических наук, доцент кафедры двигателей летательных аппаратов; Сибирский государственный университет науки и технологий имени академика М. Ф. Решетнева. E-mail: d_zhuikov@sibsau.ru.

Назаров Владимир Павлович – кандидат технических наук, профессор, заведующий кафедрой двигателей летательных аппаратов; Сибирский государственный университет науки и технологий имени академика М. Ф. Решетнева. E-mail: nazarov@sibsau.ru.

Торгашин Анатолий Сергеевич – аспирант; Сибирский государственный университет науки и технологий имени академика М. Ф. Решетнева. E-mail: ttarg23@gmail.com.

Власенко Алесксей Владимирович – аспирант; Сибирский государственный университет науки и технологий имени академика М. Ф. Решетнева. E-mail: lesha.vlasenko.94@mail.ru