AVDIEIEVA, L. Yu. (http://orcid.org/0000-0002-3434-1669),
MAKARENKO, A. A. (http://orcid.org/0000-0003-2338-5364),
and DEKUSHA, H. V. (http://orcid.org/0000-0002-8829-8221)
Institute of Engineering Thermophysics, the of National Academy of Sciences of Ukraine,
2a, Marii Kapnist St., Kyiv, 03057, Ukraine,
+380 44 456 6282, admin@ittf.kiev.ua

COMPUTER SIMULATION OF FLUID FLOW THROUGH A VENTURI NOZZLE OF DIFFERENT CONFIGURATIONS

Introduction. *Hydrodynamic cavitation, as an effective way of local energy concentration to create powerful dynamic effects, has been widely used to intensify many energy-intensive operations related to processing complex heterogeneous dispersed systems.*

Problem Statement. *The high cost of equipment for physical experiments and the difficulties related to reproducing complex hydrodynamic processes in laboratory conditions make it necessary to use modeling/simulation methods. Recently, mathematical and computer modeling has become one of the most effective information technologies that determine the rapid development of advanced fields of science and technology.*

Purpose. *The purpose of this research is to predict the behavior of fluid motion inside the Venturi nozzles of different configurations in the case of changing thermal process parameters, with the use of ANSYS Fluent computational package.*

Materials and Methods. *The Simple algorithm of the Patankar method, which involves a second-order accuracy counter flow scheme for convective terms in the momentum conservation equation, for the kinetic turbulent energy equation and the turbulent energy dissipation equation, has been used. The "Realizable" modified k-ε model of turbulence and Euler's model (multiphase model) have been applied. The standard ANSYS ICEM CFD package has been employed to generate the calculation grid.*

Results. *The model chosen for computer simulation has proven its effectiveness and allowed us to establish some patterns of fluid motion along the axis of the Venturi nozzle. Based on the simulation results, the dependences of changes in the pressure in the case of varying neck diameter and opening angle of the Venturi nozzle diffuser have been constructed. It has been shown that the highest intensity of cavitation is reported in the experimental nozzle with opening angle of the diffuser αdif = 12° for all diameters of the nozzle neck.*

Conclusions. *The use of highly specialized software and modeling systems allows us to better understand the behavior of the flow in closed channels of different configurations. Computer simulation of fluid motion in Venturi nozzles has enabled predicting the processes of occurrence and development of hydrodynamic cavitation in different sections of the nozzle.*

Keywords: computer simulation, ANSYS Fluent, hydrodynamic cavitation, and Venturi nozzle.
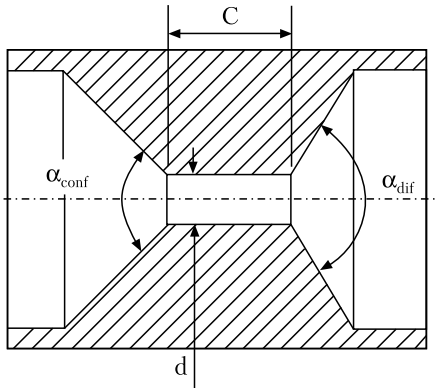
Fig. 1. Venturi nozzle configuration

Hydrodynamic cavitation, as an effective method of local energy concentration to create powerful dynamic effects, has been widely used for the intensification of many energy-intensive operations of processing complex heterogeneous dispersed systems [1, 2].

The expensive equipment for physical experiments and the difficulties related to reproducing complex hydrodynamic processes in laboratory conditions have led to the need to use modeling/ simulation methods. Recently, mathematical and computer modeling has become one of the most effective information technologies that determine the rapid development of advanced fields of science and technology, as evidenced by the appearance of application program packages for numerical modeling of gas-dynamic and heat-mass exchange processes, which have been widely used in scholarly research and engineering practice. Among them, there are the products of *ANSYS, Inc.*, including those in the field of computational fluid dynamics (CFD) [3]. This is because of the formal simplicity of the problem statement and the independence of the solution method on the operation of the node under study. Among the software products that allow us to calculate, to visualize, and to optimize a wide range of technological processes *ANSYS Fluent* computing package is the first to be mentioned.

ANSYS Fluent is a universal multi-purpose computational CFD software designed to solve problems of hydrodynamics and heat and mass transfer. *ANSYS Fluent* makes it possible to model processes related to the flow of gases, liquids, and their mixes in complex physical and chemical interactions [3—5].

ANSYS Fluent uses the finite volume method to convert the general equation of motion into the algebraic form for further numerical solution. The software complex is widely used for:

◆ numerical modeling of hydraulic and gas-dynamic processes in engines, pumps, and other power engineering equipment [6—12];
◆ simulation of mass transfer during cavitation, turbulent or convection motion of liquid [6, 10, 13—19];
◆ simulation of liquid or gas motion inside nozzles [10, 16—22];

Hydrodynamic cavitation devices of various types are widely used to intensify the processes of dissolution, mixing/blending, dispersion, emulsification, and homogenization in dispersed systems of the liquid-liquid or liquid-solid type [2]. Venturi nozzle (Fig. 1) [1, 2, 17] is the basic element of many known flow hydrodynamic cavitation reactors of the static type.

The Venturi nozzle is a device that consists of aninlet that is the convergent section (confuser), a middle cylindrical section (nozzle neck), and an outlet that is the divergent section (diffuser). The configuration of the entire cavitation apparatus, as well as the profile and geometric parameters of the reactor significantly influence the intensity of the occurrence and development of hydrodynamic cavitation effects and, as a result, the quality of dispersion of complex multiphase systems [2]. The design of the device should provide optimal conditions for the occurrence of cavitation, criteria for the efficiency of energy use and reduction of its losses in the course of useful work. The complexity and multifactorial statement of the problems of experimental studies of the behavior of a two-phase gas-liquid flow in a Venturi nozzle, depending on changes in its geometric parameters, makes computer modeling an urgent research and practical task.

The authors have made a computer simulation of the fluid motion in Venturi nozzle, using the *ANSYS Fluent* computational CFD package and the Eulerian model (multiphase model). This has made it possible to simulate the occurrence of the phenomenon of cavitation in the nozzle for the two-phase flow of liquid-gas (water-steam) system. For modeling the cavitation flow, the following fundamental laws of fluid and gas mechanics have been used: 1) the continuity equation; 2) the momentum equation; and 3) the law of change in turbulent viscosity as a function of speed (turbulence model). To solve the problem, the simple algorithm of the Patankar method has been applied, with the involvement of a counterflow scheme of the second order of accuracy for the convective terms in the momentum conservation equation, for the turbulent kinetic energy equation and the turbulent energy dissipation equation.

The choice of the turbulence model is the most difficultfor calculating cavitation, therefore, for the calculation of cavitation in local hydraulic resistances, the k-ε model based on the Boussinesq hypothesis is the most appropriate. In this research, we have used the modified *Realizable* k-ε model of turbulence. It differs from the standard k-ε model by an improved form of the presentation of turbulent viscosity and a new equation for the transport of the turbulence kinetic energy dissipation rate ε. The immediate advantage of the *Realizable* k-ε model is that it more accurately predicts the distribution of flux dissipation. This provides better prediction of rotating flows, boundary layers that are subject to strong pressure gradients, separation and recirculation flows [3].

The turbulent flow ofliquid-gas system has been considered in the research. Pressure, initial turbulence parameters, and volume fractions of phases are set at the flow inlet; pressure and mild turbulence boundary conditions are given at the flow outlet. The standard boundary conditions for turbulent flow,as built into the *Fluent* software complex, are set by assigning wall functions on the lines that correspond to solid surfaces. The symmetry condition on the axis of the nozzle is given.
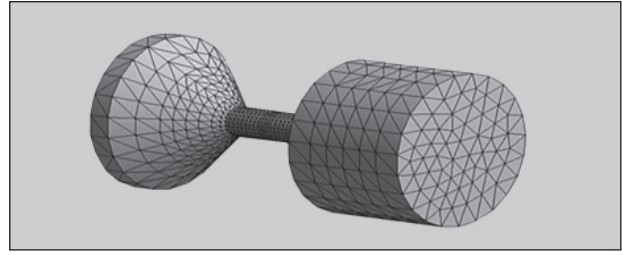

Fig. 2. Geometric 3D model of Venturi nozzle with calculation grid
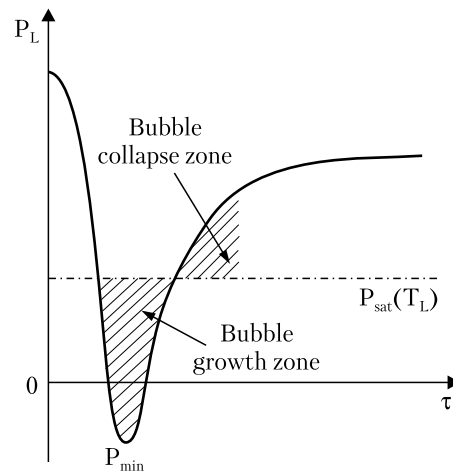

Fig. 3. Dependence of characteristic change in pressure on duration for the formation of cavitation effects in liquid

Fig. 2 shows a geometric 3D model of Venturi nozzle. The standard ANSYS ICEM CFD package is used to generate the calculation grid. This package allows us to generate structured and unstructured block grids quite accurately. The method of computational hydrogas dynamics and the *Fluent* solver have been employed to build the grid.

The use of this software complex has made it possible to divide the domain of solving the problem into a dense system of subdomains (Fig. 2). In each subdomain, the original distribution of parameters is replaced by the corresponding approximating functions. The quality of the grid model affects the accuracy, convergence, and speed of solving the problem, so the spatial extension of the grid is taken to be 1 mm. Also, at this stage, there are giventhe axis of symmetry, the in-
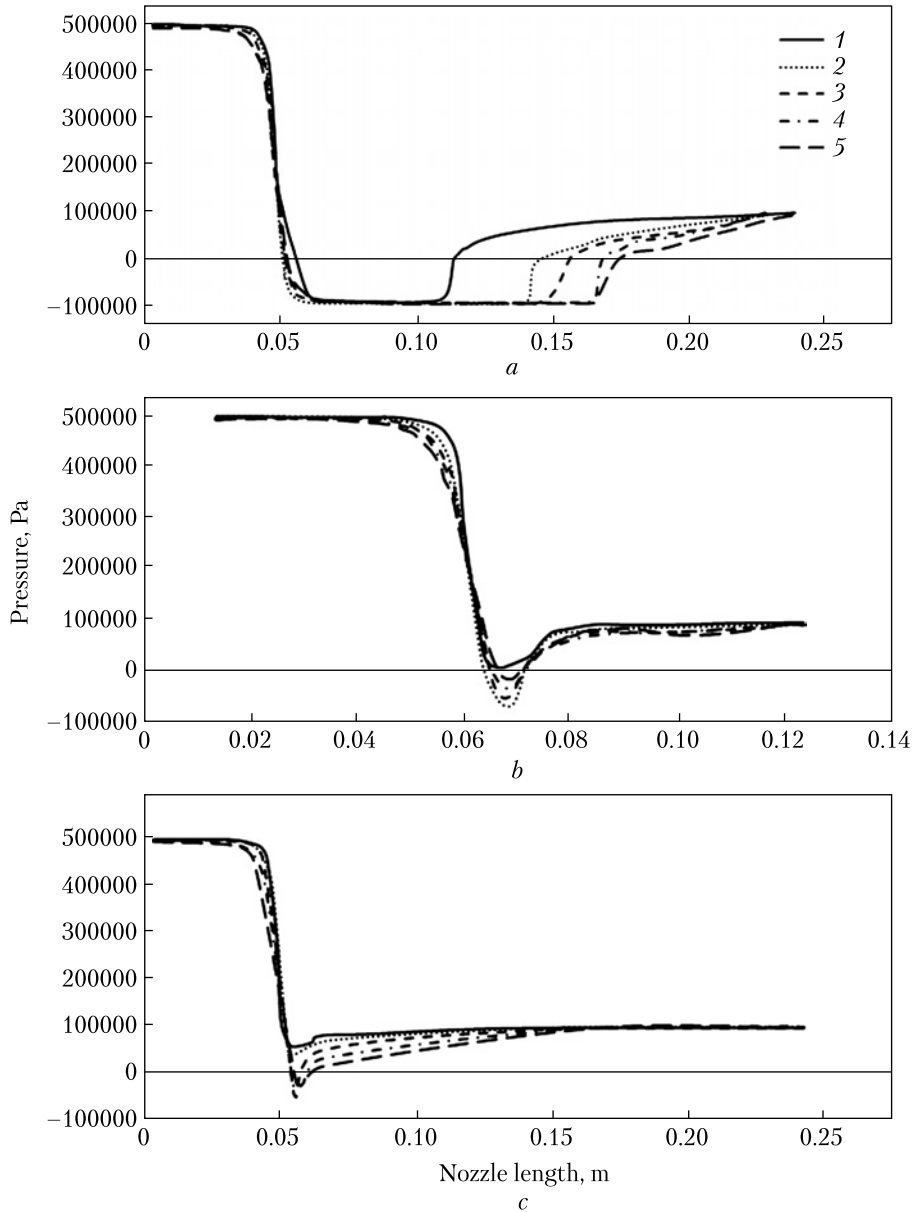
Fig. 4. Dependence of change in pressure along the axis of Venturi nozzle at different inner diameters of the neck: *1* — 8 mm; *2* — 10 mm; *3* — 12 mm; *4* — 14 mm; *5* — 16 mm and opening angle of diffuser: $a - \alpha_{\text{dif1}} = 12°$; $b - \alpha_{\text{dif2}} = 90°$; $c - \alpha_{\text{dif3}} = 120°$

let and outlet surfaces of the nozzle, as well as the walls of the model. The model makes it possible to establish regularities in the distribution of indicator values along the entire length of the test nozzle in order to determine the characteristic zones along the axis and near the walls of the re-

actor. Numerical solution of the gas-liquid flow motion dependence enables obtaining the following flow functions throughout the field volume: axial and radial velocities for each of the phases, volumetric fractions of the phases, pressure, specific kinetic energy of turbulence, its dissipation,
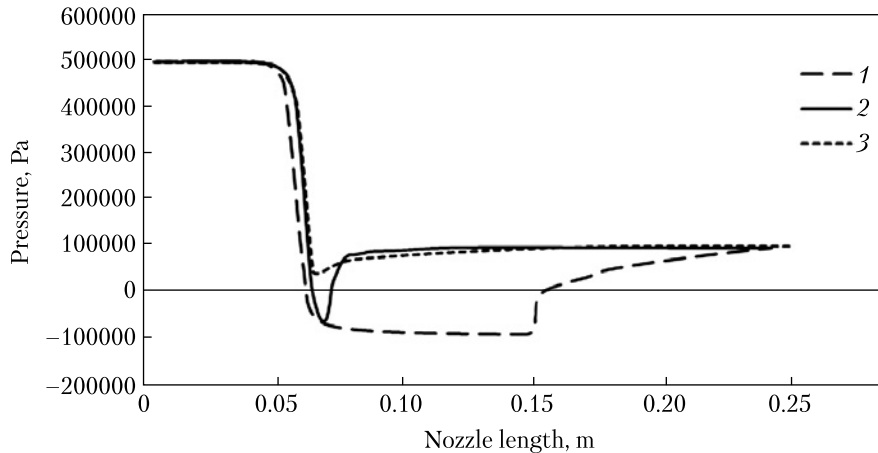
Fig. 5. Dependence of pressure change along the axis of Venturi nozzle at different opening angles of the diffuser with the same inner diameter of the nozzle neck d = 10 mm: *1* — α_{dif1} =12°; *2* — α_{dif2} = 90°; *3* — α_{dif3} = 120°

as well as other parameters that are expressed through these quantities.

It has been known that to initiate cavitation phenomena, it is necessary to decrease the pressure in liquid in a short time to values much lower than the pressure of saturated vapor of the liquid at a given temperature, and then quickly increase to values that exceed the pressure of saturated vapor (Fig. 3) [1].

Until the conditions for the occurrence of cavitation are met, in the liquid volume, there are formed and growing many vapor bubbles that are defined as a cavitation cluster. The duration of the existence of a cavitation cluster is estimated as microseconds. The duration of both stages of the cavitation process, the decrease and the subsequent sharp increase in the liquid pressure, should be as short as possible [1, 2].

On the basis of previous mathematical modeling, to determine the optimal design of Venturi nozzle, two determining geometric parameters, the angle of the diffuser and the diameter of the nozzle neck, which strongly influence the pressure and flow rate, have been established. It is accepted that for technological reasons of manufacture, the opening angle of the diffuser α_{dif} in the Venturi nozzle should not be less than 12°. Therefore, the simulation is made for a nozzle with diffuser

opening angles $\alpha_{dif} \geq 12°$. Other geometric parameters of the study areconstant confusor opening angle that is equal to $\alpha_{conf} = 90°$; the diffuser opening angles vary and are equal to $\alpha_{dif1} = 12°$, $\alpha_{dif2} = 90°$, $\alpha_{dif3} = 120°$; the inlet and the inner diameters of the main duct are $D_{in} = D_{out} = 42$ mm; the inner diameters of the nozzle neck are equal to $d_1 = 8$ mm; $d_2 = 10$ mm; $d_3 = 12$ mm; $d_4 = 14$ mm; and $d_5 = 16$ mm. The nozzle neck length is $l = 10$ mm. The pressure at the nozzle inlet is $P_0 = 5 \cdot 10^5$ Pa, the pressure at the outlet is $P_1 = 1 \cdot 10^5$ Pa, the pressure of saturated vapor at aa temperature 20 °C is $P_k = 2330$ Pa.

To determine the critical geometric parameters, we have focused our analysis on changes in pressure in Venturi nozzle at different opening angles of the diffuser. The pressure indicator is important for characterizing the occurrence and intensity of hydrodynamic cavitation, liquid consumption, and specific electricity consumption. The simulation results obtained by us (Fig. 4) illustrate the pressure distribution along the axis of the nozzle at different diameters of the neck and opening angle of the diffuser.

For all the studied configurations of the nozzles (Fig. 4), the change in the pressure in different areas has the following character: a sharp decrease in the initial value (P_0) in the confuser due

to a decrease in the cross-sectional area and an increase in the flow speed (Bernoulli's principle), constant along the length of the neck gradual increase up to a value (P_1) smaller than the initial one in the diffuser.

Turbulence energy and its dissipation reach their maximum at the nozzle neck. In this section of the nozzle, the greatest transformation of the kinetic energy of the flow into the energy of deformation and grinding of dispersed inclusions may occur. The change in the inner diameter of the nozzle neck at the same opening angle of the diffuser practically does not influence the pressure difference, but it has a significant effect on the change in the speed and flow rate of the liquid, which in turn, influences the specific consumption of electricity. Choosing the optimal geometric size of the diameter of the nozzle neck requires additional estimates of these characteristics.

The results have shown that the opening angle of the diffuser α_{dif} has the greatest influence on change in the pressure along the axis of the Venturi nozzle. At different opening angles of the diffuser, there have been established significant differences in the nature of the flow dependence of the pressure change along the Venturi nozzle axis. The most significant pressure difference has been reported at the opening angle of the diffuser $\alpha_{dif} = 12°$, which indicates the greatest intensity of hydrodynamic cavitation in this experimental nozzle

(Fig. 5). An increase in the diameter of the nozzle neck leads to an increase in the zone of bubble growth and collapse and its gradual transition into the depth of the high pressure zone of the diffuser.

With an increase in the outlet angle to $\alpha_{dif2} = 90°$ and $\alpha_{dif3} = 120°$, at all diameters of the nozzle neck, the pressure difference decreases due to the appearance of side wall turbulent vortices. This, in its turn, leads to an increase in energy losses. The given results have been confirmed by the studies of other authors [17].

Thus, the use of highly specialized software modeling systems allows a better understanding of the flow behavior in closed channels of various shapes. The computer modeling of fluid motion in Venturi nozzles has made it possible to predict the occurrence and development of hydrodynamic cavitation at different sections. Nozzles with different geometric parameters of the nozzle neck and diffuser opening angle have been studied. According to the results of the flow motionsimulation along the Venturi nozzle axis, the corresponding dependences of the pressure change along the nozzle length have been built. The results have shown that the opening angle of the diffuser α_{dif} has the greatest influence on change in the pressure along the Venturi nozzle axis and that the greatest intensity of cavitation effect is achieved in the experimental nozzle with the opening angle of the diffuser $\alpha_{dif} = 12°$ for all diameters of the nozzle neck.

REFERENCES

1. Ivanitsky, G. K., Avdieieva, L. Yu., Makarenko, A. A. (2016). Using the effects of hydrodynamic cavitation for purposeful dynamical action on the supramolecular structures. *Physics of aerodisperse systems*, 53, 142—151.
2. Dolinsky, A., Avdieieva, L., Makarenko, A. (2020). *Cavitation technologies for the production of nanopreparations*. Kyiv: Naukova dumka [in Ukrainian].
3. *ANSYS Fluent. User's Guide. Release 19.2.* (2018). Ansys Inc.
4. Ledoux, M., ELHami, A. (2021). Modeling and Simulations under ANSYS. *Heat Transfer 2: Radiative Transfer*, 10, 169—215. https://doi.org/10.1002/9781119821304.ch6
5. Zore, K., Ozcer, I., Munholand, L., Stokes, J. (2020). ANSYS CFD Simulations of Supersonic and Hypersonic Flows. *NSSW2020: 6th National Symposium on Shock Waves. IITM (26—28 February, 2020, Chennai)*. India.
6. Sulinov, A., Shabliy, L., Zubanov, V. (2015). Methods of modeling the work process of hydrogen screw-centrifugal pumps using ANSYS CFD. *Vestnik of the Samara State Aerospace University*, 14(3—2), 305—315 [in Russian]. https://doi.org/10.18287/2412-7329-2015-14-3-305-315
7. Spille, A., Hesse, J. (2016). CFD simulation of cavitation in an internal gear pump. *3rd International Rotating Equipment Conference. Pump Users International Forum (14—15 September 2016, Düsseldorf)*. Germany.

8. Sandyal, P. K. S. L. (2016). CFD Simulation of Centrifugal Blower Using ANSYS CFX. *Materials of National Conference on Advances in Mechanical Engineering Science* (*29 April, 2016, Kattankulathur*). India. 34—38.

9. Cabrera Escobar, J., Jurado, F., Vera, D. (2021). Simulation of an active indirect hybrid dehydrator using ANSYS software. *Enfoque UTE,* 12(4), 29—44. https://doi.org/10.29019/enfoqueute.771

10. Hami, E. L. A., Radi, B. (2017). *Fluid—Structure Interaction with Ansys Fluent*. Wiley-ISTE.

11. Caunii, V., Sachelarie, A. (2016). Simulation of the Air Conditioning Curtains with Turbulent Circular Jet Flows Inside the Cabin Vehicle Using ANSYS CFD. *Proceedings of the European Automotive Congress EAEC-ESFA* (*25—27 November 2015, Bucharest, Romania*). Springer. Cham. P. 357—366. https://doi.org/10.1007/978-3-319-27276-4_33.

12. Vyroubal, P., Mačák, M. (2019). Investigation of Cyclic Voltammetry in Ansys Fluent. *ECS Transactions.*, 95(1), 467—474. https://doi.org/10.1149/09501.0467ecst.

13. Kinzel, M., Lindau, J., Kunz, R. (2019). An Assessment of CFD Cavitation Models Using Bubble Growth Theory and Bubble Transport Modeling. *Journal of Fluids Engineering*, 141(4), 041301. https://doi.org/10.1115/1.4042421.

14. Chen, T., Huang, B., Wang, G., Zhao, X. (2016). Numerical study of cavitating flows in a wide range of water temperatures with special emphasis on two typical cavitation dynamics. *International Journal of Heat and Mass Transfer*, 101, 886—900. https://doi.org/10.1016/j.ijheatmasstransfer.2016.05.107.

15. Niedzwiedzka, A., Sobieski, W. (2016). Verification of cavitation models in ANSYS Fluent. *Materials of 1st Workshop on Porous Media* (*1—3 July, 2016, Olsztyn, Poland*). 38.

16. Uyanna, O., Najafi, H. A. (2017). A Numerical Analysis on Cavitation Effects in Microchannels. *Proceedings of the AS-ME-2017. International Mechanical Engineering Congress and Exposition. Volume 7: Fluids Engineering* (*November 3—9, 2017, Tampa, Florida, USA*). https://doi.org/10.1115/IMECE2017-71909.

17. Li, M., Bussonnière, A., Bronson, M., Xu, Zh., Liu, Q. (2019). Study of Venturi tube geometry on the hydrodynamic cavitation for the generation of microbubbles. *Minerals Engineering*, 132, 268—274. https://doi.org/10.1016/j.mineng.2018.11.001

18. Hilares, R. T., Dionízio, R. M., Sanchez, S., Prado, C. A., R. de Sousa Júnior, S. S. da Silva, Santos, J. C. (2020). Hydrodynamic cavitation-assisted continuous pre-treatment of sugarcane bagasse for ethanol production: effects of geometric parameters of the cavitation device. *Ultrasonics Sonochemistry*, 63, 104931. https://doi.org/10.1016/j.ultsonch.2019. 104931

19. Liao, Y., Lucas, D. (2018). Evaluation of Interfacial Heat Transfer Models for Flashing Flow with Two-Fluid CFD. *Fluids*, 3(2), 38. https://doi.org/10.3390/fluids3020038

20. Tiwari, G., Kumar, J., Prasad, V., Patel, V. (2020). Utility of CFD in the design and performance analysis of hydraulic turbines — A review. *Energy Reports*, 6, 2410—2429. https://doi.org/10.1016/j.egyr.2020.09.004

21. Xue, W., Huai, W., Qian, Zh., Yang, Zh.-H., Zeng, Yu. (2014). Numerical simulation of initial mixing of marine wastewater discharge from multiport diffusers. *Engineering Computations*, 31(7), 1379—1400. https://doi.org/10.1108/EC-06-2013-0148

22. Ionel Olaru. (2013). The fluid flow simulation through to a Venturi nozzle. *Journal of Engineering Studies and Research*, 19(1), 42—46.

Л.Ю. Авдєєва (http://orcid.org/0000-0002-3434-1669),
А.А. Макаренко (http://orcid.org/0000-0003-2338-5364),
Г.В. Декуша (http://orcid.org/0000-0002-8829-8221)

Інститут технічної теплофізики Національної академії наук України,
вул. Марії Капніст, 2а, Київ, 03057, Україна,
+380 44 456 6282, admin@ittf.kiev.ua

КОМП'ЮТЕРНЕ МОДЕЛЮВАННЯ РУХУ РІДИНИ В СОПЛАХ ВЕНТУРІ РІЗНИХ КОНФІГУРАЦІЙ

Вступ. Гідродинамічна кавітація як ефективний спосіб локальної концентрації енергії для створення потужних динамічних ефектів широко застосовується для інтенсифікації багатьох енергоємних процесів обробки складних гетерогенних дисперсних систем.

Проблематика. Висока вартість обладнання для фізичного експерименту й труднощі відтворення в лабораторних умовах складних гідродинамічних процесів спричиняють необхідність використання методів їхнього моделювання. Останнім часом математичне та комп'ютерне моделювання перетворилося в одну з найбільш ефективних технік.

Мета. Прогнозування поведінки руху рідини всередині сопел Вентурі різних конфігурацій при зміні теплотехнологічних параметрів за допомогою обчислювального пакету *ANSYS Fluent*.

Матеріали і методи. Використано алгоритм *Simple* методу Патанкар із залученням протипотокової схеми другого порядку точності для конвективних членів в рівнянні збереження імпульсу, для рівняння кінетичної турбулентної енергії і рівняння дисипації турбулентної енергії; застосовано модифіковану k-ε модель турбулентності «*Realizable*» та Ейлерову модель *Mixture* (модель багатофазної суміші). Для генерації розрахункової сітки використано стандартний пакет *Ansys ICEM CFD*.

Результати. Обрана модель для комп'ютерного прогнозування показала свою ефективність і дозволила встановити деякі закономірності руху рідини по осі сопла Вентурі. За результатами моделювання побудовано залежності зміни показників тиску при зміні діаметра горловини й кута розкриття дифузору сопла Вентурі. Показано, що найбільша інтенсивність кавітаційного впливу досягається в дослідному соплі з кутом розкриття дифузора α_{dif} = 12° для всіх діаметрів горловини сопла.

Висновки. Застосування вузькоспеціалізованих програмно-моделюючих систем дозволяють краще зрозуміти поведінку течії в закритих каналах різних профілів. Комп'ютерне моделювання руху рідини в соплах Вентурі дозволило забезпечити прогнозування процесів виникнення і розвитку гідродинамічної кавітації на різних відрізках.

Ключові слова: комп'ютерне моделювання, *ANSYS Fluent*, гідродинамічна кавітація, сопло Вентурі.