ANIMAL SCIENCE AND FOOD TECHNOLOGY

Journal homepage: https://animalscience.com.ua/en
Animal Science and Food Technology, 15(4), 80-101
Received 31.07.2024 Revised 30.10.2024 Accepted 27.11.2024

UDC 66.069.83
DOI: 10.31548/animal.4.2024.80

Research and application of nozzles with inclined inlet channels for jet sulphitators in the sugar industry

Vitalii Ponomarenko*
PhD in Technical Sciences, Associate Professor
National University of Food Technologies
01601, 68 Volodymyrska Str., Kyiv, Ukraine
https://orcid.org/0000-0002-0732-4783

Dmytro Liulka
PhD in Technical Sciences, Associate Professor
National University of Food Technologies
01601, 68 Volodymyrska Str., Kyiv, Ukraine
https://orcid.org/0000-0001-8014-5776

Volodymyr Vasyliv
PhD in Technical Sciences, Associate Professor
National University of Life and Environmental Sciences of Ukraine
03041, 15 Heroiv Oborony Str., Kyiv, Ukraine
https://orcid.org/0000-0002-2109-0522

Roman Yakobchuk
PhD in Technical Sciences, Associate Professor
National University of Food Technologies
01601, 68 Volodymyrska Str., Kyiv, Ukraine
https://orcid.org/0000-0001-9777-5790

Yevhen Babko
PhD in Technical Sciences, Associate Professor
National University of Food Technologies
01601, 68 Volodymyrska Str., Kyiv, Ukraine
https://orcid.org/0000-0003-2389-007X

Yuliia Slyva
PhD in Technical Sciences, Associate Professor
National University of Life and Environmental Sciences of Ukraine
03041, 15 Heroiv Oborony Str., Kyiv, Ukraine
https://orcid.org/0000-0003-2592-6822

Suggested Citation:
Ponomarenko, V., Liulka, D., Vasyliv, V., Yakobchuk, R., Babko, Ye., & Slyva, Yu. (2024). Research and application of nozzles with inclined inlet channels for jet sulphitators in the sugar industry. *Animal Science and Food Technology*, 15(4), 80-101. doi: 10.31548/animal.4.2024.80.

**Corresponding author*

Abstract. The paper was devoted to the study of the process of liquid spraying by nozzles used in jet sulphitators in the sugar industry or as separate equipment. Liquid sprayers play a major role in ensuring efficient operation of heat and mass transfer equipment due to the creation of a significant contact surface. When designing or auditing its technical integrity, it is necessary to have reliable hydrodynamic characteristics of sprayers. Research in this area is relevant. The purpose of the study was to establish all the characteristics of liquid flow in the elements of the nozzle and spray swath using comprehensive research (experimental and computer) for the possibility of intensifying the operation of equipment, in particular, sulphitators for the sugar industry. To achieve this goal, experimental (studies of injectors and ejectors on a hydraulic stand) and CFD studies of these objects in the modules of the ANSYS software suite (version: ANSYS 2020 R2) were conducted. The dependence of the nozzle flow rate on the liquid pressure and the number of supply channels for different types of injectors (jet, centrifugal-jet) has been experimentally established. It has been experimentally confirmed that the uniformity of liquid distribution in the nozzle swath with two supply channels was high and sufficient for practical applications. In the CFD study of the characteristics of the nozzle spray swath, it was proposed to reduce the design area to the cut cone, which reduced the calculation time. When modelling the hydrodynamics of the liquid in the nozzle, numerically characteristic velocities were determined both in the nozzle elements and in the outlet. It was established that there was a vacuum in the root zone reaching 865 Pa for this nozzle at a liquid inlet pressure of 0.25 MPa, which led to gas ejection into the swath. The above calculation methodology was relevant for determining the characteristics of injectors, in the development and design of new sprayers with a given capacity and a spray swath, used both as active ejector nozzles and as independent equipment for dispersing process liquids in the food industry

Keywords: sugar production; liquid sulphitation; spray nozzle; flow rate; CFD modelling; hydrodynamics; spray swath

Introduction

Liquid spraying is an extremely widespread process in almost all areas of human activity. It has found use starting from growing plants (irrigation), in emergency situations (extinguishing fires), during technological processes in the chemical, food, and aerospace industries. The proposed nozzles with inclined supply channels were developed for the needs of the sugar industry, namely for sulphitation of water and sugar solutions in jet sulphitation tanks, deammonisers of water vapour condensates. These processes are necessary in the production of crystalline sugar, ensuring the production of a high-quality product. However, for the efficient operation of the equipment, injectors with high both consumption, dispersion and operational characteristics must be used. No such studies were conducted for the needs of the sugar industry. It should also be added that knowledge about the characteristics of sprayers is required not only for sugar, but also for any industry where heat and mass transfer processes are carried out according to the technological process.

The injectors had to have high reliability, since the liquids were not previously cleaned of dirt, which led to the loss of their performance. A special feature of the nozzles is that the potential energy of the compressed liquid is converted into kinetic energy, followed by the formation of liquid droplets. Such transformations are accompanied by a significant increase in the contact surface of the phases and an intensification of mass transfer. When carrying out technological processes of heat and mass transfer in food, chemical, and other industries, increasing the contact surface of phases is a powerful means of increasing the amount of substance transferred. Usually, the mass transfer coefficients also increase simultaneously due to the formation of a new interface, its rapid renewal, and a decrease in the size of the boundary layer for the interfacial transition of reagents. In two-phase liquid-gas media, an effective way to increase the surface is to spray the liquid using injectors (sprayers) of various designs. The contact surface (droplet diameter) is different for different types of nozzles and depends on the physical properties of the liquid. Thus, the paper by E. Stallbaumer-Cyr *et al*. (2024) presented experimental data on determining the flow rate, volume diameter, etc. The findings were illustrated with images to confirm the results. However, these data are true only for this type of nozzle under study, and when the dimensions change, or the ratios between dimensions change, the characteristics change, which requires new research.

R. Sijs *et al*. (2023) noted that sprayers are of great importance for many applications, with the drop size being a crucial parameter. It was shown that the average drop size is determined by the area of the nozzle opening, as in the case of conventional flat fan nozzles, but it is necessary to consider the additional pressure drop in the Venturi chamber. This parameter allows reusing the ratios that were obtained for flat fan nozzles.

To reduce air dust, H. Han *et al*. (2023) compared the operation of injectors (and established the optimal phase supply pressures to obtain droplets of minimum size. According to E. Verbiawan *et al*. (2023), to accelerate the evaporation of seawater in salt production along with conventional methods, it is recommended to use liquid spray technology. Jet nozzles are also known to be used to intensify hydrodynamic mixing in tanks. Research by T. Oluwadero *et al*. (2023) aimed at establishing the main characteristics of the spray swath, but the general disadvantage of these studies is that if the conditions for conducting experiments change, it is necessary to repeat them. This is a common disadvantage of conducting a physical experiment.

For the sugar industry, the transition of sulphitation of liquids from bubbling-type apparatuses to jet apparatuses took place in the 1970s, which was progressive at that time. However, the design of the devices took place without a proper scientific and practical basis, which provoked quite serious shortcomings in their operation. One of the main elements that has a significant impact on the operation of the jet sulphitation unit is the design of the sprayer used to disperse the liquid. It has been practically proven that for effective operation of sulphitators, it is necessary to have a centrifugal-jet sprayer. Such sprayers were subsequently proposed, but required a comprehensive study to be able to use them as an active ejector nozzle.

The use of liquid dispersion in combination with the high intensity of processes allows creating extremely efficient equipment that combines a number of advantages: simplicity, compactness, low specific energy consumption. Therefore, the relevance of the subject matter is beyond doubt. The study by S. Kim *et al.* (2023) experimentally showed that the nozzle design has a decisive influence on the formation of the spray swath, its filling with sprayed liquid droplets, the spray angle, and the diameter of liquid droplets. Therefore, the experimental part of the study on swath development is an important stage in clarifying the theory of jet decay and understanding the mechanism of swath development. Along with experimental research and software development, simulation modelling of the spray process plays a significant contribution to understanding the mechanism of swath development. However, it is possible to obtain data that is quite difficult to obtain experimentally. The accuracy of data acquisition depends on the correct configuration of all software modules, the accuracy of performing a three-dimensional model of the object under study, etc.

The purpose of this study was a comprehensive investigation of the process of liquid spraying with nozzles with inclined supply channels. Identification of the influence of the design of injectors on the spray swath, in particular, on the uniformity of liquid distribution over the cross-section of the swath, the angle of the swath. This will improve the equipment for the liquid sulphitation process in the sugar industry: increase the use of SO_2, improve the processing quality of sugar solutions. To achieve this goal, the following tasks were set: to create laboratory installations for conducting research and methods for conducting them; to produce research objects with accuracy that meet the requirements of practical operation; to conduct physical modelling of the spraying process; to conduct simulation modelling of the liquid flow in the elements of the nozzle and its spray swath; to validate the results.

From the scientific originality of the obtained results of a comprehensive study of fluid hydrodynamics in the nozzle elements with inclined supply channels and the spray process, the following should be noted: the dependence of the nozzle flow rate on geometric parameters and the number of supply channels at a liquid supply pressure of up to 0.4 MPa was obtained; it was proposed to perform the design zone of the spray swath in the form of a cut cone, which will reduce the time of machine calculation by reducing the number of calculation cells.

Literature Review

Studies of the liquid spray process were mainly performed for low-performance injectors used in fuel spraying. The results of research in this area can be found in the papers by R. Wu *et al*. (2024), A. Shulhin *et al*. (2022). The main requirement for fuel injectors was to obtain small droplets, and such liquids were pre-prepared by filtration. For them, the theory of fluid flow processes in the twisting chamber, the nozzle, which is a cylindrical channel, was developed, and the theory of decay of a cylindrical jet into liquid droplets was also proposed (Xu *et al*., 2022).

Much less studies have considered the process of spraying significant volumes of liquids not cleaned of contaminants or crystalline formations in it with centrifugal or centrifugal-jet nozzles. When installing them inside a hollow container, one of the key requirements is to evenly fill the cross-section with liquid droplets. This is provided only by centrifugal jet nozzles, which have high consumption characteristics, a filled spray swath, and a fairly fine fractional composition of droplets. Such recommendations can be found in the paper by S. Park & K. Park (2022). To improve the dispersed composition of drops, the nozzle designs are complicated, and additional structural elements are installed that ensure the formation of axial and tangential liquid jets. Such injectors were investigated by H. Han *et al*. (2020) and it was shown that the hydraulic supports of injectors increase at the same time. That is, reducing the size of droplets occurs due to a decrease in the reliability of the nozzle and an increase in energy consumption for spraying.

A variety of purposes also require a variety of sprayer designs. The researchers focused only on hydraulic type injectors, namely centrifugal-jet type injectors, which have proven themselves well in many branches of the economy due to their advantages, such as simplicity of design, although as the tasks that are assigned to sprayers become more complex, according to T. Lee *et al*. (2021), reliability decreases; the possibility of creating a significant phase contact surface, which is important when conducting heat and mass transfer processes in intensive mode. When the liquid is dispersed by nozzles (centrifugal, centrifugal-jet), there is a significant increase in the contact surface of the phases, which leads to a proportional increase in the amount of substance transferred. Droplet size measurements were performed for nozzles of different designs and it was shown that the contact surface increases by an order of magnitude, as found by X. Zhu *et al.* (2021); relatively low energy losses. Thus, the nozzle flow rate (the ratio of the actual flow rate to the theoretical one) in jet injectors reaches 0.95, in centrifugal-jet injectors – 0.85 (Vambol *et al.*, 2020).

Solving industrial problems that arise during technological processes leads to the creation of innovative injectors that ensure maximum efficiency of the process flow. Some of these injectors are injectors with an inclined arrangement of the supply channels relative to its axis (Mudavath & Arumuru, 2023). They were developed for spraying crude process fluids during sulphitation processes in sugar production. For their design, when scaling, the question arose of providing a more complete understanding of the hydrodynamics of the liquid in the elements of the nozzles, the processes of forming the spray swath and obtaining the dependences of the main characteristics of the nozzles, which will allow them to be designed with the specified properties. The characteristics of the injectors can be obtained experimentally, which requires the availability of experimental samples, stands for research, and requires time to conduct and process the obtained data. However, when switching from prototypes to industrial designs, compliance with the results is not always guaranteed, which requires additional experiments.

Modern computer technology and related programmes allow us solving these problems and significantly reducing the time for conducting such studies and reducing the cost of such studies while obtaining reliable results. Computer modelling allows both visualising and obtaining reliable numerical values of various values in any control sections of the studied research objects (nozzles) and in the spray swath, and based on the results obtained, create energy-efficient equipment. It is especially important to carry out such (CFD – computational fluid dynamics) modelling when designing injectors with specified characteristics of the spray swath, or when studying new sprayer designs. Changes in dimensions, their ratios, and design features require experimental confirmation, since these changes can cause significant deviations from the predicted characteristics.

In recent years, computer modelling of nozzle operation has become widespread. The first studies on numerical modelling related to establishing the characteristics of the spray swath of centrifugal nozzles. As the calculation model improved, the data obtained were refined, new tasks were solved, and new data was obtained. Centrifugal-jet injectors were considered, in which the decay of the liquid jet occurred when two flows interact: centrifugal and axial along the nozzle axis (von Helldorff & Micklow, 2019; Santosh *et al.*, 2023).

It should be noted that despite the progressive development of CFD research methods, accurate modelling of any processes, including modelling the liquid sputtering process, is an extremely difficult task. This is due to the fact that all processes in nature occur sequentially or in parallel with the main process under study and are complicated by secondary processes that affect one another. For the liquid spray process, J. Haidl *et al.* (2021) considered the temperature at which spraying occurs. The physical parameters of the liquid, cavitation phenomena, primary and secondary decay of droplets due to interaction with each other, deformation, variable aerodynamic drag, heat transfer between phases, and the establishment of an equilibrium state were considered by K. Kramm *et al.* (2023) and Y. Xu *et al.* (2024).

In CFD modelling of fluid dynamics in nozzle elements and the spraying process, important elements of the settings are the correct setting of boundary conditions, a minimal but sufficient choice of the study area with a breakdown into elements of the computational grid, etc. (Li *et al.*, 2024). Therefore, setting up the calculation modules of the programme to display adequate results is quite an important stage of research.

Materials and Methods

Designs of the injectors under study. For the experimental study, injectors with one (Fig. 1) and two (Fig. 2) inclined supply channels. One of the advantages of such injectors is the ability to adjust the flow rate of liquid in a wide range due to the simultaneous change in the area of the supply channels and the radius of twisting of the liquid in the nozzle body.

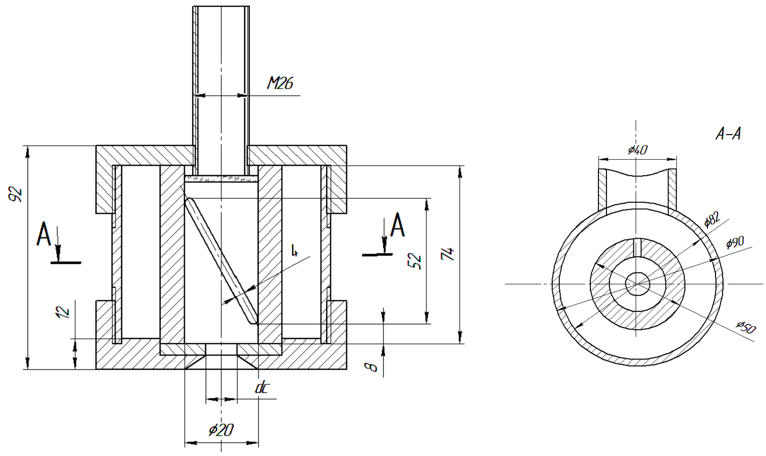

Figure 1. Nozzle with one inclined supply channel

Source: developed by the authors

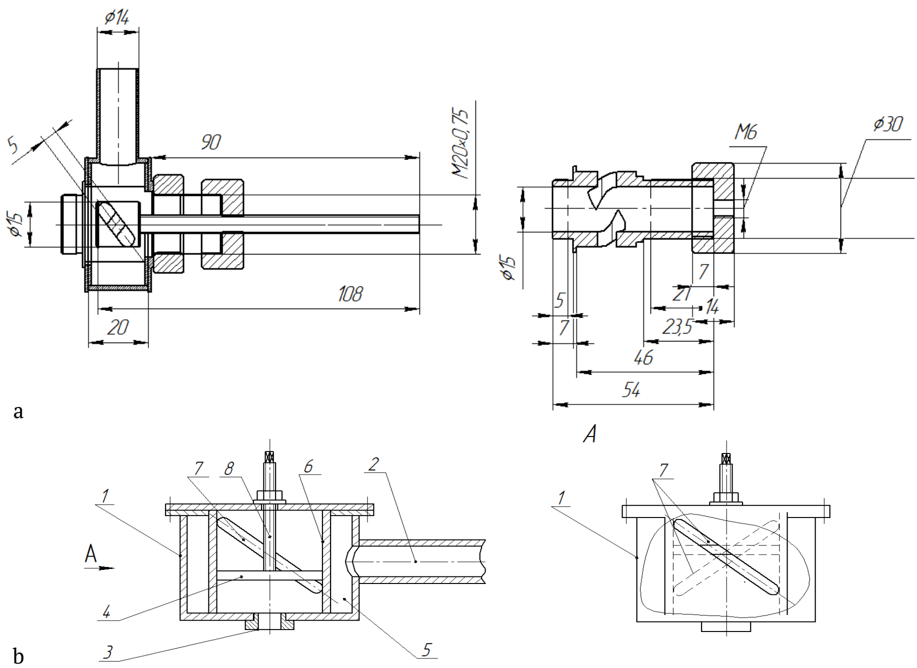

Figure 2. Nozzle with two inclined supply channels: a – structures, b – drawing

Source: developed by the authors

Nozzle (Fig. 2b) consists of a housing 1, to which a supply pipe 2 is connected, through which liquids are supplied inside the housing. At one end of the nozzle body there is an injector 3, and at the opposite end there is a plunger 4 with the possibility of axial movement by means of a screw 8 during its rotation. The supply pipe 2 is connected to the intermediate chamber 5 of the nozzle. The inner cylinder 6 of this chamber has slotted channels 7, which are made at an angle α to the longitudinal axis of the nozzle. There are two such channels on different sides of the axis of the inner cylinder 6.

The flow of liquid under pressure through the supply pipe 2 enters the intermediate part of the nozzle 5 and through the symmetrical slotted supply channels 7 of the inner cylinder 6 is fed into the vortex chamber (VC) of the nozzle. Due to the inclined placement of slotted channels relative to the axis of the housing, the liquid enters the vortex chamber at different distances from the axis and has a different moment of the amount of movement (1):

$$M = q \cdot v_{in} \cdot R, \quad (1)$$

where q – weight of liquid entering the nozzle vortex chamber in this cross-section; v_{in} – speed of the liquid trapped in the vortex chamber; R – swirling radius of the liquid in the control section of the inlet channel.

Depending on the position of the plunger, both the flow rate of liquid and the nature of the flow of liquid from the nozzle jet changes. When the plunger is in the lower position, the liquid enters the vortex chamber tangentially through the slotted channels and has the maximum radius and moment of the amount of movement, which is typical for a centrifugal nozzle (there is an air vortex along the nozzle axis, the spray swath is hollow, and the flow coefficient has a small numerical value). When the plunger is in the middle position, then in addition to the tangential flow, there is a liquid flow with zero twisting radius relative to the vortex chamber axis, that is, the liquid enters the twisting chamber with different moments of the amount of movement (from maximum to zero) and a spray swath is formed at the outlet of the nozzle, which is characteristic of a centrifugal jet nozzle.

In the upper position of the plunger (the slotted channels are completely open), the liquid entering the vortex chamber through the upper and lower parts of the supply channels have the maximum torque, but are directed towards each other. The resulting moment of rotation is zero, which is typical for a jet nozzle: the angle of the spray swath is insignificant, the flow rate of the nozzle takes a larger numerical value.

For this study, injectors were manufactured with one liquid supply channel to the vortex chamber (channel width 4 mm) and a vortex chamber diameter of 20 mm (Fig. 1). Replaceable nozzle jet diameters are 4; 5; 8 mm. Other design dimensions are shown in Figure 2a. The area of the inlet channel was changed by moving the plunger. Nozzles with two inclined supply channels had a twisting chamber diameter of 15 mm and nozzle diameters of 4, 5, 8 mm. Other nozzle sizes were selected according to the recommendations for their design, conditions of geometric similarity, and pressure characteristics of the pump unit. A spillway stand was created to conduct experiments (Fig. 3), which consists of a measuring tank 1, a pump 2, a liquid flow meter 3, a pressure gauge 4, an adjustment tap 5, and a test nozzle 6. The liquid flow rate was determined using a rotary flow meter of the KV-1.5 type, the accuracy class – 1.5. The liquid pressure was monitored with an OBM1-160 pressure gauge, accuracy class – 1.5.

The flow rate was determined using the equation (2):

$$\mu_t = \frac{Q_a}{Q_t} = \frac{V/\tau}{f_n \cdot \sqrt{2 \cdot g \cdot \Delta P}}, \quad (2)$$

where Q_a, Q_t – actual and theoretical fluid flow rates, respectively; V – volume of liquid during the experiment; τ – experiment time; f_n – spray nozzle jet area; ΔP – liquid pressure drop, m.

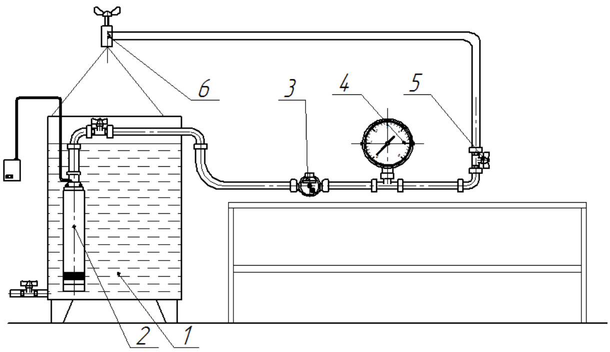

Figure 3. Nozzle test stand

Source: developed by the authors

An important characteristic of the nozzles is the uniform distribution of liquid in the spray swath. To determine this characteristic, a four-beam sampler was manufactured, in which the rays are placed at an angle of 90^0 with test tubes installed through 50 mm. Injectors with one and two supply channels were examined. Experimental data were processed using Mathcad.V15.M050, a Microsoft Office Excel table processor, graphs were created in CurveExpert-Pro-2.2.0, a Microsoft Excel table processor.

Methods of conducting a physical experiment. The study of injectors on the hydraulic stand provided for preparatory operations, including adjusting the verticality of the nozzle position, installing the pressure gauge so that its axis coincides with the middle of the nozzle mixing chamber. In the experimental studies, water was used as the working fluid to fill the container. After the working cycle, the nozzle water returned to the same tank, that is, the working fluid circulated in a closed loop, and its physical properties were the same for all experiments. This allowed excluding the influence of these factors on the results of experiments. The physical properties of the liquid from temperature were estimated based on tables of the thermophysical state of water and water vapour. Changing the operating modes of the injectors was carried out in any order (randomisation of the experiment) by adjusting the liquid pressure with a control valve, which was controlled by a pressure gauge. At least three experiments were performed on each mode. Each experiment lasted two minutes. The indicators of liquid and gas flow meters at the beginning and end of experiments, and the liquid pressure, were recorded.

Methodology of computer modelling of fluid hydrodynamics in the nozzle and spray swath. CFD modelling of the spray process and liquid flow in the nozzle elements, the specificity of which is the presence of inclined supply channels, was relevant. This will allow obtaining the required nozzle characteristics when designing nozzles of various capacities and sizes. Analysis of the software used to solve problems of this type and its functionality showed that the ANSYS software suite with the CFX hydro-gas dynamics module (ANSYS 2020 R2 version) is most widely used for modelling hydrodynamics and sputtering processes.

Physical model of the nozzle. For the simulation, a sprayer with inclined supply channels was selected in the plunger position, which leaves 60% of the height of the supply channels open. In this position, as the studies have shown, the liquid adopts the centrifugal jet type of flow and is characterised by a uniform distribution of the formed droplets over the cross-section of the spray swath, and the flow coefficient takes a high value. This distribution of liquid is most favourable for hollow absorbers, when its entire volume takes part in the mass transfer processes occurring in it. The characteristic dimensions of the nozzle under study were: the diameter of the nozzle channel – 4 mm, its length – 4.5 mm, and the radius of the twisting chamber – 7.5 mm.

Design area for calculating the movement of liquid in the elements of the nozzle and spray swath. In the three-dimensional Autodesk Inventor system, a design area has been created for modelling the hydrodynamics of liquids in sprayer elements with inclined supply channels. A separate area in the form of a cut-off cone was allocated for studying the movement of liquid at the outlet of the nozzle channel (Fig. 4).

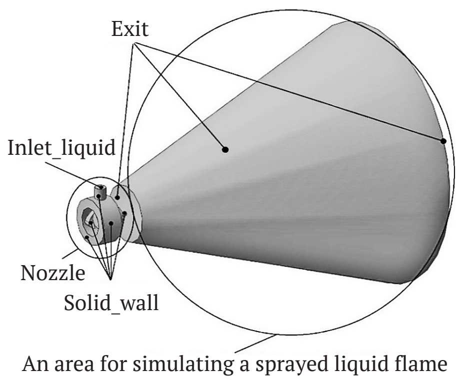

Figure 4. Calculation area and surfaces with boundary conditions
Source: developed by the authors

The design area for modelling a liquid spray swath using the method is created in the form of a cube or cylinder (Kumar *et al.*, 2022). In this case, a fairly significant calculation area was formed with an overestimated calculation zone. To optimise the numerical calculation by reducing the number of calculated cells, the authors suggest using the calculated area in the form of a truncated cone. Thus, a significant part of the calculated volume was removed, which lead to a decrease in the number of calculated cells, and therefore to a reduction in the computer calculation time.

For modelling the selected nozzles with inclined channels, the following dimensions of the design zone were established: the diameter of the design zone at the top of the cone (at the level of the end section of the nozzle) was 50 mm, the angle at the top was 38°, which coincides with the spray angle of the nozzle at this position of the plunger, the diameter of the design zone at the base of the cone – 225 mm. The created calculation area file was transferred to the Design Modeler calculation block of the CFX module in the ANSYS 2020 R2 software suite.

Calculation grid. Based on the calculation area in the mesh block, a calculation grid was automatically created with 288,380 calculation cells of tetraidal shape (the average size of the cell edge – 4 mm). The design grid of the spray swath was clarified by placing small cells in the area with a high concentration of liquid (near the nozzle channel). Larger calculation cells were placed near the outer walls. For a correct computer calculation, the number of cells and their size in the calculated area were the most important indicators of the accuracy of the simulation results. To solve problems related to calculating the size of droplets and the spray mechanism for these dimensions of the design zone, the number of design cells should be within 20 million, which requires powerful computer software.

Configuring calculation parameters. For this study, the problem type was established – non-stationary. The duration of the computer calculation, i.e., the time of stay of the liquid in the calculated area, was 0.5 seconds, and the step between iterations – 0.001. The properties of working media for the following environmental parameters are loaded from the ANSYS library database: atmospheric pressure (101,325 Pa), temperature 25°C. To solve such problems using the Navier-Stokes equations describing the motion of a viscous incompressible fluid, it is recommended to choose *k-ε* a turbulence model that provides a trade-off between the accuracy of results, the duration of machine calculations, and the requirements for the calculation grid. When modelling the swath of liquid sputtering into an unlimited space with a gas phase, the multiphase Euler model "Mixture" was used, in which the phases were considered as mutually penetrating continuums (mass and energy transfer occur without interference). The water surface tension coefficient v ($v = 0.072$ N/m) was determined (from the phase property library), and the drop resistance coefficient was assumed to be constant $C_d = 0.44$, which corresponds to the car model and turbulent motion of liquid droplets in the gas phase, assuming that the drop is spherical. The choice of the coefficient of resistance to liquid droplets should be sufficiently justified, since its numerical value depends on the mode of movement of drops (numbers *Re*) and numbers *We*, which characterises the stability of the drop shape. In the conditions of droplet flight, Weber's number is variable along the length, as is the criterion *Re* due to the resistance to droplet movement.

The initial conditions for computer calculation were the gas phase velocity equal to zero, atmospheric pressure, and volume fraction of air equal to 1 in the calculated region. The boundary conditions of simulation modelling are the area of liquid Inlet to the "Inlet_liquid" nozzle, the cylindrical and end walls of the "Solid_wall" nozzle, and the liquid outlet from the "Exit" computational area. Surfaces with applied boundary conditions are shown in Figure 4. The turbulence parameters of the model were determined: k = 0.05, ε = 10, and the value of the volume fraction of water 1 and air 0 was set. For the boundary condition for phase exit from the "Exit" computational area, the "Opening" condition type was set and the pressure was set to atmospheric pressure. The convergence criterion of results was set at 0.0001. With such phase parameters and boundary conditions, the 3D model was sent to the next calculation unit solution of the CFX module. Some specific settings were set that correspond to the technical capabilities of the computer. The possibility of performing calculations with double accuracy was established.

When the convergence criterion was reached at the level of 0.0001, the results were sent to the CFD-Post PostProcessor storage, where they were processed and printed in the form of graphic images, colour gradations, digital objects, or any other (combined) results. For liquid sprayers with inclined supply channels, important calculation results are the liquid flow lines in the nozzle elements, the liquid velocity in the control sections of the nozzle, the volume concentration of the liquid or gas phase in the design zone, the pressure distribution in the cross-section of the spray swath and at different distances from the nozzle.

Results

Experimental study of injectors with one and two supply channels. The nozzles were tested on a hydraulic test stand (Fig. 3): 1. With one inclined supply channel (Fig. 1). The area of the inlet channel was changed by moving the plunger. 2. With two inclined supply channels (Fig. 2). Addiction μ (flow rate) from A (geometric characteristics) for injectors with one supply channel $\mu = f(A)$ is shown in Figure 5. Flow rate for the jet nozzle ($A = 0$) and centrifugal-jet ($0 < A < 1$) depends on the ratio of geometric dimensions of the nozzle (d_c/D_{vc}) D_{vc} – diameter of the nozzle vortex chamber). As follows from the above graph, as the numerical value of the geometric characteristic of the nozzle increases from 0 (jet flow) to 1 (centrifugal-jet flow of liquid from the nozzle jet), the flow coefficient decreases.

Flow rate μ for the range of values of a geometric characteristic $A < 1$ expressed by an equation of the form (3):

$$\mu = B \cdot \left(\frac{d_c}{D_{vc}}\right)^n, \quad (3)$$

where B – coefficient; n – degree indicator.

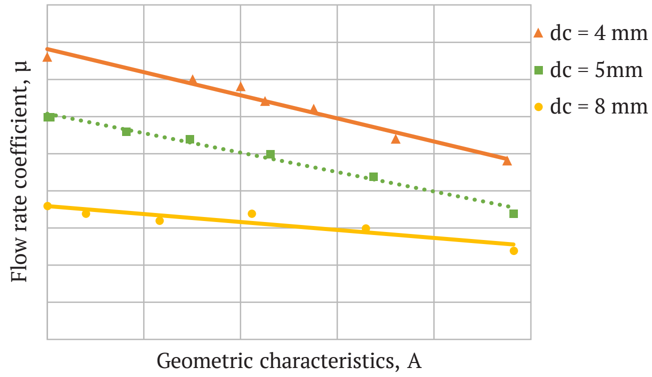

Figure 5. Dependence of µ (flow rate) from A (geometric characteristics of the nozzle)
Source: developed by the authors

The dependence of the flow coefficient μ for nozzles with a single inlet channel on the geometric simplex is known in the form (4):

$$\mu_\partial = 0.545 \cdot \left(\frac{d_c}{D_{vc}}\right)^{-0.206}, \qquad (4)$$

The amount of energy loss due to friction and local resistances during flow through the nozzle elements Σδ defined from the expression (5):

$$\mu_\partial = 0.545 \cdot \left(\frac{d_c}{D_{vc}}\right)^{-0.206}. \qquad (5)$$

A new dependence of the nozzle flow rate on the pressure of the liquid supply to the nozzle of a nozzle with one inlet channel was established for different areas of the inlet channel (different heights of the open channel) (Fig. 6). As the pressure of the liquid increases, and therefore its speed in the nozzle jet, hydraulic losses increase, which leads to a decrease in the flow rate. Within the limits of the studied pressures, the reduction in the flow coefficient was not significant.

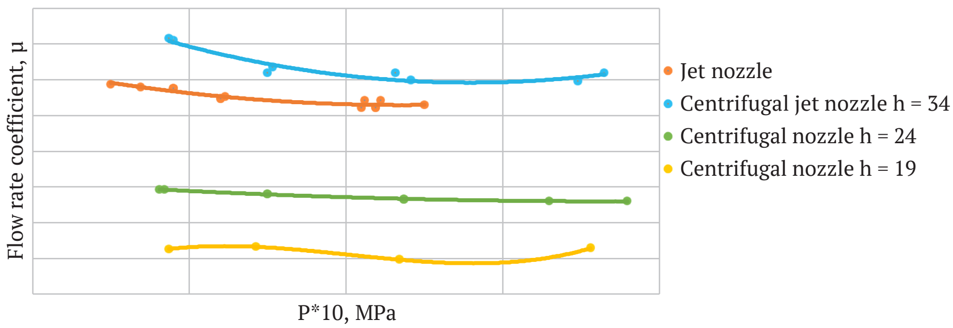

Figure 6. Dependence of the nozzle flow rate (μ) from the pressure of the liquid in the nozzle of the nozzle (P)
Note: input channel height $h = 52$ mm
Source: developed by the authors

According to this graph, in the range of studied pressures, the liquid flow rate at a given height of the open inlet channel depends only on the pressure. The flow coefficient in the studied pressure range can be assumed to be constant when calculating the performance of injectors. It has been experimentally established that the number of inclined supply channels also affects the energy efficiency of injectors: injectors with two supply channels have a higher flow rate, which is associated with a decrease in the speed of the input flow into the mixing chamber of injectors and losses in the mixing chamber itself. A graph of the dependence of the flow coefficient on the liquid inlet pressure for injectors with fully open channels (jet nozzle) is shown in Figure 7.

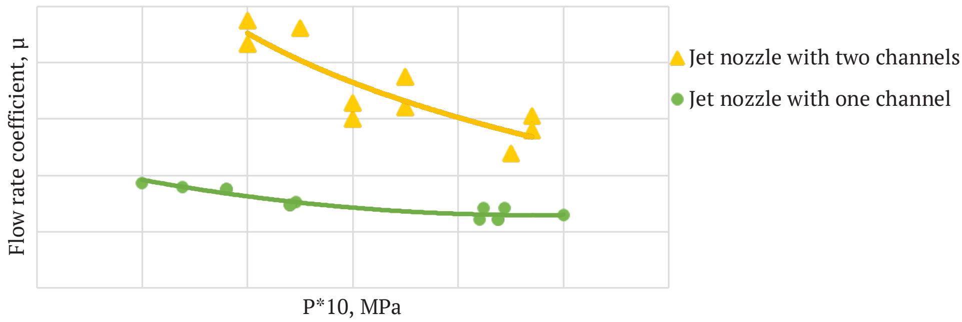

Figure 7. Dependence of the nozzle flow rate (μ) from the pressure of the liquid in the nozzle of the nozzle (P) when the channels are fully open (jet nozzle)
Source: developed by the authors

Considering the number of supply channels, the dependence for the nozzle flow rate will take the form (6):

$$\mu_\partial = n \cdot 0.545 \cdot (\frac{d_c}{D_{vc}})^{-0.206}, \quad (6)$$

where n – coefficient that considers the number of supply channels; $n = 1$ – for injectors with one supply channel; $n = 1.1$ – for injectors with two supply channels.

The relationship $\mu = f(A)$ for centrifugal jet nozzles with a nozzle diameter of 5 mm and 65% open inlet channels is shown in Figure 8.

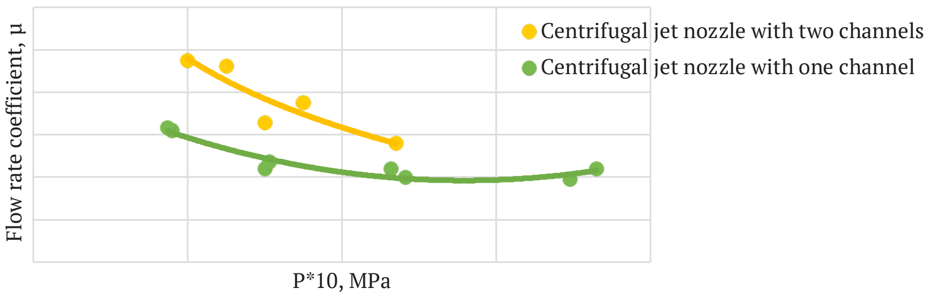

Figure 8. Dependence of the flow rate (μ) from the pressure of the liquid (P) for centrifugal jet injectors with one and two supply channels that are open by 65%
Source: developed by the authors

An important criterion for the effective operation of equipment, in particular ejectors, is the uniform distribution of liquid droplets over the cross-section of the spray swath. Dependence of relative irrigation density q_{rel} (the ratio of the mass of the liquid in the studied section to the maximum) from the relative radius R_{rel} (the ratio of the distance from the axis of the spray swath to the annular section under study to the maximum radius of the torch in this section) for a nozzle with one supply channel is shown in Figure 9, with two supply channels in Figure 10.

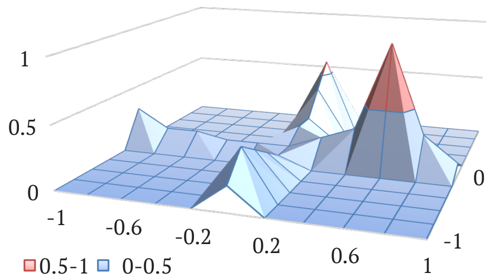

Figure 9. Dependence of the relative irrigation density (q_{rel}) from the relative radius (R_{rel}) for a nozzle with one supply channel at $P = 0.2$ MPar
Source: developed by the authors

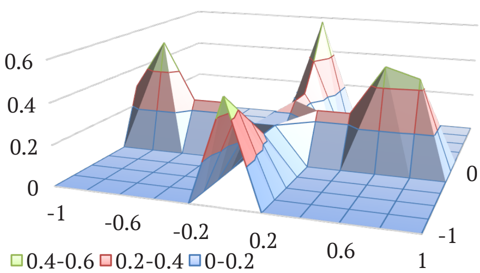

Figure 10. Dependence of the relative irrigation density (q_{rel}) from the relative radius (R_{rel}) for a nozzle with two supply channels at $P = 0.2$ MPa
Source: developed by the authors

The study of all injectors was carried out at the height of the open part of the input channels of 65% (centrifugal jet flow). The liquid pressure in the nozzle jet was 0.2 MPa. The results show that the number of input channels significantly affects the uniformity of liquid distribution over the swath cross-section. With one supply channel (Fig. 9) there was a significant uneven distribution of liquid: on the side of the supply channel, the irrigation density was higher. With two symmetrical supply channels (Fig. 10), the irrigation density in the annular sections was uniform.

CFD-modelling of liquid flow in the elements of the nozzle and spray swath. Modelling of the liquid flow in the nozzle elements was carried out at a liquid flow rate (mass) of 0.2 kg/s, which corresponds to the liquid pressure in the supply pipeline of 0.25 MPa with a nozzle diameter of 4 mm. Figure 11 shows a graph of the distribution of the liquid velocity in the nozzle elements along the axes. The speed of the liquid in the supply pipe reached a numerical value of 2.6 m/s, which is quite high, but it is within the recommended limits for the flow of liquid in pressure pipelines. In slotted channels, due to a decrease in the area, the fluid velocity increases to 2.7 m/s. The liquid in the vortex chamber has a tangential velocity that lies in the range of 1.7...2.6 m/s, and an axial velocity that is maximum along the nozzle axis – 2.2 m/s and 0.8 m/s near the walls of the vortex chamber.

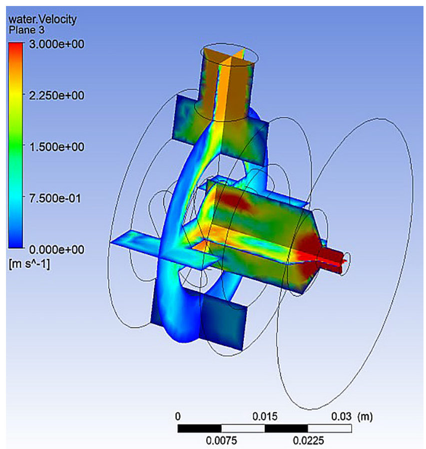

Figure 11. Fluid velocity in nozzle elements with inclined supply channels
Source: developed by the authors

In the nozzle channel, the liquid velocity increases to 19.2 m/s in Figure 12 shows a graph of the velocity distribution in the nozzle. A laminar layer is placed near the walls of the nozzle channel at a distance of up to 0.5 mm from the solid wall, in which the speed change is subject to a linear dependence. The presented data are difficult to obtain experimentally, since the dimensions of the pressure measuring probes are commensurate with the dimensions of the zone in which the velocity is measured.

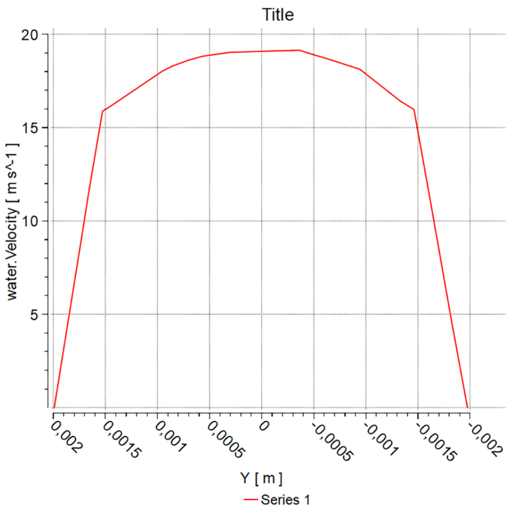

Figure 12. Graph of the dependence of the liquid velocity in the nozzle channel on its radius of the liquid velocity in the outlet section of the nozzle of a centrifugal jet nozzle with inclined supply channels
Source: developed by the authors

Analysis of the latter dependence shows that in the nozzle channel there was an asymmetry in the distribution of the fluid velocity relative to the axis, which suggests that the design of inclined supply channels was inaccurate (there was an asymmetry). This disadvantage can be corrected by performing a longer nozzle channel, in which the speeds relative to the axis are equalised. To visualise the flow of liquid in the nozzle elements, Figure 13 shows the current lines. Due to the inclined location of the inlet channels, the liquid has both a pronounced tangential flow near the walls of the twisting chamber and an axial flow along the nozzle axis. The interaction of flows leads to the development of a general flow with tangential movement of the liquid near the walls and axial in the centre of the nozzle channel.

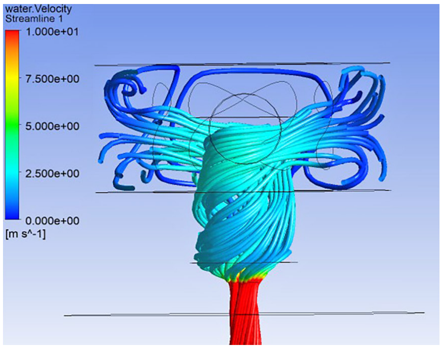

Figure 13. Development of a centrifugal jet spray swath in the nozzle when the input channels are tilted
Source: developed by the authors

Simulation of a liquid spraying swath into an unlimited space. The next logical stage of research was to determine the regularities of the development of a liquid spray swath at the outlet of the nozzle channel. The distribution of the volume concentration of liquid in the calculated zone (Fig. 14). The spray swath angle was 33⁰, and the uniformity of the liquid concentration over the cross-section was achieved at a distance of about 150 mm (cross-section *I-I*).

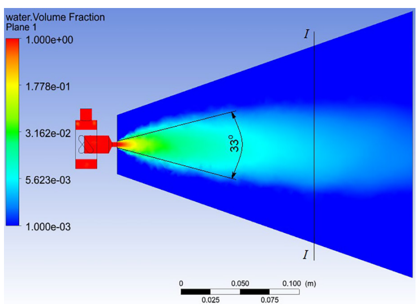

Figure 14. Volume concentration of liquid in the computational area
Source: developed by the authors

Using computer modelling, the pressure distribution over the cross-section of the sprayed liquid swath in the control sections from the nozzle channel section was established (Fig. 15).

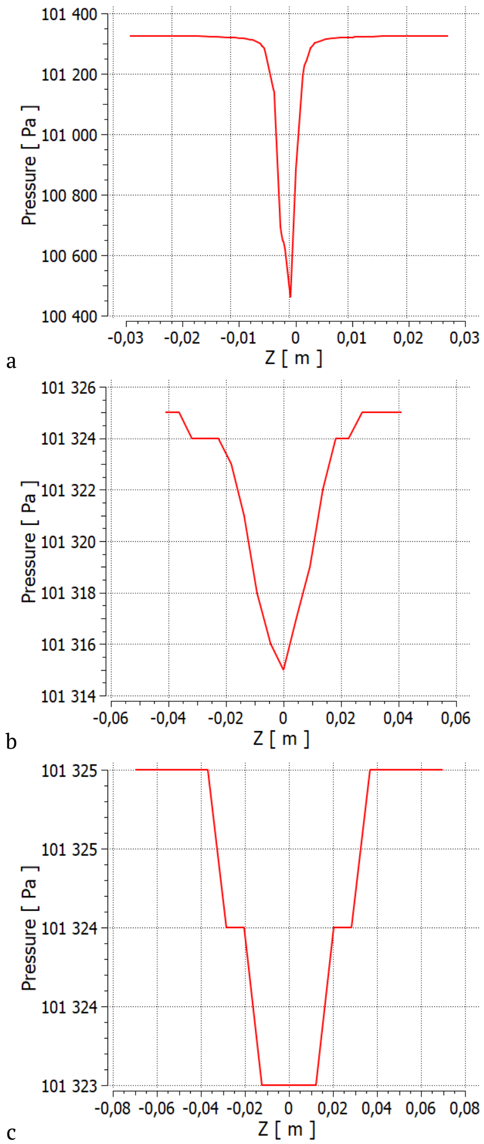

Figure 15. Dependence of the liquid pressure in the nozzle spray swath on the radius of the control sections at a distance from the nozzle channel cut-off

Note: a – 5mm; b – 50mm; c – 150mm

Source: developed by the authors

It was established that when the liquid is sprayed by the nozzle inside the nozzle swath, a vacuum occurs, and it is variable along the length of the swath: the maximum along the axis in the root zone at a distance of 5 mm from the nozzle channel, decreases with distance from the axis and with distance from the nozzle channel. If the distance from the nozzle channel is more than 150 mm, there is no vacuum along the axis of the spray swath. Experimental confirmation of this result of simulation modelling was obtained during the study of the spray swath of centrifugal jet nozzles. It was established that the maximum vacuum along the axis of the spray swath reaches 10 Pa. The obtained data on the pressure distribution in the nozzle spray swath using CFD modelling were obtained for the first time. This explains the ejection of the gas phase inside the spray swath, which is important when explaining ejection processes in jet apparatuses.

Validation of research results on the consumption characteristics of injectors. The results obtained experimentally and according to computer modelling data for one of the most important parameters of injectors – the flow coefficient (the ratio of the actual liquid flow rate to the theoretical one) are compared.

The actual volume flow rate of the liquid was calculated by the equation (7):

$$Q_r = \frac{m_r}{\rho_r \cdot \tau}, \tag{7}$$

where m_l – weight of liquid during the experiment (50 kg), ρ_l – liquid density (1,000 kg/m^3), τ – experiment time.

The theoretical fluid flow rate was as follows (8):

$$Q_\tau = v_\tau \cdot f_j, \tag{8}$$

where v_τ – theoretical rate of liquid leakage from the nozzle, f_j – nozzle jet area.

From the balance equation of energies (the potential energy of the fluid pressure is converted into kinetic energy without loss) in the form of the Bernoulli equation (9):

$$\frac{p}{\rho_r \cdot g} + \frac{v_r^2}{2 \cdot g} = const, \quad (9)$$

where p – pressure of liquid in the nozzle, g – acceleration of gravity.

The theoretical velocity of the liquid in the nozzle jet was calculated (10):

$$v_r^t = \sqrt{\frac{2 \cdot p}{\rho_r}} = \sqrt{\frac{2 \cdot 0{,}25 \cdot 10^6}{10^3}} = 22.36 \text{ m/s}. \quad (10)$$

Actual flow rate:

$$\mu = \frac{Q_r}{Q_\tau}. \quad (11)$$

Processing of experimental data made it possible to determine the dependence of the flow coefficient on the main geometric simplex of the nozzle (d_j/D_n) in the form of (12):

$$\mu = 0.545 \cdot \left(\frac{d_j}{D_n}\right)^{-0{,}206}. \quad (12)$$

The flow coefficient was calculated using the following formula for a nozzle with a jet diameter $d_j = 4$ mm and vortex chamber $D_n = 15$ mm. Numeric value: $\mu = 0.716$.

The actual flow rate can be directly calculated by considering the actual second flow rate of liquid through the nozzle and the fluid flow rate for the ideal nozzle (excluding resistances) (13):

$$\mu = \frac{Q_r}{Q_\tau} = \frac{0{,}2}{22{,}36 \cdot 3{,}14 \cdot (0{,}002)^2} = 0.71. \quad (13)$$

Determination of the nozzle flow rate using an empirical equation and directly from the experiment gave the same results for the studied nozzle with inclined supply channels.

The experimental value of μ was compared with the value obtained by CFD modelling. The average velocity of the liquid in the nozzle jet was determined from the condition that the area of the figure describing the instantaneous velocity of the liquid was equal to the area of the rectangle, the ordinate of which was the average velocity of the liquid in the nozzle. According to this method of calculating the average speed, it was equal to $v_m = 15.09$ m/s. The liquid consumption based on the simulation results was:

$$Q_m = v_m \cdot f_j = 15.09 \cdot 3.14 \cdot 0.002^2 = 0.1895 \text{ m}^3\text{/s},$$

and the flow rate was equal to $\mu = 0.696$.

The relative error in finding the flow rate from experimental data and computer modelling was 3.6%. Such comparative calculations of flow coefficients were also performed for other injectors under study.

Discussion

When modelling the process of spraying liquid with a nozzle, an important step is to choose the turbulence model that will be used in calculations. The presented studies were conducted with a standard *k-ε* model that provides for a trade-off between accuracy and calculation time. When performing a numerical study of the spraying process by N. Ayuba & G. Lopes (2023) model verification was performed using *k-ε* Realisable, *k-ε RNG*, standard *k-ω, k-ω SST*, Reynolds Stress Models, Boussinesq, Large Eddy Simulation on the possibility of application in simulation modelling. As a result, it was concluded that the standard model *k-ε* can be used with sufficient accuracy to describe tasks of this type. R. Wu *et al.* (2024) modelling of diesel fuel sputtering at high pressures using a turbulence model *k-ε* and a good agreement with experimental data is shown. The choice of a model in computer modelling was the subject of the work of A. Metsue *et al.* (2024), which compares the accuracy of calculations for six turbulence models. It was established that *k-ε RNG* and *k-ω SST* models better describe the flow structure in the flow part of the ejector, and *k-ω SST* model better describes the process of mixing two streams. J. Xiao *et al.* (2022) presented the results of modelling ejectors using *k-ω SST* and standard *k-ω* and *k-ε* turbulence models. It was established that the most reliable results are achieved when modelling the operation of ejectors on a standard *k-ε* model. Thus, the presented data indicate that the application of the standard turbulence model *k-ε* with numerical calculations of the spray swath provides a fairly high-quality picture of the process and reliable numerical results, which is also confirmed by the presented results.

One of the most important parameters for modelling sputtering or ejection processes in jet apparatuses is to set the required number of calculated cells. On the one hand, the larger the number of cells involved in the calculation process, the more accurate the result. However, this increases the requirements for the technical side of computers and the calculation time. The basic rule when creating a calculation grid is the type of problem that is solved during the calculation process. Thus, to simulate the process of forming a swath and determine its dispersed composition, the cell size should be smaller than the size of droplets. For different types of injectors and pressures at which liquid dispersion occurs, the choice of the number of calculated cells requires an individual approach based on a priori information about the geometric size of droplets. In this case, the maximum number of cells was selected.

In the case of modelling a liquid spray swath and the need to obtain consumption characteristics, a significantly smaller number of cells is sufficient without reducing the accuracy of the results. There are known studies of a spray swath when operating an ejector with a number of cells of 25,820 (Nguyen *et al*., 2024). The researchers concluded that there are enough cells for the general characteristics of the spray swath (visualisation) without detail, but the reliability of quantitative characteristics of the phases in the swath is small. Y. Hou *et al*. (2022) performed ejector modelling and comparison of calculations in a calculated grid of 53,980 cells and 107,580 cells. It was found that the error of the simulation results – 4.45%. The researchers did not specify how much the simulation data correspond to the experimental data. According to the results of studies of the spray swath, with such a large number of cells, a contour image with indistinct borders can be obtained.

When setting the problem of determining the dispersed composition of droplets formed during spraying, the number of calculated cells should be such that its average size does not exceed the geometric size of the droplets. For example, when determining the drop size of a centrifugal nozzle with a nozzle diameter of 0.94 mm and the liquid pressure in the nozzle of 0.2 MPa, the volume-surface diameter of the drops d_{32} – 130 microns, which corresponds to a geometric drop diameter of 98 microns. In other words, this is the minimum size of the calculated cell, but considering the polydispersity of liquid droplets, the cell size should be significantly smaller (at least twice).

There is a known modelling of the process of droplet development during the spraying of liquids with the number of computational cells of the order of 10 million, which allows establishing a qualitative and quantitative picture of the spraying process at different moments of time and droplet sizes, which was confirmed by experimental data of K. Vhora *et al.* (2024) with an error in the range of 11...27%. In this study, the number of design cells in the spray swath area of the nozzle was 288,380. Tetraidal cells (the average size of the cell edge – 4 mm). This number of cells is generally sufficient to determine the main characteristics of the spray swath (spray angle) and establish the distribution of phase concentrations in it along the length and cross-section, that is, to obtain a high-quality visual picture of the development of the swath and its numerical values (phase velocity, pressure, concentration distribution).

It should also be added that simulations were performed with both a larger and smaller number of calculated cells. When the calculated zone was divided into fewer than 50 thousand cells, the simulation results were not accurate. If the number of cells exceeded 500 thousand, the calculation accuracy increased, but its duration increased almost 5 times and amounted to 9…10 hours. The correctness of the calculations performed was confirmed by experimental data.

The results of numerical modelling for determining the pressure in the spray swath at different distances from the nozzle are confirmed by numerous studies, for example, J. Xiao *et al.* (2022), who analysed the operation of an ejector with a cylindrical mixing chamber based on the Navier-Stokes equation in the CFD system. Analysis of the experiment shows that the pressure in the vortex chamber gradually decreases due to the recirculation of the flow and penetration of atmospheric air from the outlet neck. Similar data for injectors with inclined supply channels were obtained as a result of simulation modelling: the maximum vacuum along the axis in the basal zone at a distance of 5 mm from the nozzle channel and decreases with distance from the end of the nozzle channel. Thus, CFD modelling in the modules of the ANSYS software suite (version ANSYS 2020 R2) demonstrates significant potential for application tasks for determining the characteristics of the spray swath and allows visualising the spray process and predicting droplet sizes with sufficient accuracy with appropriate adjustment of the calculated grid. This method deserves further development and improvement, demonstrating its versatility and practical usefulness.

Conclusions

A comprehensive study of nozzles with inclined supply channels, which are used as active ejector nozzles in jet sulphitators in sugar production, was carried out. The hydrodynamics of liquid movement in the nozzle elements was established and the process of forming a spray swath was studied. The dependence of the liquid flow coefficient on the geometric parameters of the nozzle and the number of supply channels was experimentally established: a nozzle with two symmetrically placed supply channels relative to the axis had a 20…25% higher flow coefficient. In addition, with two supply channels, the irrigation density of the spray swath cross-section was significantly improved.

A numerical study of the liquid spray process by injectors was carried out using the Ansys 2020 R2 software suite. The standard k-ε model with the number of computational cells 288,380 was used for modelling and it was proposed to reduce the computational zone to a truncated cone, which would optimise the calculation time and provide a better picture of the spray swath and its quantitative characteristics. A computer study of hydrodynamics in the form of liquid flow lines in nozzle elements with inclined channels showing the stages of development of a centrifugal jet flow of liquid was carried out. A numerical distribution of the fluid velocity in the end section was found on the nozzle section, which was maximum along the nozzle axis of 19.2 m/s, and a laminar layer was located near the walls of the nozzle channel at a distance of up to 0.5 mm from the solid wall. Computer modelling has obtained new results confirming the presence of a low-pressure zone along the spray swath axis, which is maximum in the basal zone, decreases as it moves away from the nozzle end and disappears at a distance of 150 mm from it. The results of the nozzle flow rate coefficient were validated. The relative error in determining the nozzle flow rate during CFD modelling compared to experimental results does not exceed 5%.

Further research will focus on visualising and determining the quantitative characteristics of the liquid flow in the nozzle elements and the characteristics of the spray swath of other types of nozzles and when scaling. Such data will help to design injectors of various capacities with specified characteristics, to achieve an efficient mode of operation of equipment for the food industry.

Acknowledgements

The authors express their gratitude for the opportunity to perform work on equipment and using material resources of the Department of Technological Equipment and Computer Design Technologies of the National University of Food Technologies.

Conflict of Interest

The authors of the study performed the work equally and have no mutual claims of authorship.

References

[1] Ayuba, N., & Lopes, G.C. (2023). Investigation of the influence of turbulence models on cough droplet evaporation: Comparing (SST) k-Ω, k-ε, and Reynolds Stress (RSM) turbulence models. In Y. Iano, O. Saotome, G.L. Kemper Vásquez, C. Cotrim Pezzuto, R. Arthur & G. Gomes de Oliveira (Eds.), *Proceedings of the 7th Brazilian Technology Symposium (BTSym 2021)*. *Smart innovation, systems and technologies*, 207 (pp. 151-162). Cham: Springer. doi: 10.1007/978-3-031-04435-9_15.

[2] Haidl, J., Mařík, K., Moucha, T., Rejl, F.J., Valenz, L., & Zednikova, M. (2021). Hydraulic characteristics of liquid-gas ejector pump with a coherent liquid jet. *Chemical Engineering Research and Design*, 168, 435-442. doi: 10.1016/j.cherd.2021.02.022.

[3] Han, H., Wang, H., Zhang, Q., Du, Y., Wang, H., & Wang, H. (2023). Investigations of the effects of two typical jet crushing methods on the atomization and dust reduction performance of nozzles. *International Journal of Coal Science & Technology*, 10, article number 50. doi: 10.1007/s40789-023-00606-4.

[4] Han, H., Wang, P., & Liu, R. (2020). Experimental study on atomization characteristics and dust-reduction performance of four common types of pressure nozzles in underground coal mines. *International Journal of Coal Science & Technology*, 7, 581-596. doi: 10.1007/s40789-020-00329-w.

[5] Von Helldorff, H., & Micklow, G.J. (2019). Primary and secondary spray breakup modelling for internal combustion engine applications. *Journal of Multidisciplinary Engineering Science and Technology*, 6(4), 9882-9894.

[6] Hou, Y., Chen, F., Zhang, S., Chen, W., Zheng, J., Chong, D., & Yan, J. (2022). Numerical simulation study on the influence of primary nozzle deviation on the steam ejector performance. *International Journal of Thermal Sciences*, 179, article number 107633. doi: 10.1016/j.ijthermalsci.2022.107633.

[7] Kim, S., Nocivelli, L., Zhang, A., Voice, A.K., & Pei, Y. (2024). Realistic fuel spray modeling for gasoline direct injection engine applications. *International Journal of Engine Research*, 25(3), 388-404. doi: 10.1177/14680874231210929.

[8] Kramm, K., Orth, M., Teiwes, A., Kammerhofer, J., Meunier, V., Pietsch-Braune, S., & Heinrich, S. (2023). Influence of nozzle parameters on spray pattern and droplet characteristics for a two-fluid nozzle. *Chemie-Ingenieur-Technik*, 95(1-2), 151-159. doi: 10.15480/882.4895.

[9] Kumar, A., Ghobadian, A., & Nouri, J. (2022). Numerical simulation and experimental validation of cavitating flow in a multi-hole diesel fuel injector. *International Journal of Engine Research*, 23(6), 958-973. doi: 10.1177/1468087421998631.

[10] Lee, T.-W., Greenlee, B., & Park, J.E. (2021). Computational protocol for spray flow simulations including primary atomization. *Journal of Fluids Engineering*, 143(3), article number 031402. doi: 10.1115/1.4049115.

[11] Li, W., Zheng, Y., Shao, H., Wang, X., Chen, X., & Gao, Y. (2024). Experimental and numerical simulation studies on the atomization characteristics of the internal mixing nozzle and its field application. *Mining, Metallurgy & Exploration*, 41, 1099-1120. doi: 10.1007/s42461-024-00955-x.

[12] Metsue, A., Nesreddine, H., Bartosiewicz, Y., & Poncet, S. (2024). Theoretical analysis of the optimal ejector operation and design within an ejector-based refrigeration system. *International Journal of Refrigeration*, 168, 334-344. doi: 10.1016/j.ijrefrig.2024.09.020.

[13] Mudavath, V., & Arumuru, V. (2023). Impingement heat transfer characteristics of nozzles with inclined exits. *Experimental Heat Transfer*, 37(6), 541-556. doi: 10.1080/08916152.2023.2166162.

[14] Nguyen, Q.Q., Phung, D.V., Nguyen, K.T, Pham, H.Q, Pham, T.V., Vu, T.N., Pham, P.X., & Duong, C.Q. (2024). An approach to design an air-liquid supersonic ejector for producing aerosol spray. *SAE Technical Paper*, article number 2024-01-5068. doi: 10.4271/2024-01-5068.

[15] Oluwadero, T.A., Xuereb, C., Aubin, J., & Poux, M. (2023). Effect of jet nozzle position on mixing time in large tanks. *Processes*, 11(7), article number 2200. doi: 10.3390/pr11072200.

[16] Park, S., & Park, K. (2022). Principles and droplet size distributions of various spraying methods: A review. *Journal of Mechanical Science and Technology*, 36, 4033-4041. doi: 10.1007/s12206-022-0724-3.

[17] Santosh, Shirwal, S., Sushilendra, Raghavendra, V., Mareppa, H., & Vinutha, K. (2023). Assessment of nozzle spray characteristics for agriculture spraying. *International Journal of Environment and Climate Change*, 13(12), 527-536. doi: 10.9734/ijecc/2023/v13i123710.

[18] Shulhin, A., Samulieiev, V., Chemerys, Y., & Trehub, O. (2022). The model adaptation of the two-nozzle swirl atomizer for the design-experimental research of the working process in the main combustionchambers of turbojet engines. *Collection of Scientific Works State Research Institute of Aviation*, 18(25), 117- 122. doi: 10.54858/dndia.2022-18-18.

[19] Sijs, R., Kooij, S., & Bonn, D. (2023). Droplet size from Venturi air induction spray nozzles. *Experiments in Fluids*, 64, article number 43. doi: 10.1007/s00348-023-03582-2.

[20] Stallbaumer-Cyr, E.M, Aguilar, J., Betz, A.R., & Derby, M. (2024). The effects of Surfactin on sprayed droplets in flat fan, full cone, and low energy precision application bubbler nozzles: Droplet formation and spray breakup. *Frontiers in Mechanical Engineering*, 10, article number 1354664. doi: 10.3389/fmech.2024.1354664.

[21] Vambol, S., Khan, N.A., Khan, A.H., Kiriyenko, M., Borysova, L., Taraduda, D., Zakora, A., & Bilotserkivska, N. (2020). Developed jet-centrifugal spray devices: Experimental testing to establish the possibility of their application in plants spraying technologies. *Journal of Achievements in Materials and Manufacturing Engineering*, 102(1), 30-41. doi: 10.5604/01.3001.0014.6326.

[22] Verbiawan, E.A., Ramadhan, M.R., Sumada, K., Muljani, S., & Pujiastuti, C. (2023). Spray nozzle technology to accelerate seawater evaporation in conventional salt production. *Jurnal Teknik Kimia*, 18(1), 52-57. doi: 10.33005/jurnal_tekkim.v18i1.4125.

[23] Vhora, K., Janiga, G., Lorenz, H., Seidel-Morgenstern, A., Gutierrez, M.F., & Schulze, P. (2024). Comparative study of droplet diameter distribution: Insights from experimental imaging and computational fluid dynamics simulations. *Applied Sciences*, 14(5), article number 1824. doi: 10.3390/app14051824.

[24] Wu, R., Sun, C., & Gui, Y. (2024). Numerical analysis of atomization characteristics of fuel-jet in crossflow. *AIP Advances*, 14(8), article number 085115. doi: 10.1063/5.0222575.

[25] Xiao, J., Wu, Q., Chen, L., Ke, W., Wu, C., Yang, X., Yu, L., & Jiang, H. (2022). Assessment of different CFD modeling and solving approaches for a supersonic steam ejector simulation. *Atmosphere*, 13(1), article number 144. doi: 10.3390/atmos13010144.

[26] Xu, Z., Zhao, H., & Liu, H.-F. (2022). Atomization model based on the ligament-mediated spray mechanism for coaxial two-fluid air-blast nozzle. *Chemical Engineering Journal*, 450(1), article number 137986. doi: 10.1016/j.cej.2022.137986.

[27] Xu, Y., Liu, H., Wang, Z., Zhang, J., & Wang, J. (2024). Analysis of the effects of nozzle geometry on the cavitation water jet flow field using orthogonal decomposition. *Iranian Journal of Science and Technology*, 48, 119-132. doi: 10.1007/s40997-023-00647-9.
[28] Zhu, X., Lewballah, J.K., Fordjour, A., Jiang, X., Liu, J., Ofosu, S.A., & Dwomoh, F.A. (2021). Modelling of water drop movement and distribution in no wind and windy conditions for different nozzle sizes. *Water*, 13(21), article number 3006. doi: 10.3390/w13213006.

Дослідження та застосування форсунок з нахиленими підвідними каналами для струминних сульфітаторів цукрової промисловості

Віталій Пономаренко
Кандидат технічних наук, доцент
Національний університет харчових технологій
01601, вул. Володимирська, 68, м. Київ, Україна
https://orcid.org/0000-0002-0732-4783

Дмитро Люлька
Кандидат технічних наук, доцент
Національний університет харчових технологій
01601, вул. Володимирська, 68, м. Київ, Україна
https://orcid.org/0000-0001-8014-5776

Володимир Василів
Кандидат технічних наук, доцент
Національний університет біоресурсів і природокористування
03041, вул. Героїв Оборони, 15, м. Київ, Україна
https://orcid.org/0000-0002-2109-0522

Роман Якобчук
Кандидат технічних наук, доцент
Національний університет харчових технологій
01601, вул. Володимирська, 68, м. Київ, Україна
https://orcid.org/0000-0001-9777-5790

Євген Бабко
Кандидат технічних наук, доцент
Національний університет харчових технологій
01601, вул. Володимирська, 68, м. Київ, Україна
https://orcid.org/0000-0003-2389-007X

Юлія Слива
Кандидат технічних наук, доцент
Національний університет біоресурсів і природокористування
03041, вул. Героїв Оборони, 15, м. Київ, Україна
https://orcid.org/0000-0003-2592-6822

Анотація. Стаття присвячена дослідженню процесу розпилення рідини форсунками, що застосовуються в струминних сульфітаторах цукрової промисловості або як окреме обладнання. Розпилювачі рідин відіграють основну роль в забезпеченні ефективної роботи тепломасообмінного обладнання внаслідок створення значної поверхні контакту. При проектуванні або аудиті його технічної досконалості необхідно мати достовірні гідродинамічні характеристики розпилювачів. Роботи такого направлення є актуальними. Метою статті було встановлення всіх характеристик течії рідини в елементах форсунки та факелу розпилення за допомогою комплексного дослідження (експериментального та комп’ютерного) для можливості інтенсифікації роботи обладнання, зокрема сульфітаторів для цукрової промисловості. Для досягнення поставленої мети було використано експериментальні (дослідження форсунок та ежекторів на гідравлічному стенді), CFD-дослідження зазначених об’єктів в модулях програми ANSYS (версія: ANSYS 2020 R2). Експериментально встановлено залежність коефіцієнта витрати форсунок від тиску рідини та кількості підвідних каналів для різних типів форсунок (струминної, відцентрово-струминної). Експериментально підтверджено, що рівномірність розподілення рідини в факелі форсунки з двома підвідними каналами була високою та достатньою для практичного застосування. При CFD-дослідженні характеристик факелу розпилення форсунок було запропоновано зменшити розрахункову область до зрізаного конусу, що дозволило скоротити час розрахунку. При моделюванні гідродинаміки рідини в форсунці визначено чисельно характерні швидкості як в елементах форсунки, так і в соплі. Встановлено наявність у прикореневій зоні розрідження, що досягала 865 Па для даної форсунки при тиску рідини на вході 0,25 МПа, що приводило до ежектування газу всередину факела. Наведена методологія розрахунку актуальна при визначенні характеристик форсунок, при розробці, проектуванні нових розпилювачів з заданою продуктивністю та факелом розпилення, що застосовуються і як активні сопла ежекторів, і як самостійне обладнання для диспергування технологічних рідин харчової промисловості

Ключові слова: виробництво цукру; сульфітація рідини; розпилювач; коефіцієнт витрати; CFD-моделювання; гідродинаміка; факел розпилення