Machinery & Energetics
Vol. 16, No. 2. 2025

Journal homepage:
https://technicalscience.com.ua/en

UDC 663.033
Doi: 10.31548/machinery/2.2025.58

Mykola Zablodskyi*
Doctor of Technical Sciences, Professor
National University of Life and Environmental Sciences of Ukraine
03041, 15 Heroiv Oborony Str., Kyiv, Ukraine
https://orcid.org/0000-0001-8889-8158

Denys Babak
Postgraduate Student
National University of Life and Environmental Sciences of Ukraine
03041, 15 Heroiv Oborony Str., Kyiv, Ukraine
https://orcid.org/0009-0000-8970-8598

Simulation of fluid spraying processes in a device for automatic control of anaerobic fermentation process parameters in biogas reactors

Abstract. The relevance of the study is justified by the rapid development of biogas technologies, which requires continuous improvement of methods and equipment to increase efficiency and ensure optimal productivity of the biogas and biomethane production process. The aim of this research was to improve and optimise devices for automatic control of anaerobic fermentation process parameters in biogas reactors. Methods of mathematical modelling of the 3D model in the Ansys Fluent environment, as well as methods of experimental studies to determine the spray spectrum of nozzles, jet distribution and the shape of the fluid spray cone were used. A constructive scheme of the device for automatic control of the parameters of the anaerobic fermentation process in biogas reactors, a mathematical model of the process of fluid spraying in the sensor cleaning system, which used equations describing the evolution of the kinetic energy of turbulence and the frequency of turbulence, are proposed. An automatic recorder capable of measuring voltage, current, and pressure with a frequency of about 120 measurements per second was created, which insures a high-precision removal of the load characteristics in the fluid supply system. Based on the results of modelling and experimental studies, the patterns of jet distribution and the shape of the fluid spray cone depending on the change in the diameters of the spray nozzles, were established. The simulation results demonstrated the high accuracy and reliability of the mathematical model, which was an important prerequisite for further complex simulation and optimisation. This study makes a significant contribution to the development of more efficient methods and means of automatic control of the parameters of the anaerobic fermentation process in biogas reactors, contributing to an increase in the productivity of the biogas production process

Keywords: pump load characteristic; Ansys Fluent; spray nozzle optimisatio; physical model; fluid flow lines; turbulence effects

INTRODUCTION

The development of new technologies in the field of anaerobic fermentation in biogas reactors requires continuous improvement and optimisation of the use of devices for automatic control of process parameters. For example, if the fermentation temperature is very low or fluctuates rapidly, it can reduce the efficiency and methane content of biogas (Shao *et al.*, 2021). In the work by G. Sudiartha *et al.* (2023), an in-depth comparison of the impact of

Article's History: Received: 26.01.2024; Revised: 28.03.2025; Accepted: 22.05.2025.

Suggested Citation:
Zablodskyi, M., & Babak, D. (2025). Simulation of fluid spraying processes in a device for automatic control of anaerobic fermentation process parameters in biogas reactors. *Machinery & Energetics,* 16(2), 58-69. doi: 10.31548/machinery/2.2025.58.

*Corresponding author

temperature shifts on microbial communities and biogas production was conducted, while A.C. Ibeaja *et al.* (2023) wrote that overall temperature stability helps maximise biogas production.

The regulation of microbial intracellular enzyme activity largely depends on temperature, and this, in turn, affects the metabolic activity of bacteria and the efficiency of anaerobic fermentation. W. Li *et al.* (2017) proved that changes in microbial metabolism or community dynamics also influence the functioning of anaerobic fermentation systems. An important step to create the required temperature conditions in biogas reactors is CFD analysis of heat exchanger model designs using ANSYS Fluent and appropriate control of temperature distribution and gradient (Kishan *et al.*, 2020).

The pH level should be maintained within certain limits, such as 6.8 to 7.2, to maximise microbial activity, especially methanogens, which are crucial for methane production (Lima *et al.*, 2023). Conversely, according to A. Cerón-Vivas *et al.* (2019), increasing the pH above 7.5 can decrease biogas yield.

Anaerobic fermentation processes, as investigated by scientists O.A. Aworanti *et al.* (2023), face operational difficulties such as complexity of substrate structure and characteristics, low productivity, inefficient biodegradability, and low stability. Therefore, to ensure optimal productivity of the biogas and biomethane production process, a more thorough assessment of the current state and available strategies is needed. In addition, for different applications of biogas/biomethane, it is advisable to select the appropriate biomass preprocessing technology before biogas production.

In this context, the issue of efficiency and reliability of measuring devices capable of operating in aggressive environments and various rheological properties of water substrates is particularly important. The use of measuring devices requires periodic cleaning and calibration of a significant number of sensors located in the working volume of biogas reactors. Therefore, it is important to study the process of spraying cleaning fluid and selecting the optimal characteristics of electromechanical fluid supply units and the location of nozzles in the device for automatic control of anaerobic fermentation process parameters in biogas reactors. M.A. Enache *et al.* (2012) noted that the overall productivity and reliability of the device would depend on the accuracy of calculating of the dynamic mode characteristics for low-power DC motors, which were the main electromechanical units of the fluid supply. There are known results of developing a program based on numerical calculation methods that allows solving integro-differential equations with variable coefficients by the Runge-Kutta method of the fourth order and allows establishing a preliminary determination of the dynamic mode behaviour of low-power DC motors with high accuracy. In addition, the results of the practical developments of M.A. Kovalenko & O.O. Samoilenko (2019), devoted to the creation of an experimental device for studying the parameters and characteristics of low-power DC motors, are published. Based on the results, a schematic diagram of testing these machines based on a microcontroller with the usage of modern highly sensitive sensors has been developed.

However, there is no systematic information on the design of devices for automatic control of anaerobic fermentation process parameters in biogas reactors, and therefore hydrodynamic analysis of fluid flow can only be conducted in comparison with related technological processes of spraying. Previous studies by M.H. Sadafi *et al.* (2016) had shown a significant impact of nozzle location, for example, on cooling efficiency, which confirms the importance of optimal design of these elements. The study by C. Yue *et al.* (2013) was aimed at analysing the hydrodynamic characteristics of a spherical underwater robot with three possible motions. The ANSYS Fluent software was used for hydrodynamic analysis of the three models and comparison of the simulation results with theoretical values of the drag coefficient, velocity vector, and pressure contours.

B. Monachan *et al.* (2019) investigated the behaviour of two-phase flow, which is important for various engineering applications. The authors used a volumetric multiphase fluid flow model in ANSYS Fluent to simulate two-phase air-water flow and focused on predicting stratified laminar flow. They studied the change in the volume fraction of air and water, mass flow rate, and pressure drop over time. Z. Liu *et al.* (2019) studied the spraying properties and rules of nozzle variation with different hole diameters at different atomisation pressures. The study used ANSYS Fluent software to analyse the relationship between average droplet size and spray pressure, as well as the dust suppression efficiency of different nozzle hole diameters.

Researches M. Juraeva *et al.* (2019) optimised a nozzle for a water jet loom using a computational approach. The efficiency of the nozzle was measured in terms of water resistance, which is crucial for woof insertion in water jet looms. The incompressible Navier-Stokes fluid equations with the standard k-epselon turbulence model were used to simulate the fluid flow, and the ANSYS Fluent software was used to calculate the flow. Geometrical parameters were optimised using the design of experiments (DOE) method using Minitab software. M. Gong *et al.* (2016) performed simulations of gas-liquid flows in pipes of various scales. Based on Frank's model, the emphasis is on the wall-to-wall lubrication force action distance. After modifications, the new model is used to simulate the gas-liquid flow in a transient two-phase flow apparatus with a multiphase circuit using the commercial software ANSYS Fluent.

Mathematical modelling was used in the study of the uniformity of fluid distribution during spraying in agriculture. E. Musiu *et al.* (2019) presented a mathematical model that is based on computational fluid dynamics (CFD) and showed that the flow rate affects the droplet size distribution. Models based on computational fluid dynamics are also used to predict and evaluate spray drift (Wang *et al.*, 2022). The use of the principle of maximum entropy, the model of linearised atomisation of the instability layer, and simulation of computational fluid dynamics should be considered promising (Renaudo *et al.*, 2022). There is a need for further work in order to achieve compliance of mathematical models with real values of fluid distribution during spraying.

The purpose of this study was simulation and experimental determination of the optimal parameters of the fluid spraying process when cleaning the sensors of the device for automatic control of the parameters of the anaerobic fermentation process in the biogas reactor.

MATERIALS AND METHODS

The research was conducted at the Department of Electrical Engineering, Electromechanics and Electrical Technologies of the National University of Life and Environmental Sciences of Ukraine during 2023-2024. The object of simulation and experimental research was a device for automatic control of the parameters of the anaerobic fermentation process in biogas reactors, the general view of which was presented in Figure 1a.

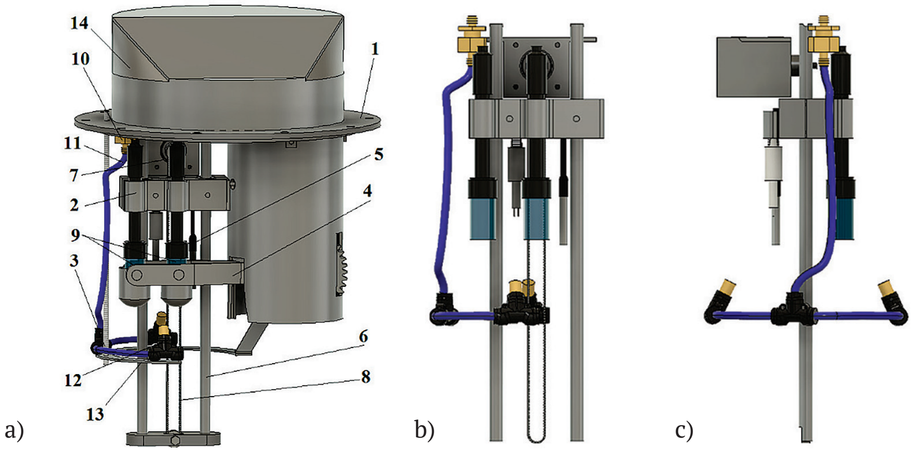


Figure 1. Device for automatic control of anaerobic fermentation process parameters in biogas reactors
Note: a) general view of the device, b) simplified model, XY plane projection, c) simplified model, XZ plane projection
Source: compiled by the authors

To optimise the time of mathematical simulation, a simplified model was used, the projections of which are shown in Figure 1b and Figure 1c, respectively. This model was used for mathematical simulation in the Ansys Fluent environment of the distribution of jets, the characteristics of the shape of the cone of the sprayed fluid depending on the change in the diameters of the spray nozzles.

The simulation was performed for atomisers with a nozzle diameter varying from 0.2 mm to 1 mm with an interval of 0.2 mm. In order to clarify the choice of the optimal atomiser, simulation was additionally carried out detailing the diameters of the nozzles with an interval of 0.1 mm. This approach provided a comprehensive view of the interaction of device parameters, taking into account various conditions and variants of atomiser designs.

The device for automatic control of parameters (Fig. 1a) contained a mounting flange 1, a movable carriage assembly 2, a cleaning system 3, a moving system of protective caps and calibration containers 4, and a calibration liquid supply unit 5. The movable carriage assembly 2 contains linear guides 6, a stepper motor with a toothed pulley 7, a movement belt 8 and sensors 9, which are fixed on a movable carriage. The cleaning system includes a pump 10, a pipeline 11 for supplying water and sprinklers 12 located on a fixed arc-shaped plate 13. Communication equipment, a power supply and a control device in the form of an automatic operation controller (not shown) are located in an isothermal container 14.

To determine the flow rate used in the simulation, the volume flow formula, which allows you to accurately calculate the speed of the liquid passing through the system per unit of time, was used:

$$Q = V \times S \times cos\theta, \qquad (1)$$

where: V is the average flow velocity in the cross-section; S is the cross-sectional surface of the flow; θ is the angle between the flow direction and the normal to the cross-sectional surface.

The flow rate was calculated for a specific pump with factory parameters: voltage – 9-14 V, current – 5A, flow – 5 l/min, pressure – 6.8 bar. The flow velocity used in the simulation was 0.0353 m/s. The adopted parameters ensure adequate reproduction of boundary conditions and turbulent effects, which is important for obtaining reliable results during mathematical modelling in Ansys Fluent.

The modelling domain had a rectangular shape with the dimensions shown in the computational diagram (Fig. 2a). This domain defined the space in which the hydrodynamic processes were modelled. The mesh of the modelling domain (Fig. 2b) had an average of 330,000 nodes and 1,760,000 elements (surfaces). The simulation environment consisted of water and air. The input condition for both phases was taken as the input velocity. In the specified area, the initial water velocity at the start of the simulation was set to 0.0353 m/s.

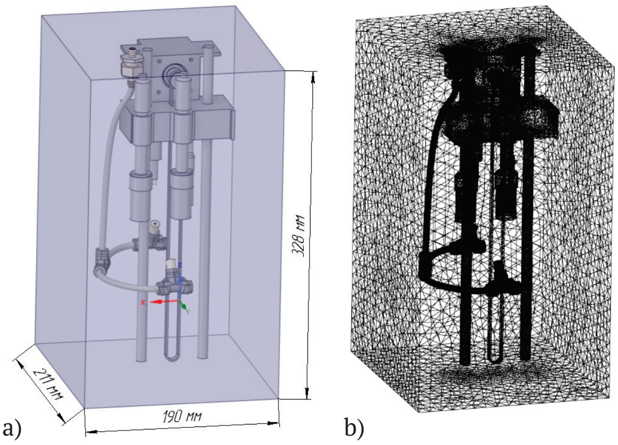


Figure 2. Modelling area
Note: a) calculation scheme; b) grid
Source: compiled by the authors

At the outlet of the system, the initial boundary conditions are used, which are given in the form of pressure. The working pressure is assumed to be 0 bar during the flow analysis, which allows to exclude the factor of pumping water to the outside. The k-omega model is used to account for turbulent effects. This model provides an opportunity to adequately reflect turbulent structures in the system and their influence on hydrodynamic processes (Monachan *et al.*, 2019).

When using the k-omega model in the ANSYS Fluent software package, equations that describe the evolution of the turbulence kinetic energy k and the turbulence frequency ω are used. The main equations of this model are as follows:

1. The equation for the kinetic energy of turbulence k:

$$\frac{\partial(pk)}{\partial t}+\frac{\partial(pku_i)}{\partial x_i}=P_k-\beta^* pk\omega+\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\sigma_k\mu_t}{\rho}\right)\frac{\partial k}{\partial x_i}\right], \quad (2)$$

2. The equation for the turbulence frequency ω:

$$\frac{\partial(p\omega)}{\partial t}+\frac{\partial(\rho\omega u_i)}{\partial x_i}=\alpha\frac{\omega}{k}P_k-\beta\rho\omega^2+$$
$$+\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\sigma_\omega\mu_t}{\rho}\right)\frac{\partial\omega}{\partial x_j}\right]+2(1-F_1)\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial\omega}{\partial x_i}, \quad (3)$$

where: ρ is the density of liquid or gas (kg/m^3); u_i – velocity components in the i direction (m/s); μ_t – dynamic viscosity of liquid or gas (Pa·s); β^* – turbulent viscosity (Pa·s), is the empirical constant of the k-ω model (dimensionless), usually equal to 0.09; α is the empirical constant of the k-ω model (dimensionless), usually equal to 5/9; β is the empirical constant of the k-ω model (dimensionless), usually equal to 0.075; σ_k is the empirical constant of the k-ω model, which affects the diffusion of k (dimensionless), usually equal to 2.0; σ_ω is the empirical constant of the k-ω model, which affects the diffusion of $\sigma_{\omega 2}$ (dimensionless), usually equal to 2.0; is an additional empirical constant for the combined model (dimensionless); F_1 is a blending function used to combine the k-ω model with other models (for example, with k-ε in the SST model) (dimensionless). This function determines what part of the combined model is used in a particular flow region. Turbulent viscosity is calculated as:

$$\mu_t=\rho\frac{k}{\omega}, \quad (4)$$

where: P_k is the term of the turbulence kinetic energy production that describes how the flow velocity is converted into turbulent kinetic energy. It was calculated as:

$$P_k=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}. \quad (5)$$

These equations take into account the production, diffusion, and dissipation of turbulent kinetic energy and turbulence frequency, which allows modelling a wide range of turbulent flows with high accuracy. The modelling mode uses a fixed time step. The size of the time step is 0.1 seconds, which ensures the necessary accuracy and detail when modelling dynamic processes in the system. The maximum number of time steps is set to 50, with an iteration (time step) of 40. This allows to achieve an optimal balance between accuracy and speed of calculations, providing a detailed reproduction of flow behaviour.

On the experimental stand (Fig. 3), the loading characteristic of the water pump was determined, the parameters of which were used for modelling.

Figure 3. The structure of the experimental stand for determining the loading characteristics of the water pump
Note: 1 – buffer tank, 2 – pump, 3 – automatic recorder, 4 – power supply unit, 5 – water intake pipe, 6 – water supply pipe to the buffer tank, 7 – connecting adapter, 8 – pressure sensor, 9 – communication cable with the computer, 10 – computer
Source: compiled by the authors

For this purpose, an automatic recorder was created, the scheme of which is shown in Figure 4. The automatic recorder is capable of measuring voltage, current and pressure with a frequency of about 120 measurements per second, which ensures high precise removal of the loading characteristic. Such a high frequency of measurements allows obtaining detailed data necessary for further analysis of pump efficiency.

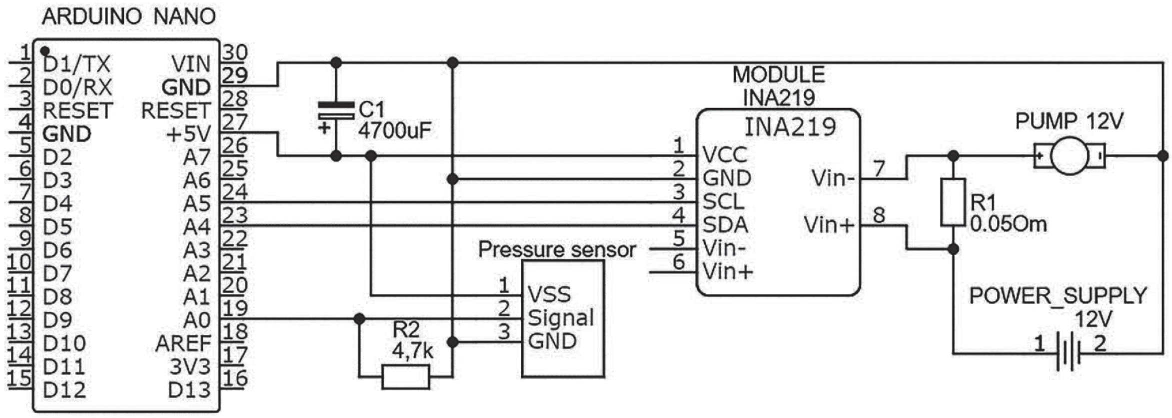


Figure 4. Scheme of the automatic recorder

Source: compiled by the authors

When studying the configuration of the liquid spray cone on a real model of the device, a pressure measurement unit in the nozzle, which is shown in Figure 5, was used.

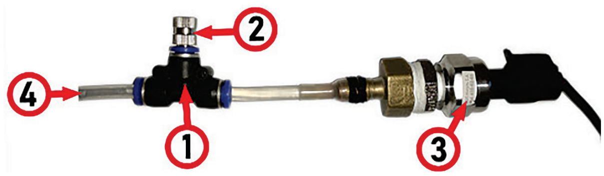


Figure 5. Pressure measurement unit in the nozzle

Note: 1 – connecting adapter, 2 – spray nozzle, 3 – pressure sensor, 4 – water supply tube from the pump

Source: compiled by the authors

To check the accuracy of the mathematical model, the nozzles were first simulated separately from the device. After confirming the adequacy of the mathematical model of hydrodynamic processes for individual nozzles, simulations were carried out on a real model of the device in the presence of at least four sensors.

To obtain data for the further improvement of the device, experiments were conducted to determine the spray spectrum for one nozzle and statistical analysis of the distribution of the amount of liquid that falls on the plane where the sensors can be located. For this experiment, a standard practice was used, which was conducted under laboratory conditions following the method described in P.A. Kluza *et al.* (2019). Tests and measurements were carried out on a corrugated table (Fig. 6) with dimensions of 900×970 mm, which is assembled from 18 plastic corners of 40×40 mm, the distance between the vertexes is 56.6 mm.

The nozzle is located at a height of 0.5 m, and the spray angle in relation to the surface of the corrugated table was approximately 110°. The sprayed liquid fell on the table and flowed down the grooves into 17 plastic containers of the same volume, located consecutively at the same distance from each other. Each measurement using electronic scales DJ-660-0.01 (SPE "Technovagy", Ukraine) with high accuracy determines the amount of liquid accumulated in individual containers. Measurements were repeated three times for experiments with different nozzle diameters. Spraying time was 2 minutes for nozzles with diameters of 0.2 and 0.4 mm and 3 minutes for nozzles with diameters of 0.6 and 0.8 mm.

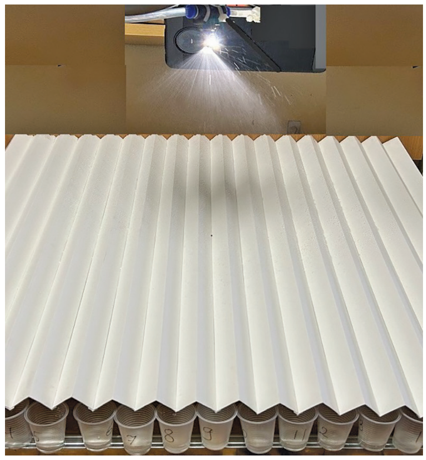


Figure 6. Determination of the spraying spectrum for one nozzle

Source: compiled by the authors

RESULTS AND DISCUSSION

Figure 7 shows the loading characteristic of the pump, which was recorded using an automatic recorder. This made it possible to obtain detailed data on the behaviour of the pump under various loading conditions, which is critically important for accurate mathematical modelling of hydrodynamic processes in the system. The loading characteristic demonstrates that the pump is able to create the necessary pressure, providing a stable and realistic flow of liquid in the system, which confirms its efficiency and compliance with the technical requirements of the simulation.

Figure 8 shows the transient process of setting the pressure in nozzles of different diameters, and Table 1 shows the data of water flow parameters when the nozzle diameter changes.

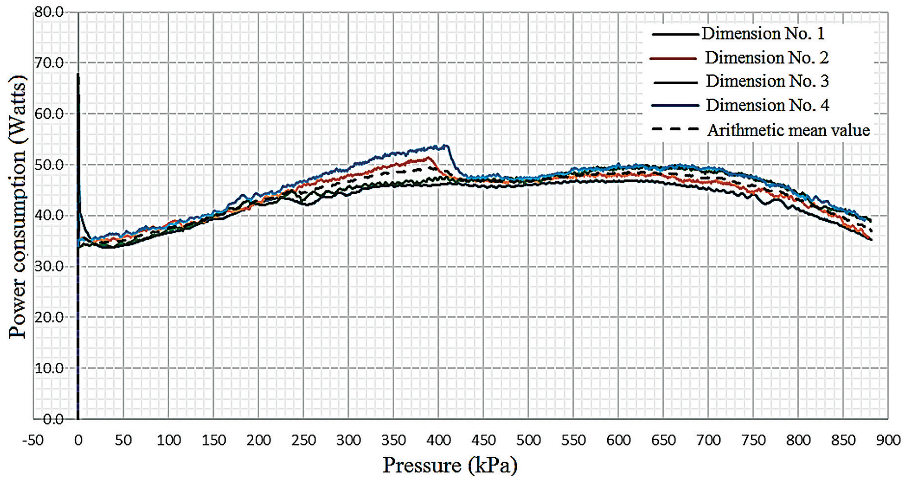


Figure 7. Loading characteristics of the pump

Source: compiled by the authors

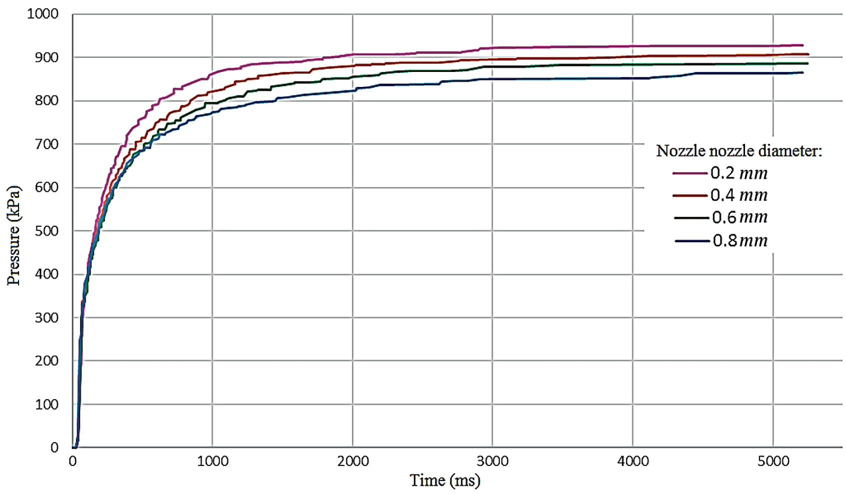


Figure 8. Water pressure in nozzle holes of different diameters

Source: compiled by the authors

Table 1. Dependency of water flow parameters when the nozzle diameter changes

Parameter name	Nozzle diameter, mm			
	0.2	0.4	0.6	0.8
Maximum recorded pressure, kPa	927	907	886	864
Water consumption, hr/min	33.3	89.0	124.0	210.3
The flow rate is determined based on the measured water consumption and the cross-sectional surface of the nozzle, m/s	17.6	13.15	7.72	8.31
The flow rate in the water supply pipeline with a diameter of 4 mm, determined on the basis of the measured water consumption, m/s	0.0441	0.118	0.1644	0.2789
The flow rate in the nozzle simulated at the real flow rate in the pipeline with a diameter of 4 mm, m/s	18.94	14.58	8.58	8.41
Deviation of experimental and simulation results, %	7.0	9.8	10.0	9.9

Source: compiled by the authors

Verification of the accuracy of the mathematical model for the nozzles separately from the device allowed to analyse their free behaviour, providing accurate conditions for checking and establishing the model's compliance with real physical parameters. Figure 9 presents the results of modelling and experimental studies of the distribution of jets and the shape of the liquid spray cone depending on the change in the diameters of the spray nozzles. The simulation results demonstrate the high accuracy and reliability of the mathematical model, which is an important prerequisite for further complex modelling and optimisation of the entire device as a whole.

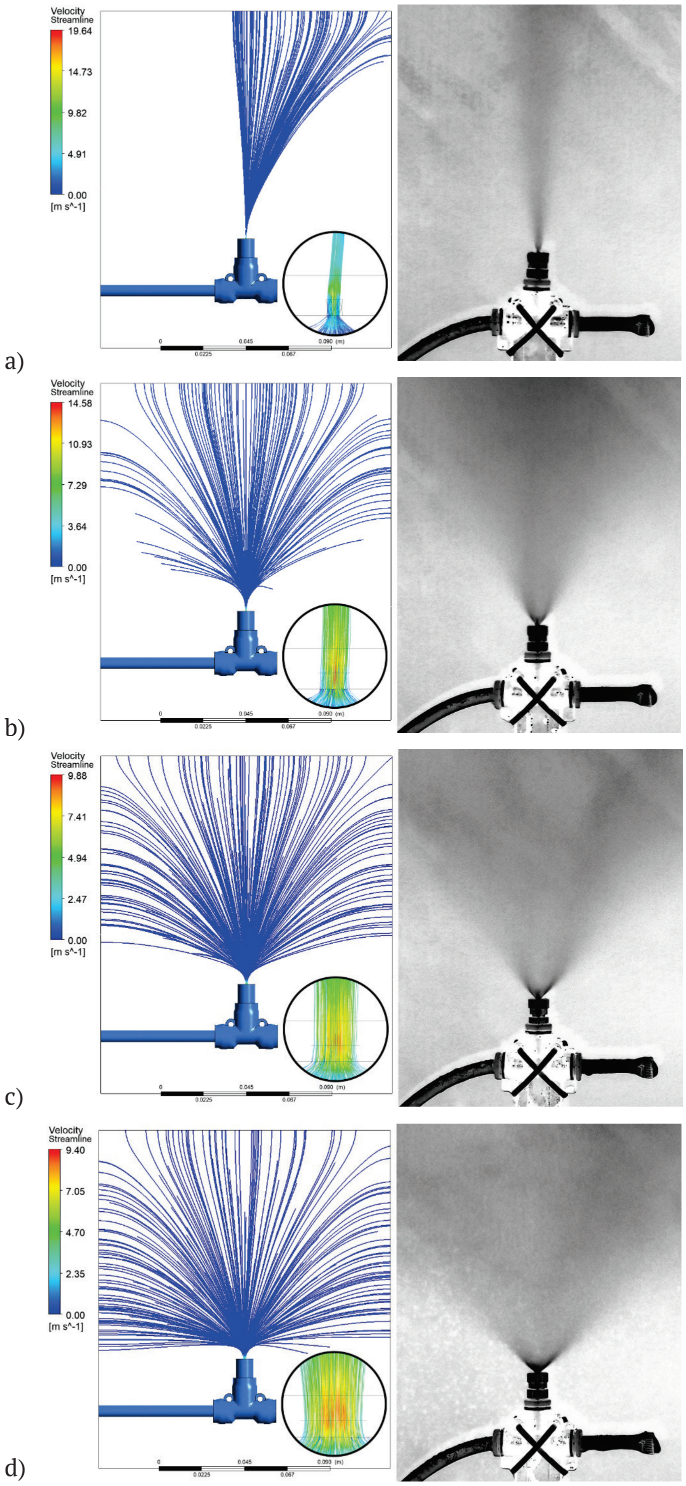


Figure 9. The distribution of jets and the shape of the liquid spray cone depending on the change in the nozzle diameters

Note: the nozzle diameter a) 0.2 mm, b) 0.4 mm, c) 0.6 mm, d) 0.8 mm; on the left – simulation results; on the right – the results of the experiment

Source: compiled by the authors

After confirming the adequacy of the mathematical model of hydrodynamic processes for individual nozzles, simulations were carried out on a real model of the device (Fig. 2) in the presence of at least four sensors. The simulation results (Fig. 10) include the analysis of the flow lines (Streamline), which gives an idea of the shape of the liquid spray cone in the presence of two nozzles in the device for automatic control of the parameters of the anaerobic fermentation process in biogas reactors. As can be seen from the results, the optimal diameter of the nozzle is 0.8 mm. The flow lines indicate that the sensors are effectively washed, ensuring their cleaning from dirt. To clarify the results, nozzles with diameters of 0.7 mm and 0.9 mm were additionally modelled. However, as in the previous case, a nozzle with a diameter of 0.8 mm turned out to be optimal. It provides an optimal wash cone that effectively covers all sensors, ensuring their cleaning.

Mathematical modelling of the 3D model in the Ansys Fluent environment, as in the work by C. Yue *et al.* (2013), made it possible to carry out a detailed analysis of the effect of spray nozzle diameters on the liquid spray cone when washing the sensors in the device.

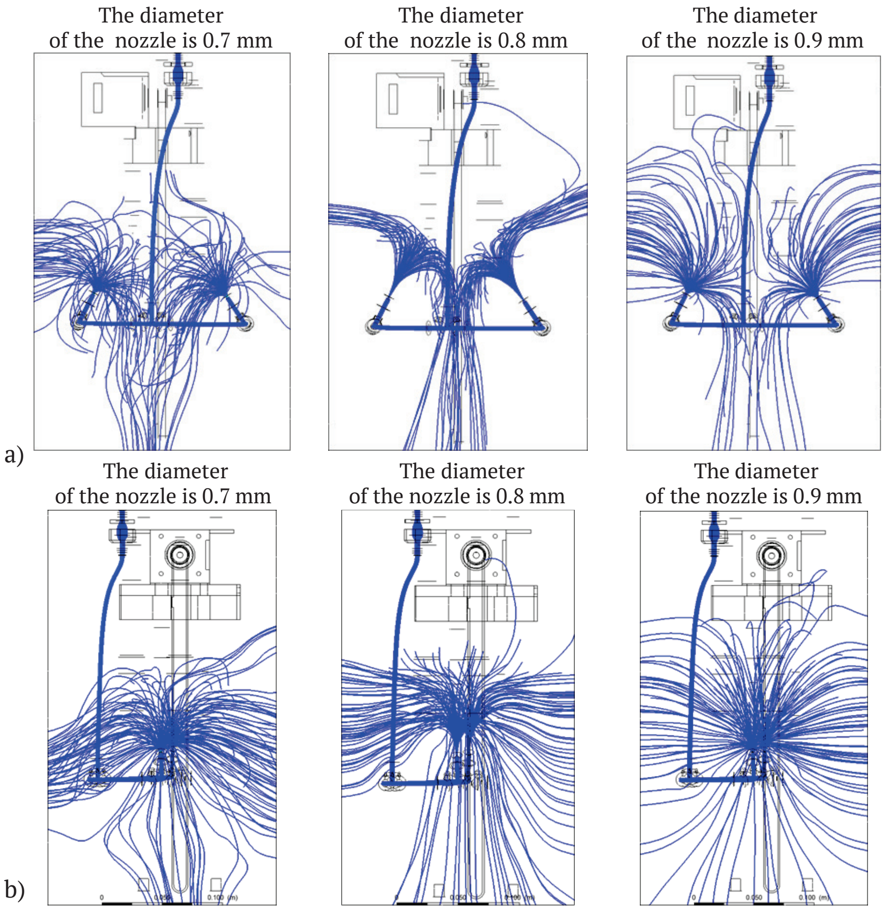


Figure 10. Distribution of flow lines and liquid spray cone in the device for automatic control of anaerobic fermentation process parameters in biogas reactors
Note: a) YZ plane; b) XZ plane
Source: compiled by the authors

Comparison of the simulation results with the experimental values showed that the error was less than 10% despite the threefold increase in leakage pressure compared to the work by C. Yue *et al.* (2013) and the presence of a spectrum of turbulent flows. This indicates the effectiveness of the analysis of hydrodynamic characteristics and the flow field by computational fluid dynamics (CFD) methods.

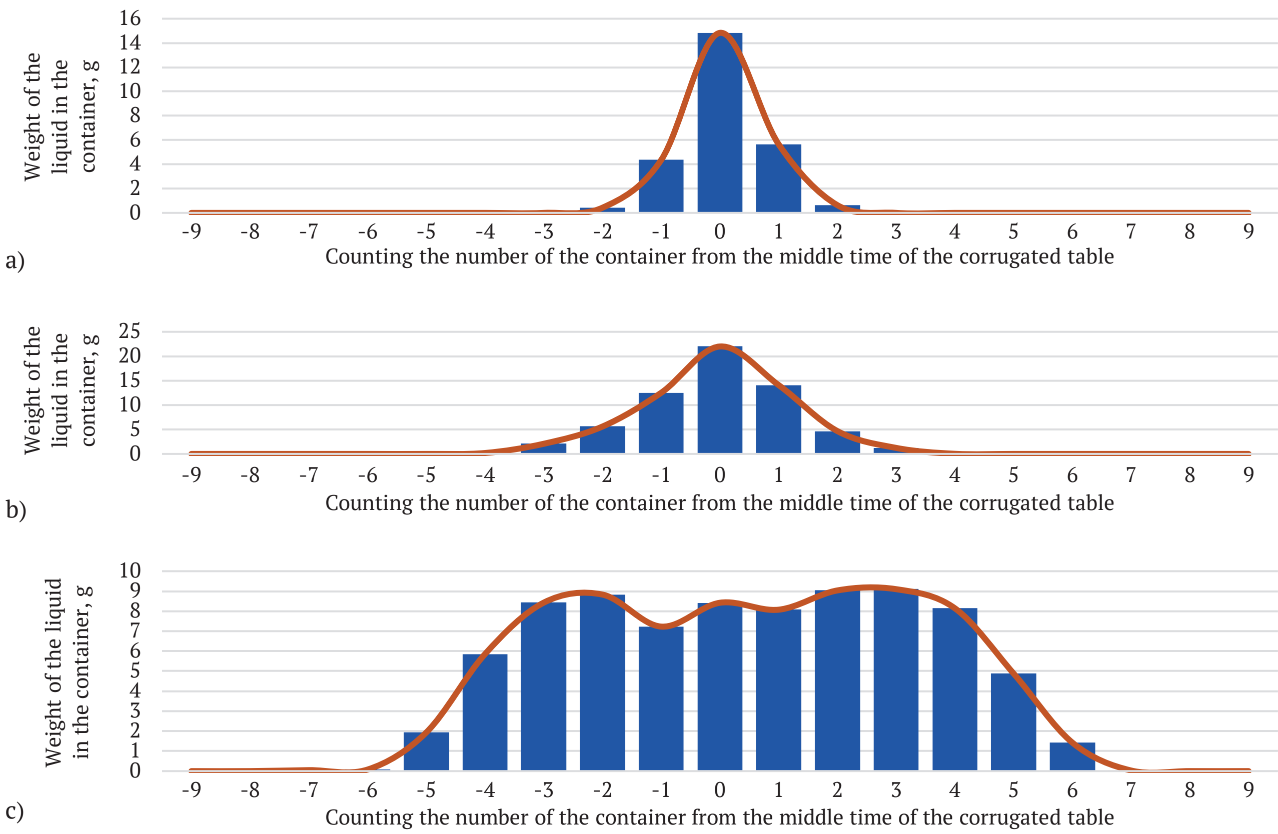
In current study, the k-omega model of liquid flow was used, where the equations describe the evolution of kinetic energy and turbulence frequency, which provided a smaller error in the analysis of hydrodynamic characteristics compared to experimental data than the standard k-epsilon turbulence model by M. Juraeva *et al.* (2019). Such a model can be used in both mesophilic and thermophilic regimes compared to Frank's model to predict water-air flow at room temperature in bubble columns in which bubble merging and collapse are not obvious M. Gong *et al.* (2016).

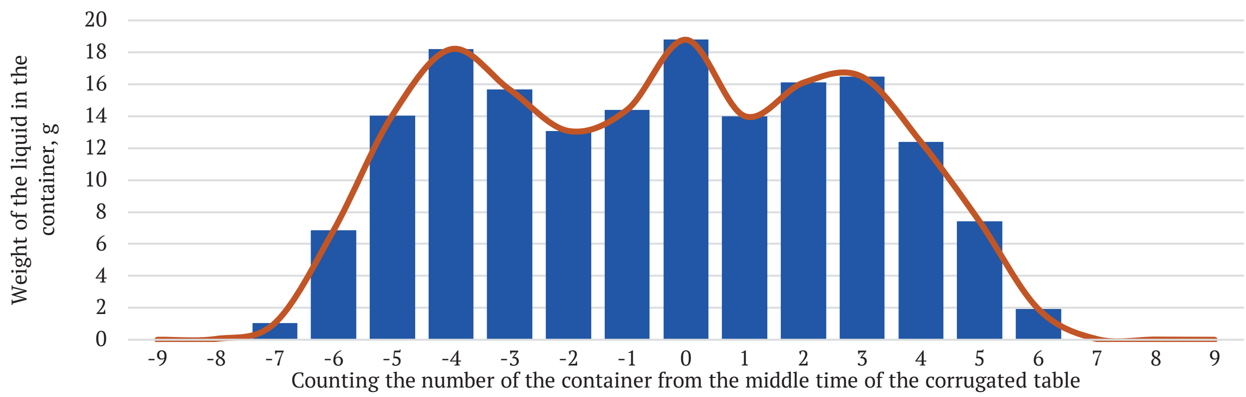
It should be emphasised that the evenness and predictability of the liquid spray distribution are the main characteristics of the cleaning system of the device for automatic control of the parameters of the anaerobic fermentation process in biogas reactors. According to F. Griesang *et al*. (2022), the evenness of liquid distribution across the entire width of the sprayer is influenced, among other factors, by liquid pressure and nozzle type, liquid jet angle, and its symmetry. In current research, it was experimentally determined that even a symmetrical arrangement of two nozzles in the YZ plane provides an optimal washing cone that effectively covers all sensors, ensuring their cleaning.

The results showed that spray pressure and hole diameter significantly affect the spray concentration, and efficiency, including energy efficiency, depends on the purpose of the spray device. For example, in the work by Z. Liu *et al.* (2019), when studying the spray properties and the corresponding rules for changing nozzles with different hole diameters (1.6 mm, 2.0 mm, 2.4 mm) at different spray pressures from 1 MPa to 5.5 MPa, a change in droplet size was observed as the pressure increased, and the evenness of the droplet size distribution reached the optimal level at a pressure of 5 MPa and a hole diameter of 2.4 mm. In our case (Table 1), nozzles of a much smaller diameter (0.2-0.9 mm) were used. When the nozzle diameter increased from 0.2 mm to 0.4 mm, i.e. 4 times, the maximum recorded pressure decreased by no more than 1.1 times, and it amounted 0.864 MPa for the nozzle diameter of 0.8 mm and was sufficient to ensure required degree of evenness of jet distribution in the target zone of sensor washing. A feature of the use of nozzles with a nozzle diameter of 0.6 mm and 0.8 mm is the presence of a spectrum of turbulent flows along chaotic trajectories with mixing in the transverse direction. At the same time, the expansion of the spray cone is observed (Fig. 9).

In general, to evaluate the selection of the number and location of the nozzles for a particular model, it is possible to use the K-factor, known as the discharge coefficient, obtained from the basic energy equations, which connects the flow rate and static pressure. But, as R.P. Fleming (2016) and P. Croce *et al.* (2020) noted, such a coefficient should be determined experimentally.

Figure 11 shows the results of experimental studies to determine the distribution of the amount of liquid that falls on the plane of the corrugated table. The jet of sprayed liquid has the shape of a cone with characteristics that depend on the design of the nozzle, and the distribution of the amount of liquid that falls on the plane of the corrugated table depends on the diameter of the nozzle. At diameters of 0.2-0.4 mm, a normal distribution was found, whereas at diameters of 0.6-0.8 mm – a liquid distribution that more closely corresponds to the distribution of the spray (Kluza *et al.*, 2019; Cieniawska *et al.*, 2023). It is important to note that with nozzle diameters of 0.6-0.8 mm, the effect of distribution of the liquid amount from one nozzle is the same as when two or three nozzles are located next to each other. At the same time, the variation coefficient, which is the percentage ratio of the mean square deviation to the mean arithmetic value of the characteristic, did not exceed the value of 11.5% for nozzles with a diameter of 0.6-0.8 mm. Accordingly, this confirms the homogeneity of the jet set and the acceptability of such spraying for cleaning the sensors.





d)

Figure 11. Distribution of the amount of liquid that falls on the plane of the corrugated table

Note: a – Nozzle with a diameter of 0.2 mm; b – Nozzle with a diameter of 0.4 mm; c – Nozzle with a diameter of 0.6 mm; d – Nozzle with a diameter of 0.8 mm

Source: compiled by the authors

It should be emphasised the complexity of the behaviour of two-phase flow and the problems of accurate prediction of flow regimes. In this regard, it will be appropriate to use experimental techniques and non-invasive methods, such as gamma tomography, X-ray methods, capacitive probes and image analysis, to study two-phase flow models.

CONCLUSIONS

The results of the research, carried out using an automated device for controlling the parameters of anaerobic digestion in biogas reactors, highlight the crucial role of optimising structural parameters to ensure effective cleaning of sensors from contamination. A detailed analysis of the effect of nozzle diameters on the shape and area of the spray cone for sensor cleaning, carried out in the Ansys Fluent environment, demonstrated the effectiveness of the developed mathematical model of the liquid spraying process. Comparison of the simulation results with the experimental values showed that the error was less than 10% despite the increase in leakage pressure and the presence of a spectrum of turbulent flows. This indicates the effectiveness of the analysis of hydrodynamic characteristics and the flow field by computational fluid dynamics (CFD) methods.

The model allowed the evaluation of the influence of the nozzle diameter on the uniformity and efficiency of the sensor cleaning, showing that the optimum nozzle diameter is 0.8 mm. This nozzle provides the best balance between cleaning efficiency and cone shape stability. Additional modelling of nozzle diameters of 0.7 mm and 0.9 mm confirmed that deviations from the selected diameter lead to reduced cleaning efficiency or cone shape instability. A feature of the use of nozzles with a nozzle diameter of 0.6 mm and 0.8 mm is the presence of a spectrum of turbulent flows along chaotic trajectories with mixing in the transverse direction. At the same time, the expansion of the spray cone is observed. The flow lines simulated in Ansys Fluent showed that the distribution of liquid effectively covered the entire sensor plane, ensuring thorough cleaning of the sensors and uninterrupted operation of the device. This demonstrates the feasibility of the chosen design and its accuracy in real operating conditions.

The chosen design parameters improve the performance of the device, reduce measurement errors and increase the reliability of the system. The uniform distribution of the liquid flow, achieved by optimally selected design parameters, significantly reduces the probability of incomplete sensor cleaning and contributes to the operational stability of the device.

The results obtained are of practical significance and serve as a basis for further analysis and improvement of the devices, taking into account variations in the number of sensors, their arrangement schemes and spraying parameters, in order to develop advanced models of automated control systems for similar technological processes.

ACKNOWLEDGEMENTS

With the support of the National Research Foundation of Ukraine competition “Science for the Recovery of Ukraine in the War and Post-war Periods” grant No. 202/0129.

FUNDING

None.

CONFLICT OF INTEREST

None.

REFERENCES

[1] Aworanti, O.A., Ajani, A.O., Agbede, O.O., Agarry, S.E., Ogunkunle, O., Laseinde, O.T., Kalam, M.A., & Fattah, I.M. (2023). Enhancing and upgrading biogas and biomethane production in anaerobic digestion: A comprehensive review. *Frontiers in Energy Research*, 11, article number 1170133. doi: 10.3389/fenrg.2023.1170133.

[2] Cerón-Vivas, A., Cáceres, K.T., Rincón, A., & Cajigas, Á.A. (2019). Influence of pH and the C/N ratio on the biogas production of wastewater. *Revista Facultad de Ingeniería Universidad de Antioquia*, 92, 70-79.

[3] Cieniawska, B., Parafiniuk, S., Kluza, P.A., & Otachel, Z. (2023). Matching the liquid atomization model to experimental data obtained from selected nozzles. *Applied Sciences*, 13(7), article number 4433. doi: 10.3390/app13074433.

[4] Croce, P., Beyler, C., Dubay, C., Johnson, P., & McNamee, M. (2020). Fast-response sprinkler technology: Hsiang-Cheng Kung, Gunnar Heskestad, Robert Bill, Roger Allard: The 2019 Phillip J. DiNenno Prize. *Fire Technology*, 56, 1981-2001. doi: 10.1007/s10694-020-00961-7.

[5] Enache, M.A., Campeanu, A., Vlad, I., & Enache, S. (2012). Study of dynamic regimes for low power direct current motors. *2012 13th international conference on optimization of electrical and electronic equipment (OPTIM)* (pp. 692-697). Brasov:IEEE. doi: 10.1109/OPTIM.2012.6231828.

[6] Fleming, R.P. (2016). Automatic sprinkler system calculations. In *SFPE handbook of fire protection engineering* (pp. 1423-1449). New York: Springer. doi: 10.1007/978-1-4939-2565-0_42.

[7] Gong, M., Dai, F., Li, C., Li, Z., & Zhang, S. (2016). Study on the wall lubrication force for water-air in multi-scale bubble columns and experimental validation. *Journal of Chemical Engineering of Japan,* 49(5), 408-416. doi: 10.1252/jcej.15we213.

[8] Griesang, F., Spadoni, A.B.D., Ferreira, P.H.U., & da Costa Ferreira, M. (2022). Effect of working pressure and spacing of nozzles on the quality of spraying distribution. *Crop Protection*, 151, article number 105818. doi: 10.1016/j.cropro.2021.105818.

[9] Ibeaja, A.C., Nwaogazie, I.L., & Udeh, N.U. (2023). Temperature effect on biogas production from co-digestion of food waste, potash and cow dung. *Journal of Engineering Research and Reports,* 24(6), 21-28. doi: 10.9734/jerr/2023/v24i6821.

[10] Juraeva, M., Song, D.J., & Kang, D.J. (2019). Computational optimization approach to design a water-jet nozzle for a water-jet loom using the design of experiment method. *Journal of Mechanical Science and Technology*, 33(2), 631-637. doi: 10.1007/s12206-019-0118-3.

[11] Kishan, R., Singh, D., & Sharma, A.K. (2020). CFD Analysis of heat exchanger models design using ANSYS fluent. *International Journal of Mechanical Engineering and Technology*, 11(2), 1-9.

[12] Kluza, P.A., Kuna-Broniowska, I., & Parafiniuk, S. (2019). Modeling and prediction of the uniformity of spray liquid coverage from flat fan spray nozzles. *Sustainability*, 11(23), article number 6716. doi: 10.3390/su11236716.

[13] Kovalenko, M.A., & Samoilenko, O.O. (2019). Development of an experimental stand for the study of low-power DC machines with electromagnetic excitation. *Energy: Economy, Technologies, Ecology*, 3, 87-93. doi: 10.20535/1813-5420.3.2019.196392.

[14] Li, W., Guo, J., Cheng, H., Wang, W., & Dong, R. (2017). Two-phase anaerobic digestion of municipal solid wastes enhanced by hydrothermal pretreatment: Viability, performance and microbial community evaluation. *Applied Energy*, 189, 613-622. doi: 10.1016/j.apenergy.2016.12.101.

[15] Lima, D., Appleby, G., & Li, L. (2023). A scoping review of options for increasing biogas production from sewage sludge: Challenges and opportunities for enhancing energy self-sufficiency in wastewater treatment plants. *Energies*, 16(5), article number 2369. doi: 10.3390/en16052369.

[16] Liu, Z., Nie, W., Peng, H., Yang, S., Chen, D., & Liu, Q. (2019). The effects of the spraying pressure and nozzle orifice diameter on the atomizing rules and dust suppression performances of an external spraying system in a fully-mechanized excavation face. *Powder Technology*, 350, 62-80. doi: 10.1016/j.powtec.2019.03.029.

[17] Monachan, B., Thomas, R.J., Steaphen, D., Skaria, M., & Shafi, K.A. (2019). Simulation of stratified two-phase flow regime using air-water model in ANSYS Fluent®. *Journal of Physics: Conference Series,* 1355, article number 012014. doi: 10.1088/1742-6596/1355/1/012014.

[18] Musiu, E.M., Qi, L., & Wu, Y. (2019). Evaluation of droplets size distribution and velocity pattern using Computational Fluid Dynamics modelling. *Computers and Electronics in Agriculture*, 164, article number 104886. doi: 10.1016/j.compag.2019.104886.

[19] Renaudo, C.A., Yommi, A., Slaboch, G., Bucalá, V., & Bertin, D.E. (2022). Prediction of droplet size distributions from a pre-orifice nozzle using the Maximum Entropy Principle. *Chemical Engineering Research and Design*, 185, 198-209. doi: 10.1016/j.cherd.2022.07.010.

[20] Sadafi, M.H., Jahn, I., & Hooman, K. (2016). Nozzle arrangement effect on cooling performance of saline water spray cooling. *Applied Thermal Engineering*, 105, 1061-1066 doi: 10.1016/j.applthermaleng.2016.01.078.

[21] Shao, X., Su, X., Tian, S., & Cai, J. (2021). The effect of biogas fermentation assisted by simple solar greenhouse. *E3S Web of Conferences*, 246, article number 03001. doi: 10.1051/e3sconf/202124603001.

[22] Sudiartha, G.A.W., Imai, T., Mamimin, C., & Reungsang, A. (2023). Effects of temperature shifts on microbial communities and biogas production: an in-depth comparison. *Fermentation*, 9(7), article number 642. doi: 10.3390/fermentation9070642.

[23] Wang, J., Liang, Q., Zeng, T., Zhang, X., Fu, W., & Lan, Y. (2022). Drift potential characteristics of a flat fan nozzle: A numerical and experimental study. *Applied Sciences*, 12(12), article number 6092. doi: 10.3390/app12126092.

[24] Yue, C., Guo, S., & Li, M. (2013). ANSYS FLUENT-based modeling and hydrodynamic analysis for a spherical underwater robot. In *2013 IEEE international conference on mechatronics and automation* (pp. 1577-1581). Takamatsu: IEEE. doi: 10.1109/ICMA.2013.6618149.

Микола Заблодський

Доктор технічних наук, професор
Національний університет біоресурсів і природокористування України
03041, вул. Героїв Оборони, 15, м. Київ, Україна
https://orcid.org/0000-0001-8889-8158

Денис Бабак

Аспірант
Національний університет біоресурсів і природокористування України
03041, вул. Героїв Оборони, 15, м. Київ, Україна
https://orcid.org/0009-0000-8970-8598

Моделювання процесів розпилення рідини у пристрої для автоматичного контролю параметрів процесу анаеробного бродіння у біогазових реакторах

Анотація. Актуальність дослідження обґрунтована стрімким розвитком біогазових технологій, що потребує постійного вдосконалення методів та обладнання для підвищення ефективності і забезпечення оптимальної продуктивності процесу виробництва біогазу та біометану. Метою цього дослідження було вдосконалення та оптимізація пристроїв для автоматичного контролю параметрів процесу анаеробного бродіння в біогазових реакторах. Використані методи математичного моделювання 3D-моделі в середовищі Ansys Fluent, а також методи експериментальних досліджень по визначенню спектру розпилення форсунок, розподілення струменів та форми конуса розбризкування рідини. Запропоновано конструктивну схему пристрою для автоматичного контролю параметрів процесу анаеробного бродіння в біогазових реакторах, математичну модель процесу розпилення рідини в системі очищення датчиків, в якій застосовуються рівняння, що описують еволюцію кінетичної енергії турбулентності та частоти турбулентності. Створений автоматичний реєстратор, здатний вимірювати напругу, струм та тиск із частотою близько 120 вимірювань на секунду, що забезпечує високоточне зняття навантажувальних характеристик в системі подавання рідин. За результатами моделювання та експериментальних досліджень встановлено закономірності розподілення струменів і форми конуса розбризкування рідини в залежності від зміни діаметрів сопел розпилювачів. Результати моделювання продемонстрували високу точність і надійність математичної моделі, що є важливою передумовою для подальшого комплексного моделювання та оптимізації роботи. Це дослідження робить значний внесок до розробки більш ефективних методів і засобів автоматичного контролю параметрів процесу анаеробного бродіння в біогазових реакторах сприяючи збільшенню продуктивності процесу виробництва біогазу

Ключові слова: навантажувальна характеристика помпи; Ansys Fluent; оптимізація сопла розпилювача; фізична модель; лінії потоку рідини; ефекти турбулентності