Numerical simulation of bubble formation process from a stagnant nozzle immersed in a quiescent liquid (water) and from a nozzle located on a rotating cylinder set in a cross-flowing liquid was performed with the aid of a computational fluid dynamics (CFD) code. Solution of momentum balance equations was coupled with VOF algorithm for tracking the gas-liquid interface in 2D and 3D domains. The results of the simulation are compared with the experimental data obtained from a high-speed video camera observation. The agreement was favourable in the case of rotating nozzle. In the case of a stationary nozzle, the VOF algorithm seems to perform best when the process is dominated by inertial forces (higher gas flow rates).

1. INTRODUCTION

The degree of gas dispersion in a gas–liquid contactor (or reactor) often controls the product yield (e.g., oxygen absorption rate in a bioreactor) and the contactor performance, especially when a gaseous reactant is consumed in a fast reaction occurring in the liquid phase [4]. The gas is usually sparged by means of a nozzle, a perforated membrane or a porous sparger. When the gas phase is introduced into the liquid flowing past a sparger, especially in downwards direction, a large attached gas cavity is often formed instead of a swarm of small bubbles. Under certain conditions in the loop reactors, the cavity grows in length and occupies almost the entire cross-sectional flow area, which can cause serious operational problems by reducing the velocity of the recirculating flow and may cause instability in the reactor. In our earlier papers [1–3], we have presented a novel method of gas sparging from a rotating...
cylinder into a cross-flowing liquid. Such a method of gas dispersion prevents from formation of large attached gas cavities at the discharging orifice. Since dispersing gas in form of fine bubbles is a common way in achieving high interfacial area in the gas–liquid-contacting devices, elimination of gas slugs is of importance. Experimental and theoretical results regarding critical rotational speed necessary to remove the attached cavity, bubble formation process and size distribution of the produced bubbles in a low viscosity system (air–water) have been presented in our earlier paper [3]. The present paper deals with the numerical simulation of bubble formation process at the surface of a rotating cylinder in a cross-flowing liquid. In order to verify the modelling approach first, we have also performed the simulation of bubble formation process at a stationary nozzle in a stagnant liquid.

Bubble formation, even in a stagnant liquid, is a fairly complicated process influenced by many factors including properties of gas and liquid phases (density, viscosity, surface tension), orifice geometry and orientation, gas flow rate, liquid velocity in the nozzle neighbourhood. The gas flow rate through the nozzle is determined by the pressure difference between the growing bubble and the gas chamber located on the other side of the nozzle. In the case of a small flow resistance in the orifice, changing pressure in the forming bubble results in gas flow variability. The gas chamber pressure also alters unless volume of the chamber is large enough to damp the pressure fluctuations. This situation is known in the literature as the “constant pressure” regime. On the other extreme, if the pressure drop in the nozzle is larger than pressure fluctuations in the forming bubble, the chamber volume no longer exerts any influence on the process of bubble formation. It can now be treated as that corresponding to the “constant flow” regime.

Bubble formation process was extensively studied over the last few decades, both experimentally and theoretically (see e.g., the review paper [7]). The first models proposed by Davidson and Schüler [7, 8] were approximate (but simple) and assumed one- or two-stage growth and detachment of a spherical bubble. The more complicated and realistic models assumed neither spherical bubble shape nor imposed any arbitrary criteria for its detachment (see e.g., [9–13]). A few studies focused on modelling bubble formation in liquid cross-flow conditions (see e.g., [18, 19]).

Application of the CFD techniques based on numerical solution of discretised Navier–Stokes equations, coupled with some method of tracking the gas–liquid interface during bubble formation and its rise, is relatively new. One of the possible approaches is based on a deformable grid, which aligns with the gas liquid interface and follows its movement during the simulation [14]. The other, like the marker and cell (MAC) method applied by Welch et al. [15], is based on fixed grids but introduces artificial marker particles used for tracking the liquid phase during simulation. An alternative technique based on a fixed grid is the volume of fluid (VOF) method developed by Hirt and Nichols [16]. This method is actually referred to in the literature as a front capturing rather than a front tracking technique. According to this method, the continuity equation (1) for the volume fraction of the gas phase is solved during the simulation procedure additionally to momentum conservation equations.
Fluid properties in every cell are calculated additively with respect to the value of this quantity as the average weighted with the volume fraction. The volume fraction of the gaseous phase is treated as a marker, which designates a given computational cell filled either with liquid ($\alpha_g = 0$) or with gas phase ($\alpha_g = 1$) or crossed by the interface ($0 < \alpha_g < 1$). A number of algorithms were developed to reconstruct the interface within the computational domain based on the coordinates of cells in which $0 < \alpha_g < 1$ and to adjust the values of $\alpha$ in the remaining cells in order to reduce numerical diffusion and ensure total gas volume conservation (see e.g., [21]).

The method is known to be computationally more efficient than the MAC method, especially in 3D domains, where it requires tracking a large number of tracer particles. The VOF method also inherently enables simulation of fragmentation and/or coalescence of the phases. Therefore, we have chosen this approach to simulate the bubble formation process. The major aim of the computations was to check applicability of the VOF method for predicting the bubble size formed in a stagnant liquid and under cross-flow conditions.

### 2. MODELLING APPROACH

#### 2.1. SIMULATION SET-UP

The simulation was performed with the aid of FLUENT 6.1 CFD code, using the finite volume approach [20] for discretisation of the mass, momentum and void fraction balance equations on unstructured computational meshes. Second order upwinding scheme was used for calculation of the convective terms. Both phases were treated as incompressible and the SIMPLE method was used for pressure–velocity coupling [20]. The position of the interface between the gas and the liquid was calculated using the volume of fluid model (VOF) with a piecewise-linear geometric reconstruction of the interface (PLIC VOF) [21].

<table>
<thead>
<tr>
<th>Property</th>
<th>Liquid (water)</th>
<th>Gas (air)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density [kg/m³]</td>
<td>998.2</td>
<td>1.225</td>
</tr>
<tr>
<td>Viscosity [Pa·s]</td>
<td>$1.003 \times 10^{-3}$</td>
<td>$1.7894 \times 10^{-3}$</td>
</tr>
<tr>
<td>Surface tension [N/m]</td>
<td>0.073</td>
<td>–</td>
</tr>
</tbody>
</table>

Time dependence was resolved in an implicit marching scheme by iterating in time with a constant time step. Contrary to the momentum equations, Eq. (1) was in-
tegrated using an explicit time marching scheme. In order to ensure numerical stability, the time step was internally subdivided to meet the Courant number criterion.

Surface tension was accounted for in the simulation with the aid of continuum surface force (CSF) model by Brackbill et al. [22]. According to the model, the force resulting from surface tension was included as an additional source term in the momentum equation being dependent on the local curvature of the interface.

The properties of the phases used in the simulation are presented in Table 1.

### 3.2. STAGNANT LIQUID

In order to check suitability of the VOF methodology to realistic simulating of the bubble formation process we have initially modelled bubble formation (in the constant gas flow regime) from a stationary nozzle submerged in a stagnant liquid. Two modelling approaches were undertaken:

- axial symmetry was assumed and the problem was solved in the 2D domain,
- no symmetry was assumed which required complete solution in the 3D domain.

The details of the two simulation approaches were reported in our earlier paper [2]. Here, we will briefly summarize the 2D approach in order to discuss the results and compare them with the experiments.

![Fig. 1. Schematic of the 2D computational domain and the adopted grid corresponding to the interface at a sample time instant. Stationary nozzle case](image)

The simulation illustrated the process of bubble formation in a stagnant liquid from a 0.3 mm diameter orifice held horizontally. The gas was supplied to the orifice via a 4 mm long capillary (nozzle) of the same diameter, at a constant rate (the constant flow conditions). The axisymmetric computational domain (2D) is shown in Fig. 1. It extends 5 mm in the radial direction and 24 mm in the axial direction (including the 4 mm long nozzle, 0.3 mm in diameter). The “symmetry” boundary condition was set at the axis (vanishing momentum, mass and void fraction fluxes in radial direction). The constant pressure of 101 325 Pa was set as a boundary condition at the outlet from the domain. In order to stabilize the bubble attachment point to the edge of
the orifice, zero static contact angle of liquid at the bottom wall (i.e., good wettability of the orifice plate and poor wettability of the nozzle wall) was set as the boundary condition for determination of the interface shape at the attachment point. The mass flow rate of the gas phase was set as the boundary condition at the inlet to the nozzle. Three different gas mass flow rates were investigated: $2 \times 10^{-8}$ kg/s, $1 \times 10^{-7}$ kg/s, $1 \times 10^{-6}$ kg/s (corresponding to volumetric flow rates: 0.0163 ml/s, 0.0816 ml/s and 0.816 ml/s, respectively).

The first grid used in the simulation consisted of approx. 10,000 quadrilaterals. The grid was refined during the solution procedure for better resolution of the interface between the fluids. A normalized value of the void fraction gradient was used as the grid adoption criterion. The second tested grid was finer and consisted of approx. 18,000 quadrilaterals. This grid was not refined during solution.

The solution was obtained in an unsteady manner with the time step of $1 \times 10^{-6}$ s in the case of two smaller gas flow rates and $1 \times 10^{-7}$ s in the case of the highest gas flow rate.

### 3.3. ROTATING SPARGER

In this case, the bubble formation process occurs during gas outflow from a nozzle located at the surface of a rotating cylinder immersed in a stream of liquid flowing perpendicularly to its axis. The sizes of the produced bubbles depend on the ratio of the cylinder peripheral speed and liquid cross-flow velocity ($\alpha$). As it was demonstrated elsewhere [1, 3] bubble size distribution in the generated gas dispersion is bi-modal at the lower rotational rates ($\alpha < 4$) and mono-modal at the higher ones ($\alpha >> 4$).

![3D computational grid. Rotating nozzle case](image)
Preliminary computations were conducted assuming a two-dimensional (2D) flow past the cylinder. The results of these simulation significantly deviated from the experimental observations (the bubbles did not form and the entire circumference of the cylinder was surrounded by gas). Further computations were thus performed in a 3D domain. Computational grid is schematically shown in Fig. 2. The sizes of the computational domain were as follows: width – 10 cm, length – 15 cm, and thickness – 3 mm. The cylinder of 20 mm in diameter was located at a distance of 5 cm from the channel inlet. The nozzle diameter fixed at the cylinder surface was 0.3 mm. Two sides of the domain (the cutting planes) formed the planes of symmetry for the fluid flow separated from each other by 3 mm. One side of the domain crossed the nozzle along its axis. Application of the VOF model required the appropriate grid densification in the nozzle region (Fig. 3). The total number of the grid elements equaled to 120,000.

![Fig. 3. Computational grid (nozzle region)](image)

A region around the cylinder was selected, in which the grid rotating together with the cylinder formed a rotating system of coordinates. The fluid flow within the remaining part of the domain was computed using a fixed system of coordinates. The following boundary conditions were assumed:

- nozzle outlet: air mass flow rate: $1 \cdot 10^{-7}$ and $1 \cdot 10^{-6}$ kg/s (depending on the nozzle position),
- channel inlet: liquid velocity $u_L = 0.35$ m/s,
- channel outlet: fixed value of pressure,
- surface of the cylinder: fixed value of the rotational frequency – 700 rpm.

The computations were performed taking the time step equal to $1 \cdot 10^{-5}$ s.

Since the gas flow rate through the nozzle is not constant in this case, an appropriate boundary condition at the nozzle could be set by specifying, at each time step, the value of pressure that results from the mass balance of gas in the gas chamber (6 in Fig. 6).
Assuming a constant gas supply into the gas chamber, the pressure in the chamber fluctuates in response to changes in the gas outflow rate. The process can be treated as adiabatic (cf. e.g., [18]) and the pressure fluctuations are described by:

$$\frac{d \ln(p_c)}{dt} = \kappa \frac{(Q_g - Q_{GN})}{V_c}$$ (2)

However, such an approach would require a very long computation time to reach a semi-steady conditions, which will be established after many revolutions of the cylinder. For the sake of comparison with the experimental results pertaining to the semi-steady conditions, a simplified approach was taken. Initially, the computations were performed assuming a single phase flow (liquid) until a uniform velocity profile in the neighbourhood of the cylinder was established (see Fig. 14). Then a non-zero gas stream discharging from the nozzle was set up. Two values of the gas flow rate corresponding to those determined from experiments were used in the simulations. The lower value (1·10⁻⁷ kg/s) corresponded to the position of the nozzle for which the difference between vertical velocity components of the cylinder surface and the liquid in the channel is the highest, while the higher one, equal to 1·10⁻⁶ kg/s – to the nozzle position on the opposite side of the cylinder.

4. EXPERIMENTAL

In order to verify the ability of VOF methodology to realistic simulating of the bubble formation process, a series of experiments were performed to visualize the bubble formation process.

Stagnant liquid. The schematic of the experimental set-up used for visualization of bubble formation process from a nozzle submerged in a stagnant liquid, is shown in Fig. 4.

![Fig. 4. Schematic of the experimental rig (stagnant liquid)](image)
The air bubbles were formed at the outlet of the capillary nozzle (0.3 mm ID, 100 mm long) placed in a rectangular glass container (8x5x12 cm³) filled with distilled water. The gas flow rate was varied within the range: $10^{-7}$–$10^{-6}$ kg/s ($0.0816$–$0.816$ ml/s). The images of the forming bubbles were recorded by a high-speed digital video camera (Redlake Motionscope 8000 fps) with time resolution of 2000 frames per second.

**Rotating sparger.** Bubble formation from an orifice located on a rotating cylinder surface in a liquid cross-flow was visualised in a transparent rectangular duct with the aid of the high-speed digital camera. Bubble formation was recorded at 1000 fps (240x210 pixel resolution) and 2000 fps (160x140 pixel resolution).

![Fig. 5. Schematic of the experimental rig (rotating nozzle)](image)

Rys. 5. Schemat stanowiska doświadczalnego do wizualizacji procesu formowania pęcherzy podczas przepływu poprzecznego (dysza wirująca)

The experimental rig is shown in Fig. 5. The liquid circulation loop (approx. 3 m high) consisted of high-pressure PVC piping (DN50 and DN100). The vertical glass duct of a rectangular cross-section (52x92 mm²) was connected to a calming section (DN100) with a convergent nozzle designed to avoid flow separation and to produce
Numerical simulation of bubble formation in liquids

A uniform velocity profile. Liquid circulation was accomplished by the action of a centrifugal pump controlled by a PID controller with an input signal from the differential manometer (DP) connected to the orifice plate flow meter. The pump allowed the maximum water flow rate of approx. 20 m³/h, corresponding to 1.1 m/s superficial liquid velocity in the duct. The gas sparger was supplied through a mass flow controller from the oil-free air compressor. The bubbly mixture created below the sparger flew into the reservoir tank (0.25 m³) where the gas separated from the liquid phase.

Fig. 6. Schematic cross-section of the rotating cylindrical gas sparger in a rectangular water tunnel

Rys. 6. Schematyczny przekrój wirującego dystrybutora gazu umieszczonego w prostokątnym kanale

A schematic cross-section of the sparger is shown in Fig. 6. The sparger was mounted in a transparent rectangular glass duct (1). The orifice (3) drilled perpendicularly to the axis of the cylinder (2) was supplied with gas through a 2 mm (external diameter) pipe (4). The orifice and cylinder diameters were 0.3 mm and 20 mm, respectively. In order to avoid excessive friction, 1–2 mm gap was maintained between the cylinder ends and the duct walls. The pipe inlet and the main bearing were sealed from the surroundings by the housing (6) connected to the gas supply (7). The cylinder was driven by a 15W DC motor (5) of a variable rotational speed between 300 and 18 000 rpm. The rotational speed of the cylinder was constantly monitored with the aid of a sensor (transoptor) (8) and stabilised electronically.

5. RESULTS

5.1. STAGNANT LIQUID

Void fraction contours corresponding to the value of 0.5, which indicate the location of the gas-liquid interface, are shown in Figs. 7–9 for different time instants. Single bubbling was observed in the simulation only for the case of the smallest gas flow rate $Q_g = 0.0163$ ml/s (Fig. 7). In the case of higher flow rates, the results indi-
Fig. 7. Simulated interface during axisymmetric bubble growth (single bubbling regime, $Q_G = 0.0163$ ml/s). The numbers indicate relative time in milliseconds.

Rys. 7. Obliczony kształt powierzchni międzyfazowej w czasie formowania pęcherza osiowosymetrycznego (zakres pojedynczych pęcherzy $Q_G = 0.0163$ ml/s). Liczby oznaczają czas względny w milisekundach.

Fig. 8. Simulated interface during axis-symmetric bubble growth (incipient bubble pairing regime, $Q_G = 0.0816$ ml/s). The numbers indicate relative time in milliseconds.

Rys. 8. Obliczony kształt powierzchni międzyfazowej w czasie formowania pęcherza osiowosymetrycznego (zakres pierwotnej koalescencji dwóch pęcherzy, $Q_G = 0.0816$ ml/s). Liczby oznaczają czas względny w milisekundach.

Fig. 9. Simulated interface during axis-symmetric bubble growth (multiple incipient coalesce during, $Q_G = 0.816$ ml/s). The numbers indicate time in milliseconds elapsed after starting the gas flow through the nozzle.

Rys. 9. Obliczony kształt powierzchni międzyfazowej w czasie formowania pęcherza osiowosymetrycznego (zakres pierwotnej koalescencji kilku pęcherzy, $Q_G = 0.816$ ml/s). Liczby oznaczają czas względny w milisekundach.
Numerical simulation of bubble formation in liquids

cated coalescence of two or more primary bubbles (Figs. 8 and 9). This is in accordance with the experimental observations of Walters and Davidson [23] who observed that incipient bubble pairing occurs for:

\[ Q_g > C_g^{12}d_N^{5/2} \]  

In Eq. (2) \( C \) takes the values from 1.3 to 6.2 but Wraith [24] suggested the value of 7.44.

Equation (2) predicts the critical gas flow rate equal to 0.0363 ml/s for a 0.3 mm ID nozzle. However, we have scarcely observed incipient bubble coalescence even at the gas flow rate of 0.333 ml/s in the case of the 100 mm in length capillary nozzle tested in our experimental set-up. The discrepancy between this observation and the critical gas flow rate given by Eq. (3) may be attributed to the difference in gas nozzle lengths used in this study. Eq. (2) pertains to much shorter nozzles (or even orifices) used by other authors, where occurrence of constant flow conditions are less likely and incipient bubble coalescence is observed at the lower gas flow rates.

The pictures taken during one cycle of bubble formation at 0.167 ml/s and 0.833 ml/s are shown in Figs. 10 and 11, respectively. It can be observed that the computed bubble envelopes compare favourably with the experimental ones, especially at the higher gas flow rates. Although in this range of flow rates the axis-symmetry during bubble formation is lost and the bubbles just formed rise along a characteristic meandering trajectory. Such behaviour was perfectly confirmed by the simulation performed in the 3D domain (see [2]).

![Bubble formation in a quiescent liquid](image-url)

Fig. 10. Bubble formation in a quiescent liquid (single bubbling regime, \( Q_G = 0.167 \) ml/s).

The numbers indicate relative time in milliseconds

Rys. 10. Formowanie się pęcherzy w nieruchomej cieczy (zakres pojedynczych pęcherzy \( Q_G = 0.167 \) ml/s). Liczby oznaczają czas względny w milisekundach.
In the case of the highest simulated gas flow rate ($Q_G = 0.816 \text{ ml/s}$), gas momentum exerts significant influence on the bubble formation process. The tip of the forming bubble is clearly distorted in Fig. 9 (e.g., $t = 9.125 \text{ ms}$) by the gas jet emerging from the orifice. It is also evident from that figure that the closing bubble neck is immediately destroyed causing ejection of secondary droplets inside the bubble and initiation of a capillary wave travelling along the bubble envelope.

It can be observed that the clashing capillary wave at the top of the bubble ejects small secondary satellite bubbles from the forming bubble top (Fig. 12). Such phenomenon was also observed and reported recently by Tse at al. [25] in regard to coalescing bubbles in a bubbly dispersion.

Equivalent (spherical) diameters of the bubbles formed in the simulation are compared with the experimental ones in Fig. 13. Shown is also a theoretical line corresponding to quasi-static conditions of bubble formation ($Q_G \to 0$) when bubble de-
attachment is defined by the equilibrium between gravitational and surface tension forces only. In the case of totally hydrophilic nozzle the equivalent bubble diameter is then given by:

\[ d_{Bph} = \sqrt[3]{\frac{6 \Delta \sigma}{\rho_L - \rho_G}} \]  

\[(4)\]

Fig. 13. Spherical equivalent diameter of bubbles formed in stagnant liquid

Rys. 13. Średnica zastępcza pęcherzy uformowanych z dyszy zanurzonej w nieruchomej cieczy

Fig. 14. Velocity field of liquid in the neighbourhood of the cylinder (stationary single-phase flow; the cylinder rotates counterclockwise)
Rys. 14. Pole prędkości cieczy w sąsiedztwie wirującego walca (wyniki obliczeń dla ustalonego przepływu jednofazowego, walec wiruje w kierunku przeciwnym do ruchu wskazówek zegara)

Fig. 15. Computed shape of the interface during bubble formation. The cylinder rotates counterclockwise. The grayscale shows relative static pressure on the surface

Rys. 15. Obliczony kształt powierzchni międzyfazowej w czasie formowania pęcherza przy powierzchni walca wirującego w kierunku przeciwnym do ruchu wskazówek zegara. Odcień szarości odzwierciedla względne ciśnienie statyczne na powierzchni
The experimental results shown in the Fig 13 seem to be consistent with the theory. It is also evident from Fig. 13 that agreement of the VOF simulation with the experimental data is favourable only at the highest simulated flow rate ($Q_G = 0.816$ ml/s) when the inertial forces prevail. The discrepancy between the VOF simulation and experiment for the slow bubble formation case (lower flow rate), when the process is dominated by surface tension rather than inertial forces, may be attributed to the presence of spurious (parasitic) currents connected with the VOF methodology and the CSF model of surface tension used in the simulations (c.f. [17]). These currents cause non-physical surface oscillations (waves) that apparently close the forming bubble neck prematurely. We also believe that these oscillations were augmented by dynamic changes of the grid due to the refinement procedure. The simulation was thus rerun with the dynamic refinement procedure turned off, using a modified (finer) grid that consisted of 17,000 quadrilaterals. The resultant bubble size was greater this time (see Fig 13, grid 2) but the discrepancy was still significant.

The velocity field calculated for a single-phase flow (liquid) past the rotating cylinder is shown in Fig. 14.

The stationary single phase flow field was used as a starting point for the two-phase simulation. The calculated shapes of the gas–liquid interface for one revolution of the cylinder are shown in Fig. 15 (the figures correspond to the subsequent time instants separated by 10 ms). In the studied case, the ratio of the tangential velocity of the cylinder and the superficial liquid velocity in the channel is smaller than 4. At the cylinder side where its velocity is opposing the bulk liquid velocity in the channel, many small bubbles form (with size of about 1 mm), while on the other side of the
cylinder, one big bubble is produced instead (Figs. 15c and d). The size of this bubble exceeded the thickness of the computational domain, which resulted in the visible discontinuity of its surface. The shape of the interface and sizes of the formed bubbles are in fair agreement with the images acquired during visualisation of the process in the experimental part (cf. Fig. 16).

ACKNOWLEDGEMENTS

Financial support of the Polish Scientific Committee (KBN, grant 7 T09C 045 20) is gratefully acknowledged.

SYMBOLS – OZNACZENIA

\( D \) – cylinder diameter, m
\( d_N \) – nozzle diameter, m
\( g \) – gravitational acceleration, m/s\(^2\)
\( Q_G \) – gas flow rate, m\(^3\)/s
\( p_C \) – chamber pressure, Pa
\( V_C \) – chamber volume, m\(^3\)
\( u_L \) – superficial liquid velocity in the duct, m/s
\( u \) – velocity vector

GREEK SYMBOLS – SYMBOLE GRECKIE

\( \alpha \) – ratio of the cylinder peripheral speed and liquid cross-flow velocity.
\( \alpha_G \) – gas void fraction
\( \kappa \) – ratio of gas specific heat capacities \( C_p/C_v \)
\( \sigma \) – surface tension, N/m
\( \rho_L \) – liquid density, kg/m\(^3\)
\( \rho_G \) – gas density, kg/m\(^3\)

REFERENCES

Numerical simulation of bubble formation in liquids

303


Andrzej K. Bni, Piotr M. Machniewski, Leszek Rudniak

SYMULACJA NUMERYCZNA PROCESU FORMOWN1A PĘCHERZY
W CIECZY NIERUCHOMEJ ORAZ W STRUMIENIU POPRZECZNYM

Przedstawiono wyniki symulacji numerycznej procesu formowania się pęcherzy z dyszy nieruchomej oraz z powierzchni wirującego wałca umieszczonego w strumieniu płynącym poprzecznie do jego osi. Równania bilansu pędu rozwiązano numeryczne w ramach pakietu CFD w przestrzeni 2D oraz 3D. Do śledzenia ewolucji powierzchni międzyfazowej wykorzystano algorytm VOF (Volume of Fluid) wraz z modelem CSP napięcia powierzchniowego. Wyniki porównano z obserwacjami eksperymentalnymi uzyskanyymi za pomocą kamery cyfrowej 2000 klatek/s. W przypadku wirującego wałca uzyskano zadowalającą zgodność wyników symulacji z eksperymentem. W przypadku nieruchomej dyszy algorytm VOF dawał najlepsze wyniki dla większego przepływu gazu, kiedy w procesie dominowały siły bezwładności. Uzyskano bardzo dobrą zgodność kształtu i rozmiarów formujących się pęcherzy z pomiarami. W zakresie małych przepływów gazu, kiedy moment oderwania pęcherza był zdetymowany równoważą siły wyporu i napięcia powierzchniowego, wyniki symulacji przewidywały mniejsze rozmiairy pęcherzy w porównaniu z wartościami eksperymentalnymi.

Wpłynęło 18 października 2005