Modified airlift reactor with a cross-shaped internal and its hydrodynamic simulation by computational fluid dynamic method

Tao Zhang, Hai-zhen Wu, Chun-hua Feng and Chao-hai Wei

ABSTRACT
A modified airlift reactor (ALR) with a cross-shaped internal was developed to achieve beneficial flow conditions for an advantageous biological process. To study the effect of introduced internal on the reactor’s hydrodynamics, computational fluid dynamics simulation method was applied to predict the flow characteristics verified with experimental data. The internal introduction into a conventional ALR resulted in increased liquid velocities in the riser and the downcomer for 12% and 16%, respectively, maintaining the superficial gas velocity of 3 cm/s. The internal allowed the total gas holdup slightly decreased for 4.8%, whereas the turbulent kinetic energy in the riser was significantly reduced for 13%. The cross-shaped internal modified the flow structure in the ALR’s bottom zone due to, presumably, substitution of non-elastic collisions of liquid micro-lumps moving in counter-current directions with elastic collisions of micro-lumps with the wall of the internal, reducing the energy dissipation at the U-turn of the flow. The internal unified the direction of liquid velocity vectors in the bottom zone bringing an order to the liquid momenta in the flows close to the parallel ones and thus, saving energy in the ALR’s operation.

Introduction
Airlift reactors (ALRs) are considered as promising ones in various industrial applications including process industries, biological fermentation and wastewater treatment [1,2]. The ALRs’ advantages include their simple construction, absence of moving parts, large throughput on a continuous basis, low shear stress, efficient mixing and mass transfer with low energy requirement [3,4]. A number of research reports describe various ALRs’ configurations, their performance and hydrodynamics in the past decades [1,2,5]. A few kinds of internals, such as baffles, plates and screens, have been introduced into the conventional ALRs for their modifications [6–8]. It has been shown that these internals enhance the performance of the reactors or gain special destination in the flow pattern.

Airlift reactors are applied massively as biological reactors in large-scale wastewater treatment plants (WWTPs) [9]. However, instability of the gas–liquid flow in practical applications of the conventional ALRs is observed. For example, up-flow bubble swarms in the risers tend to swing periodically; quasi-periodic undulating bubble swarms in flat bubble column were described by Sokolichin and Eigenberger [10]. The unstable flow pattern in the ALRs used in wastewater treatment results in increased turbulence shear stress and energy consumption making the efficiency of mixing and mass transfer decreased.

To overcome the mentioned phenomena, a cross-shaped internal was introduced into the bottom zone of a conventional ALR approaching from the following considerations. At first, the cross-shaped internal effectively reduces the inlet hydraulic radius of the riser thus leading to an obvious increase in the Froude number ensuring the flow stability. Second, the internal is convenient in construction without noticeable impact to the investment cost: the concrete beams supporting the weight of the riser may be designed as the cross-shaped internal. Third, the energy dissipation at the bottom part of the ALRs is expected to decrease substantially: the micro-lumps of the liquid from the downcomer elastically collide with rigid walls of the cross-shaped internal, whereas non-elastic collisions of liquid micro-lumps with each other dominate in absence of the internal. This
makes the kinetic energy of liquid dissipated to a lesser extent in the U-turn of the flow in the bottom zone.

Limited possibilities of the full-scale studies on impacts of the internals require determining the flow patterns in the ALRs together with the limiting factors of the scale-up in numerical simulations. In the past decades, computational fluid dynamics (CFD) has emerged as a powerful tool for predicting hydrodynamic characteristics [11]. The advantage of CFD modelling consists of the reactor geometries and the scale-up effects considered in its mesh procedure [12]. The CFD simulation also helps to greatly reduce the test costs at the post-construction stage [13]. Visual display of the flow field characteristics in, for example, mixing and separation, together with the energy dissipation is also an advantage of the CFD approach.

The objectives of this study include the development of a modified ALR with a cross-shaped bottom internal using the CFD method for the hydrodynamic characteristics of the reactor. The CFD simulation consequences for the reactor hydrodynamics are the subject of verification in comparison with the experimental data. The impact of the cross-shaped internal on the reactor hydrodynamic characteristics was studied by analyzing the detailed flow field data for ALRs with and without the internal.

**Materials and methods**

**Reactor configuration and operating conditions**

The ALR, modified with a cross-shaped internal, used in the experiments was composed of a conventional ALR and a cross-shaped internal mounted beneath the riser. Figure 1 shows the axial profile of the proposed reactor and the detailed geometry structure of the cross-shaped internal. The sparger with 409 holes of 0.5 mm in diameter was placed at the bottom of the reactor. Air is escaped from the sparger embedded in the conical bottom of the reactor. The dimensions of the experimental reactors are given in Table 1. The reactor was made of transparent 5-mm thick Plexiglas®.

The experiments were carried out at room temperature and at atmospheric pressure. Air and tap water were used as gas and liquid phases, respectively. The reactor was run in a recirculating batch mode at the static height of the gas-free water of 1.2 m. Out of the preliminary knowledge [14], the critical superficial gas

![Figure 1](image.png)

Figure 1. Schematic diagram of the modified reactor.

Table 1. Dimensions for the experimental reactor.

| Symbol | Dimension | $H_1$ | $H_2$ | $H_3$ | $H_4$ | $D_1$ | $D_2$ | $D_3$ | $D_4$ | $L_1$ | $\alpha$ |
|--------|-----------|-------|-------|-------|-------|-------|-------|-------|-------|-------|---------|
|        |           | 1425  | 56.5  | 1100  | 50    | 160   | 110   | 47    | 60    | 5     | 45      |

Note: The values for $H_{1,2,3,4}$, $D_{1,2,3,4}$ and $L_1$ are in mm and the value for $\alpha$ is in degrees.
velocity for the flow regime transition from a bubbly flow to a turbulent one in the conventional ALRs is about 2.0 cm/s. Six superficial gas velocities of 0.1, 0.5, 0.75, 1.0, 2.0 and 3.0 cm/s, relative to the cross-sectional area of the riser, were experimentally tested to cover both flow regimes. These velocities also cover the typical operation ranges in practical bioprocesses, including fermentation processes and wastewater treatment. Air was supplied through an air compressor (model No. 550W-8L, Outstanding Co., China) with the flow rate adjusted using a pre-calibrated rotameter (model No. LZB-4W/6W/10W, S-Huan, China).

**Measurements**

The liquid velocity in the downcomer was estimated by the conductivity tracer technique [12,15]. A pulse of 2 mL of saturated aqueous solution of sodium chloride was injected at the top of the downcomer. Signals of the conductivity probe (model No. DJS-0.1CF, Leici, China) in response to the salt tracer injection were recorded at two positions near the top and the bottom of the downcomer at a distance of 0.8 m from each other. The probe signals were transmitted by a conductivity meter (model No. DDG-5205A, Leici, China) to the PC processor through an A/D convector (model No. PCI8532, ART, China) with the signals sampling rate of 10 Hz. Since the liquid exhibits nearly a plug flow behaviour [15], the liquid velocity \( V_{ci} \) in the downcomer may be calculated from the distance between the vertical positions of the probes divided by the time interval between the signal peaks. The reported values of liquid velocity were the average ones of four parallel measurements. The gas hold-up in the riser was determined by measuring the differential pressure between two sampling ports near the top and the bottom of the riser using U-tube manometers [16]. The overall gas hold-up was measured by the ordinary gas cut-off method when the levels of aerated and static gas free liquid were compared. The aerated liquid level fluctuates at high superficial gas velocities requiring the average of four measurements taken for each operating condition [14].

Based on the above-mentioned methods, average flow parameters for the whole reactor or some of its regions were acquired, such as total gas hold-up, gas hold-up in the riser, and liquid velocity in the downcomer. To compare with these experimental measurement data, the CFD simulation results, including the gas hold-up and the liquid velocities, were also the average values in a specific region. These average values can be easily acquired by integrating corresponding variable on volume of the referred region and averaging it on the volume.

**CFD modelling**

In the present work, the Euler-Euler two-fluid model was used to simulate the fluid flow in the reactor under consideration. This model has an advantage when the dispersed phase volume fraction is relatively high. The liquid was treated as the continuous phase and bubbles were the dispersed phase. Interacting mixed gas and liquid phases satisfy the unity rule across the computational domain. Both phases were considered incompressible and sharing a uniform pressure field. Computation proceeded under the isothermal condition omitting the energy equation. The volume-averaged mass and momentum conservation equations were solved for each phase.

(1) Continuity equations

\[
\frac{\partial (\alpha_i \rho_i)}{\partial t} + \nabla \cdot (\alpha_i \rho_i \tilde{u}_i) = 0
\]

where \( i \) denotes gas or liquid phase, \( t \) is time, \( \rho_i, \tilde{u}_i \) and \( \alpha_i \) are density, velocity vector and volume fraction of phase \( i \), respectively. The total sum of phasic volume fractions should equal 1.0.

(2) Momentum equations

\[
\frac{\partial (\alpha_i \rho_i \tilde{u}_i)}{\partial t} + \nabla \cdot (\alpha_i \rho_i \tilde{u}_i \tilde{u}_i) = -\alpha_i \rho_i \tilde{g} + M_{ij} + \alpha_i \rho_i \tilde{g} + M_{ij}
\]

where \( p \) is a uniform pressure field, \( \mu_{\text{eff}} \) is effective viscosity, \( \tilde{g} \) is gravitational acceleration vector, \( M_{ij} \) is interphase momentum exchange terms.

(3) Interphase momentum exchange

The interphase momentum exchange between the continuous liquid phase and the dispersed bubbles has been extensively studied earlier [17,18]. For a typical gas-liquid flow, the interphase forces include drag, lift, and virtual mass forces and the turbulent dispersion force [19,20]. Drag force is usually the most important interphase force, which is in proportion with the slip velocity between two phases. Equation (3) is commonly used to describe the drag force [18,21]:

\[
F_D = \frac{3}{4} \alpha_g \rho_l C_D \frac{d_b}{d_b} |\tilde{u}_g - \tilde{u}_l||\tilde{u}_g - \tilde{u}_l|
\]

where \( C_D \) is a drag force coefficient, \( d_b \) is an equivalent diameter of bubbles set as 5 mm based on experimental
At high gas holdups, Equation (4) should be used instead for substantially smaller drag force in a low-density foam [17]:

$$F_D = \frac{3}{4} \alpha_g \alpha_l \rho_l \frac{C_D}{d_0} \left[ \bar{u}_g - \bar{u}_l \right] \left[ \bar{u}_g - \bar{u}_l \right]$$

(4)

The $C_D$ was calculated as shown below [23]:

$$C_D = \max \left( \frac{24}{Re_b} \left( 1 + 0.15 \frac{Re_b^{0.87}}{Eo} \right), \frac{8}{3} \frac{Eo}{Eo + 4} \right)$$

(5)

$$Re_b = \frac{\rho_l d_b \left[ \bar{u}_g - \bar{u}_l \right]}{\mu_l}$$

(6)

$$Eo = \frac{g \left( \rho_l - \rho_g \right) d_b^2}{\gamma}$$

(7)

where $Eo$ is Eotvos number, $Re_b$ is Reynolds number of bubbles, $\gamma$ and $\mu_l$ are the surface tension and molecular viscosity of the liquid, respectively.

According to the suggestion made by Sokolichin et al. [24], neglecting the lift force can still lead to satisfying results if no clear experimental evidences for its direction and magnitude are available. They also pointed out that the effect of virtual mass force is negligible except for high frequency fluctuations of the slip velocity. Besides, the virtual mass force is rather small compared to the drag force in the gas–liquid flow. The turbulent dispersion force mainly influences the radial distribution of the bubbles, and it was not considered in this study. Therefore, the lift force, virtual mass force and turbulent dispersion force were not considered in the simulation.

(4) Turbulence closure

The choice of turbulence closure model is of great importance for the multiphase flow simulations. Although there are sophisticated models in use, the turbulence of the two-phase flow is more often modelled using an extension of the standard $k-\varepsilon$ model for its offering a reasonable accuracy at low computational cost as well as a robust convergence. Taking into consideration relatively low superficial gas velocities, the dispersed gas phase appears to be dilute in the continuous liquid phase, making the dispersed standard $k-\varepsilon$ model acceptable to model the turbulence in this study [25,26]:

$$\frac{\partial (\alpha_l \rho_l \kappa_l)}{\partial t} + \nabla \cdot (\alpha_l \rho_l \bar{u}_l \kappa_l) = \nabla \cdot \left( \alpha_l \frac{k_{l1}}{\sigma_k} \nabla \kappa_l \right) + \alpha_l \left( G_{k,l} - \alpha_l \rho_l \bar{u}_l \right) + \alpha_l \rho_l \prod_{ij}$$

(8)

where $C_{\mu}$ is a constant set at 0.09, $k_l$ and $\varepsilon_l$ are the turbulence kinetic energy and the turbulence dissipation rate of continuous liquid phase, which are obtained by solving the above transport equations. The standard constants used in the turbulence equations are $C_{1_{e}} = 1.44$, $C_{2_{e}} = 1.92$, $\sigma_k = 1.0$ and $\sigma_{\varepsilon} = 1.3$. $G_{k,l}$ is the generation of the turbulent kinetic energy, $\prod_{v_0}$ and $\prod_{ij}$ are the source terms which can be included to the model taking into account the influence of the dispersed phase on the continuous phase [27]. All the definitions for the three last terms may be found in the Fluent user’s guide [25]. The turbulence of the dispersed gas phase was modelled in a set of algebraic equations relating the dispersed phase turbulence quantities to those of the continuous phase [21].

The turbulence viscosity of the liquid phase ($\mu_{t,l}$) was calculated by Equation (10), the effective viscosity of the liquid phase ($\mu_{eff,l}$) used in Equation (2) was set as the sum of the molecular viscosity and the turbulence viscosity of the liquid phase as in Equation (11):

$$\mu_{t,l} = C_{\mu} \rho_l \left( \frac{k_{l1}^2}{\varepsilon_l} \right)$$

(10)

$$\mu_{eff,l} = \mu_l + \mu_{t,l}$$

(11)

Boundary conditions

The no-slip boundary condition was applied to the liquid phase, and the free-slip boundary condition, to the gas phase at the walls of the reactor [28]. The enhanced wall function was used in the vicinity of the walls. The gas inlet at the sparger was set as the velocity inlet. The gas was considered to be delivered evenly to the surface of the sparger, and the normal velocity was considered equal to the experimental gas velocity [29]. Pressure at the ALR gas vent was set at the ambient value [30]. Because the modified ALR was planar symmetrical about the plane, crossing the reactor axis along the direction of angular bisector of the right-angle formed by the cross-shaped internal, the Neumann boundary conditions with zero gradients for the flow parameters were applied to the symmetrical plane of the reactor.

Mathematical solution

The above-given partial differential equations were solved using commercial CFD software Fluent v. 6.3.
Unstructured tetrahedral meshes were implemented in the reactor model with or without the cross-shaped internal. Different mesh interval sizes of 2, 3 and 4 mm were selected for the independence test of simulation. Figure 2 shows the calculation model of the reactor and the relevant mesh details. An optimal mesh resolution was acquired when the mesh with the smaller size resulted in the final solution varying no more than 2%. The optimal mesh size was used in the subsequent simulations.

The segregated pressure-based solver was adopted to solve the model equations using the phase coupled SIMPLE algorithm for the pressure–velocity coupling. Discrete equations for momentum, turbulence kinetic energy and the dissipation rate were developed following the second-order upwind scheme. The volume fraction equation was developed following the QUICK scheme. Time-dependent unsteady calculations and the first-order implicit scheme were used in the iteration process. Under-relax factors for all variables were set as the default values. For each time step, internal iterations of 100 times were calculated to ensure an adequate convergence degree for all the model equations.

To enhance the stability of calculation, the variable time stepping strategy was used, viz. 200 steps at 0.0001 s, 100 steps at 0.0003 s, 300 steps at 0.0005 s, 800 steps at 0.001 s.

Figure 2. Reactor model and its mesh characteristics: unmeshed modified airlift reactor (a); meshed modified airlift reactor (b); meshed conical reactor bottom (c); cross section mesh of riser and downcomer (d); vertical section mesh of riser and downcomer (e); mesh of degassing zone (f).
steps at 0.001 s, 3000 steps at 0.003 s, 2000 steps at 0.005 s and a final time step at 0.01 s. The total flow calculation time was over 120 s, in which the first 80 s were to ensure the flow reaching a steady or pseudo-steady state, and the following 40 s were used for the result evaluation. The residual convergence criterion for all variables was set as $1 \times 10^{-4}$.

Results and discussion

Mesh independence assurance

To avoid the impact of inadequate mesh on the numerical simulation results, the study of the mesh independence was carried out at the superficial gas velocity of 0.01 m/sin the riser. Figure 3 shows the CFD simulation results at different mesh sizes for the modified ALR, including the gas holdup and the liquid velocity in the riser, and the liquid velocity in the downcomer. In order to assure the universal character of the comparison, the dimensionless mesh sizes, velocities and gas holdups based on data from the coarsest mesh were used.

One can see that the decreasing grid size leads to small variances for the gas holdup and liquid velocities: when the mesh sizes were of 2 and 3 mm, the differences of simulation results for the gas holdup and the liquid velocity in the riser, and the liquid velocity in the downcomer were within 8.7%, 4.5% and 2.5%, respectively. No finer meshes were tested because of the limited calculation resource. In fact, for the three-dimensional meshes used in this study, increasing the mesh resolution below the mesh size of 3 mm did not significantly increase the accuracy of the simulation results considerably, however, increasing the computation expense. Therefore, the mesh size of 3 mm was sufficient to assure the valid simulation results and was used in subsequent simulations.

Experimental verification of the CFD models

Data acquired from the non-modified ALR were used for verification of the CFD model. The hydrodynamic experiments were carried out at five superficial gas velocities of 0.1, 0.5, 1.0, 2.0, and 3.0 cm/s. Since the non-modified reactor satisfies the axis-symmetry condition, a two-dimensional (2D) CFD simulation method was applied [31]. For comparison, the three-dimensional (3D) CFD simulation was also carried out. Finally, for each superficial gas velocity, 2D and 3D CFD simulation results were compared with the experimental results.

Liquid flow in the downcomer, considered as a plug flow, of the relatively stable velocity was extensively used in studies of the ALRs hydrodynamic properties [12,32]. The results obtained in the present study for the liquid velocity in the downcomer are shown in Figure 4. One can see that both 2D and 3D CFD simulation results agree well with the experimental values. The maximum difference between the experimental values and the simulation results does not exceed 9.3%. An insignificant difference observed between the simulation results of 2D and 3D allows the simulation dimension being the matter of choice. This result contradicts the previously reported larger deviations of the 2D simulation results from the experimental ones [25,33]. Smaller deviations were attributed to the 2D axisymmetric models. In this study, the 3D simulation showed better results than the 2D reproduction of the experimental results, especially at the middle range of the superficial gas velocities. Besides, only 3D simulation may be used for the non-axisymmetrical reactor with the cross-shaped internal, making the 3D CFD simulation adopted in the following studies.

Figure 3. Analysis of the mesh independence.

Figure 4. Comparison of experimental data and CFD simulation results for liquid velocity in the downcomer.
Effect of the cross-shaped internal on the ALR hydrodynamics

To study variations in the fluid flow, comparative CFD simulation studies were conducted on reactors with and without the cross-shaped internal at the inlet superficial gas velocities of 0.25, 0.5, 0.75, 1.0, 2.0 and 3.0 cm/s. Liquid velocities and total gas holdups were used as the resultant hydrodynamic parameters.

Figure 5 compares the variance of liquid velocities in the riser and those in the downcomer between the reactors with and without the internal. It can be seen that the introduction of the cross-shaped internal increased the liquid velocities in the riser and in the downcomer by 12% and 16%, respectively. Regardless of the presence of the internal, both velocities increased with the increasing superficial gas velocity, with a rapid increase within the low gas superficial velocity numbers. The introduction of the internal enabled faster circulating velocity of liquid at equal superficial gas velocities. This was attributed to the reduced energy consumption in the bottom section of the reactor with the internal introduced.

Figure 6 shows that, regardless of the presence of the internal, the total gas holdup gradually increases with the increasing superficial gas velocity. It should be noted that the introduced internal leads to a slight reduction in the total gas holdup when it is compared with the reactor without the internal. At the maximum superficial gas velocity of 3 cm/s, the total gas holdup descending range also reached its maximum at 4.8%. It can be thus inferred that the introduction of the internal results in the increased liquid velocities, which shortens the residence time of bubbles in the reactor.

Effect of the internal on turbulent kinetic energy (TKE)

In the conventional ALR, the turbulence of gas-liquid flow mainly occurs in the riser. In order to study the effect of the cross-shaped internal on TKE, average values of the latter in the riser acquired by the CFD simulation method were compared with and without the cross-shaped internal.

As shown in Figure 7, the average TKE in the riser gradually increased with the increase in the superficial gas velocity, regardless of the presence of the internal. Similar to the liquid velocities, the TKE variation is more pronounced at the small superficial gas velocities. Compared with the reactor without internal, the cross-shaped internal distinctly decreased the TKE in the riser: at the superficial gas velocity of 3 cm/s, the maximum
reduction of TKE reached 13%. This arrangement may be favourable for shear stress-sensitive microorganisms, for example. The reason behind the reduced turbulence may be attributed to the substantial decrease in the hydraulic radius of the flow expressed numerically by the reduction of the Reynolds number and the increased Froude number. At the reduced turbulence, the liquid flow in the riser is thus stabilized. The impact of the cross-shaped internal, similar to a tetra-spoke one, brings the idea of multi-spoke internal showing stronger effect in TKE reduction at even more stabilized flow.

Figure 8(a,b) show the profiles of the velocity vector at the bottom of the reactor, with respect to the reactor without the cross-shaped internal and the reactor with it, respectively. One can see that the velocity distribution at the radial section of the riser is more homogeneous when the internal is introduced. It means that the velocity gradient in the riser is relatively low reducing the friction energy loss and moderating the shear strain in the liquid flow. Also, the velocity distribution at the conical bottom of the reactor is more homogeneous and the direction of the liquid movement is more uniform with the internal introduced. This kind of uniform movement may form the maximum resultant force and lead to a higher velocity of the liquid in the riser. On the contrary, as one can see in Figure 8(a), the velocity vectors at the conical bottom of the reactor are nearly in the opposite directions when the internal is absent thus making the non-elastic collisions between liquid micro-lumps dominant. These collisions may result in noticeable energy losses, depending on the magnitude and direction of velocity vectors. This makes random fluctuations in the liquid flow influencing the movement status in the bottom zone of the reactor. In other words, the random fluctuations may lead to swings of the bubble swarms in the gas-liquid flow. One can see from Figure 8(b) the major liquid velocity vectors in the downcomer interacting with the rigid wall of the internal, changing their directions and moving upwards. This proved the idea of the cross-shaped internal substituting the opposite direction non-elastic collisions with the elastic collisions between the liquid micro-lumps and the walls of the internal. At the same time, the internal may play an important role splitting the bubbles emitted by the gas sparger into smaller ones. This helps to avoid the bubbles concentrating in the middle part of the riser and thus, to form a more uniform radial distribution of bubbles in the radial direction [34]. The dead volume of the reactor, where the liquid velocity is rather low, is also minimized by the internal.

Conclusions

A cross-shaped internal introduced into the bottom zone of a conventional ALR stabilizes the flow and reduces the turbulence intensity. The CFD simulation method established on Eulerian–Eulerian two-fluid platform was used to study the flow details in the modified ALR. The mesh independent simulation results were acquired and verified with the relevant experimental data. The verified CFD models were extended to study the ALR flow characteristics. The results showed that the cross-shaped internal significantly increased the liquid velocities in the riser and the downcomer, the total gas holdup in the reactor slightly decreased, although the turbulent kinetic energy in the riser decreased. Therefore, the cross-shaped internal, acting as a multi-spoke one, presents a feasible modification of the ALR achieving improved characteristics in applications of biological fermentation and wastewater treatment.
Notations

**General symbols**

- \( C_D \): drag force coefficient
- \( d_b \): bubble diameter (m)
- \( E_o \): Eotvos number
- \( F \): interphase momentum exchange force (N/m³)
- \( g \): gravitational acceleration (m/s²)
- \( \text{g} \): gravitational acceleration vector (m/s²)
- \( \text{Re}_b \): bubble Reynolds number
- \( \bar{u} \): phase velocity vector (m/s)

**Greek symbols**

- \( \alpha \): phase volume fraction
- \( \mu \): molecular viscosity (Pa/s)
- \( \gamma \): surface tension (N/m)
- \( \rho \): density (kg/m³)

**Subscripts**

- \( g \): gas phase
- \( i \): liquid phase
- \( i \): phase
- \( D \): drag force

Acknowledgments

The authors are grateful to the School of Chemistry and Chemical Engineering of the South China University of Technology for their generous help with the CFD computing platform. The authors also thank Sergei Preis, professor of Lappeenranta University of Technology, for English-language editing and helpful suggestions.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

This study was supported by the National Natural Science Foundation of China [grants number 21406096, [grants number 51278199]; the State Key Program of National Natural Science of China [grant number 21037001]; the Program of Youth Science Foundation of Jiangxi Province [grant number 20142BAB213022] and the research initiation project of Zhongshan Institute [grant number 415YKQ07].

ORCID

Tao Zhang http://orcid.org/0000-0003-1296-2947

References

[1] Cockx A, Liné A, Roustan M, et al. Numerical simulation and physical modeling of the hydrodynamics in an air-lift internal loop reactor. Chem Eng Sci. 1997;52:3787–3793.
[2] Merchuk JC. Airlift bioreactors: review of recent advances. Can J Chem Eng. 2003;81:324–337.
[3] Feng W, Wen J, Liu C, et al. Modeling of local dynamic behavior of phenol degradation in an internal loop airlift bioreactor by yeast Candida tropicalis. Biotechnol Bioeng. 2007;97:251–264.
[4] Jia X, Wen J, Zhou H, et al. Local hydrodynamics modeling of a gas–liquid–solid three-phase bubble column. AIChE J. 2007;53:2221–2231.
[5] Luo H-P, Al-Dahhan MH. Local characteristics of hydrodynamics in draft tube airlift bioreactor. Chem Eng Sci. 2008;63:3057–3068.
[6] Zhang T, Wang J, Wang T, et al. Effect of internal on the hydrodynamics in external-loop airlift reactors. Chem Eng Process. 2005;44:81–87.
[7] Yu W, Wang T, Liu M, et al. Bubble circulation regimes in a multi-stage internal-loop airlift reactor. Chem Eng J. 2008;142:301–308.
[8] Yuguo Z, Zhao W, Xiaolong C. Liquid circulation time in an external loop airlift bioreactor with inner sieve plates. Chinese J Antibiot. 1999;2:105–108.
[9] Wei C, Zhang T, Feng C, et al. Treatment of food processing wastewater in a full-scale jet biogas internal loop anaerobic fluidized bed reactor. Biodegradation. 2011;22:347–357.
[10] Sokolichin A, Eigenberger G. Applicability of the standard \( k-e \) turbulence model to the dynamic simulation of bubble columns: part I. Detailed numerical simulations. Chem Eng Sci. 1999;54:2273–2284.
[11] Terasima M, Goel R, Komatsu K, et al. CFD simulation of mixing in anaerobic digesters. Bioresource Technol. 2009;100:2228–2233.
[12] van Baten JM, Ellenberger J, Krishna R. Hydrodynamics of internal air-lift reactors: experiments versus CFD simulations. Chem Eng Process. 2003;42:733–742.
[13] Ding J, Wang X, Zhou X-F, et al. CFD optimization of continuous stirred-tank (CSTR) reactor for biohydrogen production. Bioresource Technol. 2010;101:7005–7013.
[14] Luo H-P, Al-Dahhan MH. Local gas holdup in a draft tube airlift bioreactor. Chem Eng Sci. 2010;65:4503–4510.
[15] Vial C, Porcin S, Wild G, et al. Experimental and theoretical analysis of axial dispersion in the liquid phase in external-loop airlift reactors. Chem Eng Sci. 2005;60:5945–5954.
[16] Mohanty K, Das D, Biswas MN. Hydrodynamics of a novel multi-stage external loop airlift reactor. Chem Eng Sci. 2006;61:4617–4624.
[17] Huang Q, Yang C, Yu G, et al. Sensitivity study on modeling an internal airlift loop reactor using a steady 2D two-fluid model. Chem Eng Technol. 2008;31:1790–1798.
[18] Panneerselvam R, Savithri S, Surender GD. CFD simulation of hydrodynamics of gas–liquid–solid fluidised bed reactor. Chem Eng Sci. 2009;64:1119–1135.
[19] Law D, Battaglia F, Heindel TJ. Model validation for low and high superficial gas velocity bubble column flows. Chem Eng Sci. 2008;63:4605–4616.
[20] Monahan SM, Fox RO. Effect of model formulation on flow-regime predictions for bubble columns. AIChE J. 2007;53:9–18.
[21] Roy S, Dhotre MT, Joshi JB. CFD simulation of flow and axial dispersion in external loop airlift reactor. Chem Eng Res Des. 2006;84:677–690.

[22] Wadaugsorn K, Limtrakul S, Vatanatham T, et al. Hydrodynamic behaviors and mixing characteristics in an internal loop airlift reactor based on CFD simulation. Chem Eng Res Des. 2016;113:125–139.

[23] Tomiyama A. Struggle with computational bubble dynamics. Multiphase SciTechnol. 1998;10:369–405.

[24] Sokolichin A, Eigenberger G, Lapin A. Simulation of buoyancy driven bubbly flow: established simplifications and open questions. AIChE J. 2004;50:24–45.

[25] Fluent incorporation, Fluent 6.3 user’s guide. Lebanon: Fluent Inc.; 2006.

[26] Kerdouss F, Bannari A, Proulx P. CFD modeling of gas dispersion and bubble size in a double turbine stirred tank. Chem Eng Sci. 2006;61:3313–3322.

[27] Mohajerani M, Mehrvar M, Ein-Mozaffari F. CFD analysis of two-phase turbulent flow in internal airlift reactors. Can J Chem Eng. 2012;90:1612–1631.

[28] Rzehak R, Ziegenhein T, Kriebitzsch S, et al. Unified modeling of bubbly flows in pipes, bubble columns, and airlift columns. Chem Eng Sci. 2017;157:147–158.

[29] Xu T, Jiang X, Yang N, et al. CFD simulation of internal-loop airlift reactor using EMMS drag model. Particuology. 2015;19:124–132.

[30] Chen L, Bai Z. CFD simulation of the hydrodynamics in an industrial scale cyclohexane oxidation airlift loop reactor. Chem Eng Res Des. 2017;119:33–46.

[31] Zhang T, Wei C, Feng C, et al. A novel airlift reactor enhanced by funnel internals and hydrodynamics prediction by the CFD method. Bioresource Technol. 2012;104:600–607.

[32] Blażej M, Cartland Glover GM, Generalis SC, et al. Gas–liquid simulation of an airlift bubble column reactor. Chem Eng Process. 2004;43:137–144.

[33] Appasani P. CFD simulation of the hydrodynamics and interphase mass transfer in airlift reactors [dissertation]. Halifax (NS): Dalhousie University; 2007.

[34] Ziegenhein T, Lucas D. Observations on bubble shapes in bubble columns under different flow conditions. Exp Therm Fluid Sci. 2017;85:248–256.