Numerical analysis of hydrodynamics in a mechanically agitated gas–liquid pseudophase system

Magdalena Cudak¹

Received: 27 July 2018 / Accepted: 25 September 2018 / Published online: 1 October 2018
© The Author(s) 2018

Abstract

The paper presents the results of numerical simulations of hydrodynamics in a gas–liquid pseudophase system (sucrose solution with yeast suspension). The simulations were performed in a bioreactor with a working volume of 0.02 m³, equipped with a Rushton turbine or Smith turbine stirrers. The results of the study were developed in the form of vectors of liquid velocity and the contours of the analyzed magnitudes. Also, the local values of the gas hold-up φ in the gas–liquid pseudophase system were presented. The results of numerical simulations of the gas hold-up were compared to the results of my own experimental study.

Keywords Bioreactor · Mixing · Gas hold-up · Numerical modeling · Gas–liquid pseudophase system

List of symbols

B	Width of the baffle (outer diameter of the tubular baffle), m
D	Diameter of the impeller, m
d_d	Sparger diameter, m
H	Liquid height in the vessel, m
h	Distance between impeller and bottom of the vessel, m
k_c	Consistency index, Pa s ^{m1}
m	Flow index
n	Impeller speed, s ⁻¹
T	Inner diameter of the agitated vessel, m
V_g	Gas flow rate, m ³ s ⁻¹
z	Axial coordinate, m

Greek letters

φ	Gas hold-up
η	Dynamic viscosity of the liquid, Pa s
ρ	Density of the liquid, kg m ⁻³
k	Interfacial tension, N m ⁻¹

Introduction

Multi-phase systems in a mixer are widely used in chemical and biochemical engineering processes. Their hydrodynamics is very complicated due to the interactions occurring between the rotating stirrer and the baffles. The application of the computational fluid dynamics (CFD) method allows to reduce the costs and time needed to perform experimental research (Wang et al. 2014; Sommerfeld and Decker 2004).

Due to the fact that the processes in bioreactors run in multiphase systems (biophase–gas–liquid), the following should be defined in numerical calculations: due to the presence of more than one phase (multiphase model), due to turbulent flow (turbulence model) and due to different sizes and properties of the dispersed phase (models that take into account division and connection (birth and death), for example, gas bubbles or microbial cells) (Ranade 2002; Jaworski 2005).

In the case of modeling the hydrodynamics of the gas–liquid system, characterized by a relatively high proportion of the dispersed phase (gas phase), the two-phase model is often used as a model in which both phases are treated as two interpenetrating continuums (Euler–Euler approach) (Kerdouss et al. 2008; Jaworski 2005; Joshi et al. 2011; Kaiser et al. 2011; Knopkar and Ranade 2006). To obtain better agreement between the data obtained as a result of simulations and experimental data, the computational fluid dynamics (CFD) method is combined with the population balance method (PBM) (Gimbun et al. 2009; Ranganathan and Sivaraman 2011; Musiał et al. 2014, 2017). Based on the

✉ Magdalena Cudak
cudak@zut.edu.pl

¹ Department of Chemical Engineering, West Pomeranian University of Technology, Szczecin, al. Piastów 42, 71-065 Szczecin, Poland

results of numerical simulations of hydrodynamics of multiphase systems, a number of information on the distribution of liquid and gas velocities, local values of gas hold-up and distributions of kinetic energy of turbulence and its dissipation in a stirred tank can be obtained (Oniscu et al. 2002; Vrabel et al. 2000; Delafosse et al. 2014; Gelves et al. 2014).

Gimbun et al. (2009) used a two-fluid model and an Euler–Euler approach to perform a simulation in gas–liquid system in a tank with a Rushton turbine or Smith turbine stirrer. Local bubble size distributions were obtained using a quadrature method of moments (QMOM), whereas local $k_L a$ values were estimated using the Higbie permeation theory and the model of the renewed surface. Obtained results regarding hydrodynamics of the gas–liquid system (power number, local bubble sizes, dissolved oxygen concentration, average two-phase flow velocities) were quite consistent with the literature data, which proves that the simulations were performed correctly.

The results of numerical simulations showing the influence of baffles and the curvature shape of stirrer blades (concave or convex blades) on hydrodynamics in liquid or gas–liquid systems mixed in a tank equipped with a turbine stirrer with curved blades were obtained by Musial et al. (2014). They used the k – ε or SST turbulence model and Euler–Euler approach to predict turbulent flow. Based on the obtained simulations, they found that hydrodynamic interactions between the curved stirrer blades and mixing tank baffles are responsible for disturbing the symmetry of the fluid flow in the axial and radial planes of the stirrer tank. Numerical analysis of the impact of sucrose concentration on the distribution of continuous phase velocity, share of gas hold-up and size of gas bubbles was the subject of research by Musial et al. (2017). Based on the conducted simulations, they found that the local liquid velocity values decrease with the increase in sucrose concentration in the system, while the distributions of gas hold-up and the size of gas bubbles to a small extent depend on sucrose concentration.

The modeling of the impact of apparent gas velocity w_{og} on the hydrodynamics of flow and the processes of mass exchange in the gas–liquid system in bioreactor was examined by Devi and Kumar (2014). They performed numerical simulations in a bioreactor with two Rushton turbine or CD 6 stirrers. They found that dissipation rate ε increases with the increase in apparent linear gas velocity w_{og} . Significantly higher values of relative mixing power P_g/P_o were obtained for a CD 6 stirrer than for Rushton turbine stirrer. However, Devi and Kumar (2014) did not find a significant influence of the stirrer type on the value of the average coefficient of mass transfer $k_L a$. Xia et al. (2009) modeled the flow dynamics in the bioreactor with various stirrer combinations. Simulation results were compared with experimental results obtained in analogously equipped bioreactors. The *Streptomyces avermitilis* yeast suspension was a biophase

in all cases. Measurements and numerical simulations were carried out in bioreactors in which three high-speed stirrers were mounted on a common shaft. Comparing the results of experimental tests and numerical simulations, it was found that the most favorable conditions are provided by a system of three stirrers: two pumping down modified propeller stirrers and a turbine stirrer with six curved blades (bottom).

The study presented in this paper aimed to determine the influence of stirrer speed and volumetric gas flow rate on the gas hold-up in a bioreactor with a Rushton turbine or Smith turbine stirrer. The results of numerical simulations of the gas hold-up were compared to the results of my own experimental study.

The range of simulations

Numerical simulations of hydrodynamics of a gas–liquid pseudophase system were carried out using ANSYS Workbench and CFX solver version 16.1. A biophase–liquid system was used as a liquid pseudophase, which was a 1% aqueous solution of sucrose with yeast suspension. Thus, the yeast suspension was not treated as a separate phase during the numerical simulations. The geometry of the system was created using the ANSYS Design Modeler (Fig. 1). The bioreactor with diameter $T=0.288$ m was filled with an aqueous solution of sucrose with yeast suspension up to $H=T$. There were four standard baffles with a diameter of $B=0.1$ T and a centrally placed stirrer shaft in the tank. On the shaft of the stirrer, at a distance of $h=0.33$ H from the bottom, a high-speed Rushton (RT) turbine or Smith turbine stirrer (CD 6) was placed (Fig. 2). In both cases, the diameter of the stirrer was $D=0.33$ T . Halfway between the bottom of the tank and the stirrer, there was a gas distributor ($d_d=0.7D$) in the form of a perforated ring with six holes of 0.002 m in diameter each.

Experimental studies were carried out in a bioreactor with the same geometrical parameters as those used in numerical simulations. Experimental research were carried out for several stirrer speeds n and several volumetric gas flow rates V_g . The frequency of stirrer speed and volumetric gas flow rates were used for comparison, which were used in numerical simulations. The gas hold-up was calculated from the Eq. 1 as the average values from 10 reading of the increase in the height of the gas–biophase–liquid system h_{g-b-c} on the vertical wall of the stirred tank.

$$\varphi = \frac{h_{g-b-c}}{h_{g-b-c} + H} \quad (1)$$

Using the ANSYS Mesh software, a tetrahedral numerical grid with a cell number of about 1 million was generated.

Fig. 1 Geometrical parameters of the stirrer tank

Fig. 2 Stirrer **a** Rushton turbine stirrer (RT), **b** Smith turbine stirrer (CD 6)

Characteristic parameters of the mesh were as follows: (a) skewness measure 0.23; (b) orthogonal quality 0.86. Due to the use of multiple reference system (MRF), the grid was divided into two zones:

- in the vicinity of the stirrer,
- in the rest of the tank.

In the area of the stirrer and in the vicinity of the tank walls, the grid was characterized by a much denser structure than in the remaining volume of the tank. The MRF zone is much larger than the stirrer volume. It was designated as a disk with a diameter of 0.164 m. In the MRF zone, the unstructured mesh consisted of about 150,000 tetrahedral elements.

The mathematical model of the process and the corresponding initial and boundary conditions were defined using ANSYS CFX-Pre. The continuous phase was modeled using the Shear Stress Transport model (SST). The assumptions of the k – ϵ model regarding the volume of the stream and the assumption of the k – ω model taking into account mixing functions in the boundary areas are taken into account in the SST model (ANSYS 2015). Due to the non-uniform size of gas bubbles in the gas–liquid system, numerical modeling

also included coalescence and bubbles disruption models as well as a model determining the bubbles size distribution (Podgórska 2006). To describe the gas bubble disruption, Luo Svendsen's model was used, taking into account the theory of isotropic turbulence and the theory of probability (Wang et al. 2014; ANSYS 2015). The phenomenon of gas bubbles coalescence was described using the Prince Blanch model (ANSYS 2015). In this model, the authors assumed that the connection of two bubbles into one occurs in three stages: in the first—the bubbles collide with a small amount of fluid between them, in the second—the liquid film reaches a critical thickness, in the third—the film breaks and the bubbles connect. Thus, the process of bubbles connection depends on the speed of collision between the two bubbles and on the effectiveness of the collision determined on the basis of the time needed for the connection. This model is often simplified taking into account only the frequency of collisions (Wang et al. 2014).

The dispersion of gas bubbles was determined using the Particle model (ANSYS 2015). In addition, the forces of inter-phase resistance were taken into account, as defined by the Schiller–Naumann equation, as well as the uplift force according to the Tomiyama model (ANSYS 2015). The transfer of turbulence between the phases was modeled on the basis of the extended Sato correlation (ANSYS 2015). Gas bubble sizes, divided into ten classes, were modeled using the Multiple Size Group (MUSIG) model that combines flow simulations with the population balance method (Wang et al. 2014; Ahmed et al. 2010; Frank et al. 2005).

Numerical calculations were performed for liquid pseudophase, a non-Newtonian liquid, which was a 1% aqueous sucrose solution with yeast suspension with the following physical parameters: density $\rho = 1004 \text{ kg/m}^3$, surface tension $\sigma = 0.072 \text{ N/m}$, apparent viscosity η determined for a pseudoplastic liquid based on the equation:

$$\eta = K \cdot \dot{\gamma}^{m-1}, \quad (2)$$

where $K=0.003528 \text{ Pa s}^m$, $m=0.8781$. The non-Newtonian character of the pseudophase liquid (the system containing the aqueous solution of sucrose and yeast suspension) was determined on the basis of own studies carried out with the rheometer Haake RT 10. The simulations were made for a non-Newtonian liquid (shear thinning liquid)—using the Ostwald de Waele model. The addition of yeast suspension to the sucrose aqueous solution results in a change of the rheological properties of the Newtonian liquid to the non-Newtonian liquid.

Numerical calculations were carried out for three stirrer speeds $n=8, 10, 12 \text{ 1/s}$, and two gas flow rates $V_g=1.67 \times 10^{-4} \text{ m}^3/\text{s}$ and $V_g=3.33 \times 10^{-4} \text{ m}^3/\text{s}$.

Results and discussion

The results of numerical simulations for a Rushton (RT) or Smith (CD6) turbine stirrer were developed in the form of vectors of liquid velocity and speed contours. Figure 3 shows

the vector field of liquid velocity in the axial plane for a gas–liquid pseudophase system in a bioreactor with Rushton turbine (Fig. 3a) or Smith turbine stirrer (Fig. 3b). In both cases, two circulation loops can be observed: smaller—under the stirrer and larger—above the stirrer suspension, according to the radial–axial circulation of the liquid produced in the mixer.

Radial and axial contours of liquid phase velocity in the bioreactor in the gas–liquid pseudophase system, obtained for the stirrer speed $n=8 \text{ 1/s}$, $n=10 \text{ 1/s}$ and $n=12 \text{ 1/s}$ and gas flow rate $V_g=1.67 \times 10^{-4} \text{ m}^3/\text{s}$ and $V_g=3.33 \times 10^{-4} \text{ m}^3/\text{s}$, are shown on Figs. 4, 5 and 6. The comparison of axial contours of liquid velocities in the gas–liquid pseudophase system (Fig. 4) shows that the most intense mixing takes place in the area of the stirrer, which reveals the influence of the shape of the stirrer blades on the liquid circulation. These relationships confirm liquid velocity distributions presented in Figs. 5 and 6 made in radial planes located under the stirrer ($z/H=0.25$, Figs. 5a, 6a), at the height of the stirrer ($z/H=0.33$; Figs. 5b, 6b) and above the stirrer ($z/H=0.42$,

Fig. 3 Vectors of liquid velocity in a gas–liquid pseudophase system in a bioreactor **a** Rushton turbine stirrer (RT); **b** Smith turbine stirrer (CD 6); $n=10 \text{ 1/s}$; $V_g=3.33 \times 10^{-4} \text{ m}^3/\text{s}$

Fig. 4 Axial contours of liquid velocity in a gas–liquid pseudophase system; **a, b** bioreactor with Rushton turbine stirrer; **c** Smith turbine stirrer; $V_g=3.33 \times 10^{-4} \text{ m}^3/\text{s}$; **a** $n=8 \text{ 1/s}$; **b, c** $n=12 \text{ 1/s}$

Fig. 5 Radial contours of liquid velocity in a gas–liquid pseudophase system; Rushton turbine stirrer; $n=10$ 1/s; $V_g=1.67 \times 10^{-4}$ m³/s; **a** $z/H=0.25$; **b** $z/H=0.33$; **c** $z/H=0.42$

Fig. 6 Radial contours of liquid velocity in a gas–liquid pseudophase system; Smith turbine stirrer; $n=10$ 1/s; $V_g=1.67 \times 10^{-4}$ m³/s; **a** $z/H=0.25$; **b** $z/H=0.33$; **c** $z/H=0.42$

Figs. 5c, 6c). In the case of bioreactor with Rushton turbine stirrer, the area characterized by the most intense mixing is placed symmetrically around the stirrer (Fig. 5), while in the bioreactor with Smith turbine stirrer, the areas with the highest liquid phase velocity are located behind the stirrer blades (Fig. 6). In both cases, the liquid velocities in the bioreactor decrease toward the surface of the liquid. The area with the highest mixing intensity for a bioreactor with a Rushton turbine stirrer increases with increasing stirrer speed (Fig. 4a, b).

Axial and radial contours of the gas hold-up φ for different speeds of the stirrer n and different gas flow rates V_g are compared in Figs. 7, 8, 9, 10 and 11. In the case of the bioreactor with Rushton turbine stirrer (Fig. 7a–c), the axial contours of the gas hold-up φ are more symmetrical than those obtained for the bioreactor with Smith turbine stirrer (Fig. 7d). A better dispersion of the gas bubbles in the system is obtained increasing the speed of the stirrer n , assuming a constant value of the gas flow V_g , (Fig. 7a,

b). Areas with the largest values of the gas hold-up φ are placed symmetrically to the vertical axis of the bioreactor and coincide with the central points of the circulation loops (Fig. 7a–c). These areas increase, assuming $V_g=\text{const}$, with the increase of stirrer speed (Fig. 7a, b). In the case of bioreactor with Rushton turbine stirrer, assuming a constant value of the stirrer speed n , an increase in the gas flow V_g in the system improves the symmetry of the distribution of the gas hold-up φ (Fig. 7b, d).

Radial contours of the gas hold-up φ obtained for the bioreactor with Rushton turbine stirrer, for three axial coordinates $z/H=0.25$; 0.33 and 0.42, are compared in Figs. 8, 9, 10 and 11. Distinct differences in the values of the proportion of gas hold-up φ are visible for the coordinate $z/H=0.25$ (zone under the stirrer). For this coordinate, a rosette-shaped area is visible between the gas distributor and the stirrer, with a minimal gas hold-up φ . The highest values of the gas hold-up φ , for $z/H=0.25$ are located near the baffles (Fig. 8a) and near the walls of the tank (Figs. 9a, 10, 11a).

Fig. 7 Axial contours of the gas hold-up φ ; **a–c** bioreactor with Rushton turbine stirrer; **d** bioreactor with Smith turbine stirrer: **a** $n=8$ 1/s; $V_g=3.33 \times 10^{-4}$ m³/s; **b, d** $n=10$ 1/s; $V_g=3.33 \times 10^{-4}$ m³/s; **c** $n=10$ 1/s; $V_g=1.67 \times 10^{-4}$ m³/s

Fig. 8 Radial contours of the gas hold-up φ ; bioreactor with Rushton turbine stirrer; $n=8$ 1/s; $V_g=3.33 \times 10^{-4}$ m³/s; **a** $z/H=0.25$; **b** $z/H=0.33$; **c** $z/H=0.42$

Fig. 9 Radial contours of the gas hold-up φ ; bioreactor with Rushton turbine stirrer; $n = 10$ 1/s; $V_g = 3.33 \times 10^{-4}$ m³/s; **a** $z/H = 0.25$; **b** $z/H = 0.33$; **c** $z/H = 0.42$

Fig. 10 Radial contours of the gas hold-up φ ; bioreactor with Rushton turbine stirrer; $n = 12$ 1/s; $V_g = 3.33 \times 10^{-4}$ m³/s; **a** $z/H = 0.25$; **b** $z/H = 0.33$; **c** $z/H = 0.42$

Fig. 11 Radial contours of the gas hold-up φ ; bioreactor with Rushton turbine stirrer; $n = 10$ 1/s; $V_g = 1.67 \times 10^{-4}$ m³/s; **a** $z/H = 0.25$; **b** $z/H = 0.33$; **c** $z/H = 0.42$

Fig. 12 Comparison of the value of the gas hold-up ϕ obtained as part of experimental and numerical studies; 1— $n=8$ 1/s, $V_g=3.33 \times 10^{-4}$ m³/s; 2, 5— $n=10$ 1/s, $V_g=1.67 \times 10^{-4}$ m³/s; 3, 6— $n=10$ 1/s, $V_g=3.33 \times 10^{-4}$ m³/s; 4, 7— $n=12$ 1/s, $V_g=3.33 \times 10^{-4}$ m³/s

With an increase of the speed of the stirrer n ($V_g = \text{const}$), the area with the lowest gas hold-up ϕ is reduced (Figs. 8a, 9a).

In all the analyzed cases, characteristic gas caverns were found behind the stirrer blades (Figs. 8b, 9, 10, 11b). The size of these caverns decreases with the increase in the speed of the stirrer n (assuming $V_g = \text{const}$; Figs. 8b, 9, 10b) and increases with the increase in V_g (assuming $n = \text{const}$; Figs. 9b, 11b).

The average values of the gas hold-up ϕ_{num} integrated in the entire volume of the bioreactor were compared with the average values of the gas hold-up ϕ_{exp} obtained as part of my experimental research and are shown in Fig. 12. On this basis, quite good consistency of numerical and experimental values of the share of gas retained in the liquid can be observed.

Conclusions

On the basis of numerical tests, it was found that:

1. The most intense mixing takes place in the area of the stirrer, and the influence of the shape of stirrer blades on the liquid circulation is revealed. In the case of bioreactor with Rushton turbine stirrer, the area characterized by the most intense mixing is placed symmetrically around the stirrer. In the bioreactor with Smith turbine stirrer, the areas with the highest liquid phase velocities are behind the stirrer blades. In both cases, the liquid velocities in the bioreactor decrease toward the free liquid surface.
2. A better dispersion of the gas bubbles in the system is obtained increasing the speed of the stirrer n , assuming a constant value of the gas flow V_g . The areas with the largest gas hold-up ϕ are placed symmetrically to the vertical axis of the bioreactor and coincide with

the central points of the circulation loops. These areas increase, assuming $V_g = \text{const}$, with an increase in the stirrer speed.

3. In all the analyzed cases, characteristic gas caverns were found behind the stirrer blades. The size of these caverns decreases with an increase in the speed of the stirrer n (assuming $V_g = \text{const}$) and increases with an increase in V_g (assuming $n = \text{const}$).

Open Access This article is distributed under the terms of the Creative Commons Attribution 4.0 International License (<http://creativecommons.org/licenses/by/4.0/>), which permits unrestricted use, distribution, and reproduction in any medium, provided you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons license, and indicate if changes were made.

References

- Ahmed SU, Ranganathan P, Pandey A, Sivaraman S (2010) Computational fluid dynamics modelling of gas dispersion in multi impeller bioreactor. *J Biosci Bioeng* 109:588–597. <https://doi.org/10.1007/j.jbiosc.2009.11.014>
- ANSYS (2015) ANSYS CFX-Solver Theory Guide, Release 15.0. ANSYS, Inc
- Delafosse A, Collignon M-L, Calvo S, Delvigne F, Crini M, Thonart P, Toye D (2014) CFD-based compartment model for description of mixing in bioreactors. *Chem Eng Sci* 106:76–85. <https://doi.org/10.1016/j.ces.2013.11.033>
- Devi TT, Kumar B (2014) Effects of superficial gas velocity on process dynamics in bioreactors. *Thermophys Aeromech* 21:365–382. <https://doi.org/10.1134/S0869869431403010X>
- Frank T, Zwart PJ, Shi JM, Krepper E, Lucas D, Rohde U (2005) Inhomogeneous MUSIG model—a population balance approach for polydispersed bubbly flows. In: International conference “nuclear energy for new Europe 2005”, Bled
- Gelves R, Dietrich A, Takors R (2014) Modeling of gas-liquid mass transfer in a stirred tank bioreactor agitated by Rushton turbine or a new pitched blade impeller. *Bioprocess Biosyst Eng* 37:365–375. <https://doi.org/10.1007/s00449-013-1001-8>
- Gimbun J, Rielly CD, Nagy ZK (2009) Modelling of mass transfer in gas-liquid stirred tanks agitated by Rushton turbine and CD-6 impeller: a scale-up study. *Chem Eng Res Des* 87:437–451. <https://doi.org/10.1016/j.cherd.2008.12.017>
- Jaworski Z (2005) Numeryczna mechanika płynów w inżynierii chemicznej i procesowej. Akademicka Oficyna Wydawnicza, Warszawa, p 2005
- Joshi JB, Nere NK, Rane ChV, Murthy BN, Mathpati ChS, Patwardhan AP, Ranade VV (2011) CFD simulation of stirred tanks: comparison of turbulence models. Part I: radial flow impellers. *Can J Chem Eng* 89:23–82. <https://doi.org/10.1002/cjce.20446>
- Kaiser SC, Eibl R, Eibl D (2011) Engineering characteristics of a single-use stirred bioreactor at bench-scale: the Mobius Cell Ready 3L bioreactor as a case study. *Eng Life Sci* 11:359–368. <https://doi.org/10.1002/elsc.201000171>
- Kerdouss F, Bannari A, Proulx P, Bannari R, Skrga M, Labrecque Y (2008) Two-phase mass transfer coefficient prediction in stirred vessel with a CFD model. *Comput Chem Eng* 32:1943–1955. <https://doi.org/10.1016/j.compchemeng.2007.10.010>

- Knopkar AR, Ranade VV (2006) CFD simulation of gas-liquid stirred vessel: VC, S33, and L33 flow regimes. *AIChE J* 52:1654–1672. <https://doi.org/10.1002/aic.10762>
- Musiał M, Karcz J, Cudak M (2014) Zastosowanie metody CFD do analizy hydrodynamiki w mieszalniku z przegrodami i mieszadłem CD 6. *Przemysł Chemiczny* 93:1599–1603. <https://doi.org/10.12916/przemchem.2014.1599>
- Musiał M, Cudak M, Karcz J (2017) Numerical analysis of momentum transfer processes in a mechanically agitated air-biophase-liquid system. *Chem Process Eng* 38(3):465–475. <https://doi.org/10.1515/cpe-2017-0036>
- Oniscu C, Galaction A-I, Cascaval D, Ungureanu F (2002) Modeling of mixing in stirred bioreactors 2. Mixing time for non-aerated broths. *Biochem Eng J* 12:61–69 (pii:S1369-703X(02)00042-6)
- Podgórska W (2006) Rozpad i koalescencja kropeł w intermitentnym polu burzliwym. Oficyna Wydawnicza Politechniki Warszawskiej, Warszawa
- Ranade VV (2002) Computation flow modeling for chemical reactor engineering. Academic, San Diego
- Ranganathan P, Sivaraman S (2011) Investigations on hydrodynamics and mass transfer in gas-liquid stirred reactor using computational fluid dynamics. *Chem Eng Sci* 66:3108–3124. <https://doi.org/10.1016/j.ces.2011.03.007>
- Sommerfeld M, Decker S (2004) State of the art and future trends in CFD simulation of stirred vessel hydrodynamics. *Chem Eng Technol* 27(3):215–224. <https://doi.org/10.1002/ceat.200402007>
- Vrábel P, van der Lans RGJM, Luyben KChAM, Boon L, Nienow AW (2000) Mixing in large-scale vessels stirred with multiple radial or radial and axial up-pumping impellers: modeling and measurements. *Chem Eng Sci* 55:5881–5896
- Wang H, Jia X, Wang X, Zhou Z, Wen J, Zhang J (2014) CFD modeling of hydrodynamic characteristics of gas-liquid two-phase stirred tank. *Appl Math Model* 38:63–92. <https://doi.org/10.1016/j.apm.2013.05.032>
- Xia J-Y, Wang Y-H, Zhang S-L, Chen N, Yin P, Zhong Y-P, Chu J (2009) Fluid dynamics investigation of variant impeller combinations by simulation and fermentation experiment. *Biochem Eng J* 43:252–260. <https://doi.org/10.1016/j.bej.2008.10.010>