CHEMICAL ENGINEERING TRANSACTIONS Volume 17, 2009
Editor Sauro Pierucci
Copyright © 2009, AIDIC Servizi S.r.l., ISBN 978-88-95608-01-3 ISSN 1974-9791
DOI: 10.3303/CET0917099

Application of the CFD method for modelling of floating particles suspension

Joanna Karcz, Lukasz Kacperski, Marcelina Bitenc
Szczecin University of Technology, Dept. of Chem. Eng.
al. Piastow 42, PL-71-065 Szczecin, Poland

The results of the CFD simulations presenting both distributions of the fluid velocity and the concentration of floating particles in the liquid agitated by means of the up-pumping pitched blade turbine in the baffled vessel have been presented in the paper. Preprocessor MixSim 2.0 was used to create computational grid with multiple reference frames (MRF). Version 6.2.16 of the FLUENT was used in the simulations of the turbulent fluid flow in the agitated vessel. The computations were performed, assuming $k-\omega$ model of turbulence. Distributions of the fluid velocity vectors and concentration of the floating particles in the liquid agitated, obtained from the numerical simulations, were analysed.

Introduction

Literature data show (Etchells (2001), Ozcan-Taskin and Wei (2003), Ozcan-Taskin (2006), Karcz and Mackiewicz (2006, 2007)) that most often the effects of different parameters on the production of a suspension of the floating particles in an agitated vessel have been studied experimentally. CFD simulations for such systems were described by Ozcan-Taskin et al. (2003), Kantarci et al. (2005) and Murthy et al. (2007), only. Montante and Bakker (2004) carried out numerical modeling for the suspended heavier particles than liquid phase.

In this paper, the results of the CFD simulations presenting both distributions of the fluid velocity and the concentration of floating particles in the liquid agitated by means of the up-pumping pitched blade turbine in the baffled vessel have been presented.

Range of the simulations

Preprocessor MixSim 2.0 was used to create computational grid with multiple reference frames (MRF). Using this preprocessor, geometry of the vessel, type of the impeller and baffles (Fig. 1a), as well as geometry and density of the numerical grid (Fig. 1b) and part of the boundary conditions were defined. Numerical grid with the 260 000 of computational cells was generated. Version 6.2.16 of the FLUENT was used in the simulations of the turbulent fluid flow in the agitated vessel. The computations

Please cite this article as: Karcz J., Kacperski L. and Bitenc M., (2009), Application of the cfd method for modelling of floating particles suspension, Chemical Engineering Transactions, 17, 591-596 DOI: 10.3303/CET0917099

were performed, assuming $k-\omega$ model of turbulence and the *mixture* multiphase model for the steady-state fluid flow.

Numerical simulations were carried out for a baffled agitated vessel of inner diameter $D = 0.295$ m, filled with a liquid up to height $H = D$. Up-pumping pitched six-blade turbine (PBT↑) of diameter $d = 0.33D$ was located on the shaft at the distance $h = 0.67H$ from the flat bottom of the vessel. The vessel was equipped with four baffles of full length $L = H$ and width $B = 0.1D$. The simulations were performed for constant values of both impeller speeds n and mean concentration x_m of floating particles in the liquid. The computations were carried out for distilled water as a liquid phase and for the polyethylene particles of mean diameter $d_p = 4$ mm and density $\rho_p = 952$ kg/m^3.

Results and discussion

The simulations of the velocity fields and distribution of solids concentration in a suspension of the floating particles in the agitated vessel were carried for non-standard position of the impeller in the vessel ($h = 0.67H$). Standard value of the parameter h is equal to $0.33H$. It can be expected that the impeller location near free surface of the fluid with the non-dispersed floating particles will improve efficiency of the particles suspending. The results of the CFD simulations have been presented graphically as the distributions of the fluid velocity vectors (Fig. 2) and the concentration distributions of the floating particles (Fig. 3) in a liquid at the radial and axial cross sections of the agitated vessel, respectively.

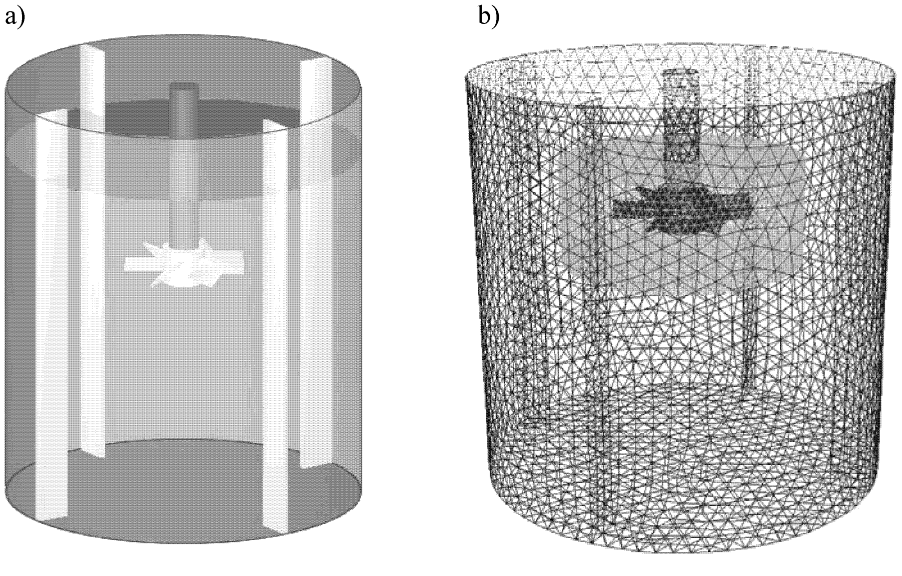
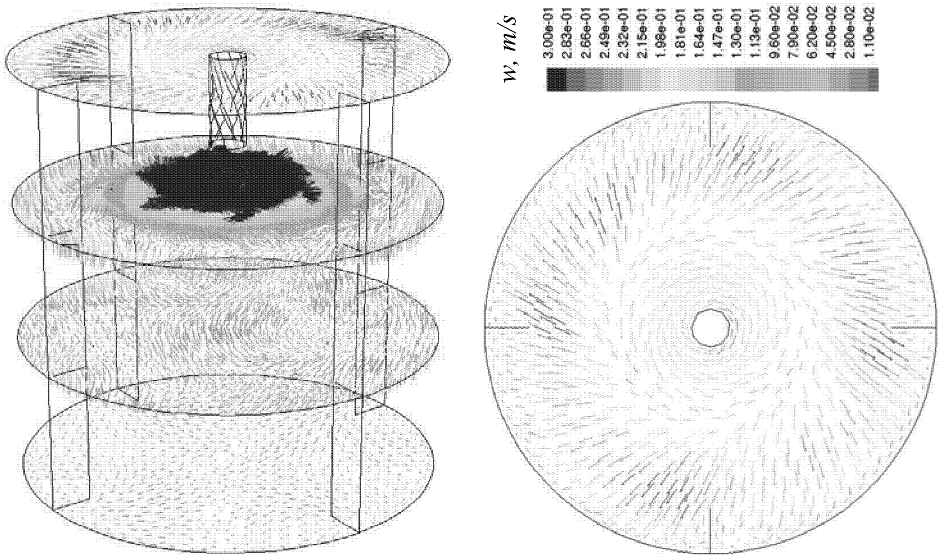


Fig. 1. a) Geometry of the agitated vessel with baffles and PBT turbine; b) View of the numerical grid with multiple reference frame (MRF), 260 thousands of cells

a) b)



c)

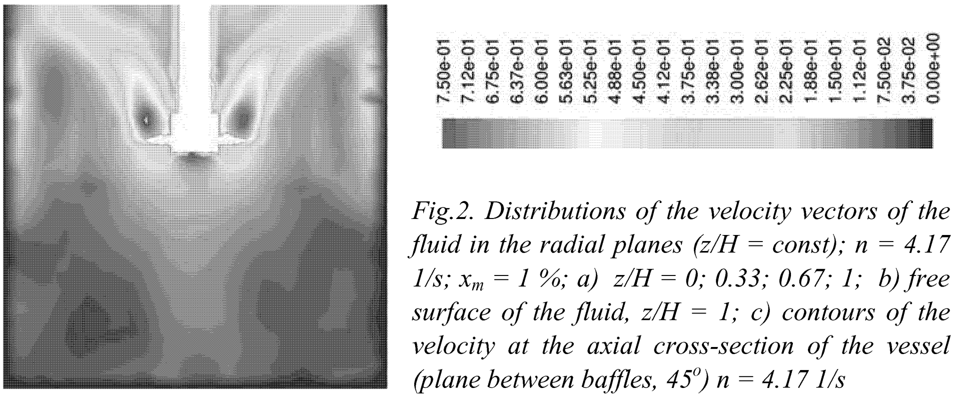


Fig.2. Distributions of the velocity vectors of the fluid in the radial planes (z/H = const); n = 4.17 1/s; x_m = 1 %; a) z/H = 0; 0.33; 0.67; 1; b) free surface of the fluid, z/H = 1; c) contours of the velocity at the axial cross-section of the vessel (plane between baffles, 45^o) n = 4.17 1/s

Fig. 2 shows four radial profiles of the velocity vectors of the fluid obtained for a given agitator speed n = 4.17 1/s, mean concentration of the particles x_m = 1 % and for constant level of the axial dimensionless coordinate z/H (z/H = const, z – axial coordinate). In Fig. 2c, contours of the velocity at the axial cross-section of the vessel are shown. The highest simulated intensity of the fluid circulation generated by up-pumping pitched blade turbine in the vessel is obtained in the surroundings of the impeller and in the region above the impeller. The results of the numerical simulations are in agreement with the observations of the fluid flow in the baffled agitated vessel equipped with a up-pumping pitched blade turbine.

In Fig. 3, axial distributions of the dimensionless concentration x/x_m as a function x/x_m = f(z/H) are compared for a given impeller speed n, mean concentration x_m of the particles in the liquid agitated and for dimensionless radial coordinate $r^* = r/R$ (where: z – axial coordinate, r – radial coordinate, $R = D/2$ – radius of the vessel, x, x_m – local and mean value of the particles concentration, volume fraction of the particles in the liquid, m^3/m^3). The results are shown for the mid-plane of the axial-cross section of the vessel located between both opposite baffles (i. e. angle between a baffle and axial cross-section plane is equal to 45°). The coordinate $r^* = 0.34$ describes the nearest distance to the impeller, whereas the $r^* = 0.88$ corresponds to the longest one (and the nearest to the vessel wall).

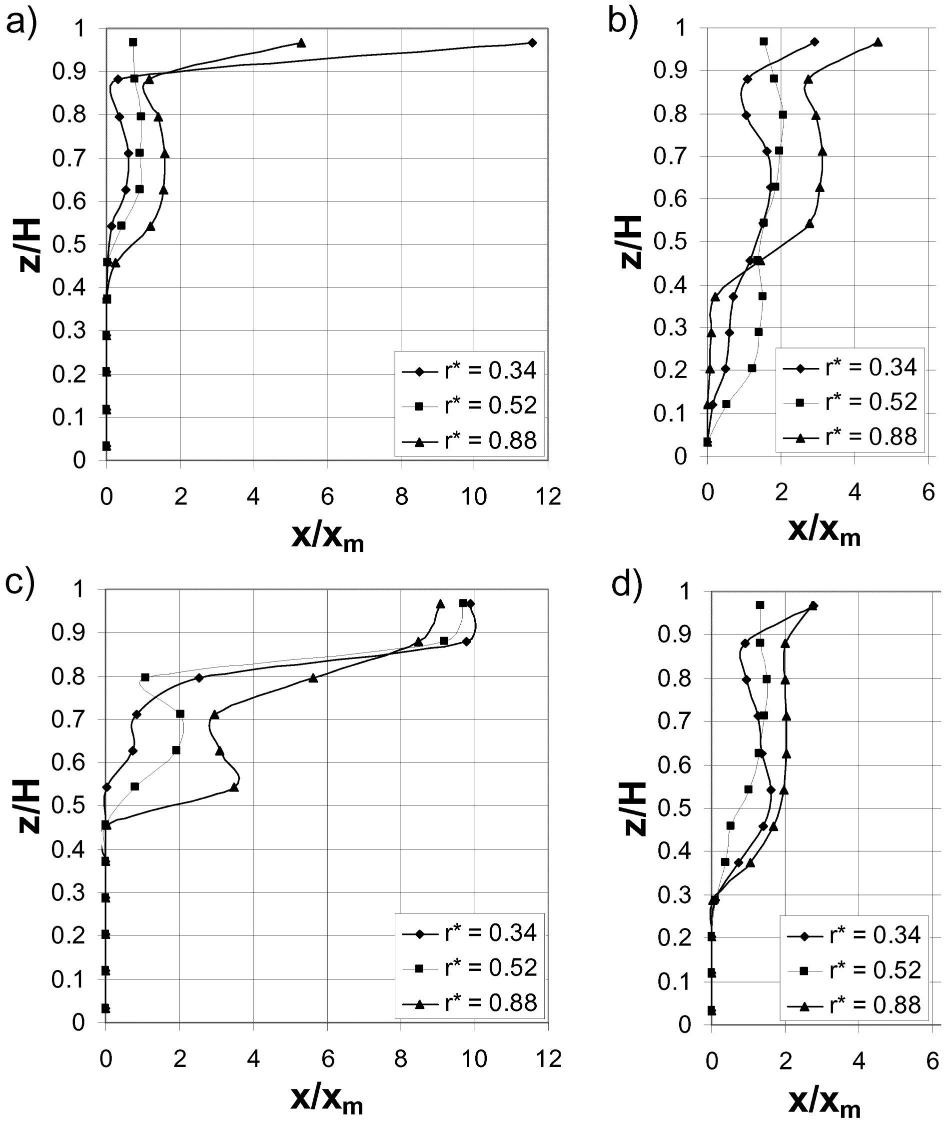


Fig. 3. Axial distributions of the dimensionless concentration ($x/x_m = f(z/H)$) for different dimensionless radial coordinate $r^* = r/R$); cross-section of the axial plane located between both opposite baffles (angle 45°between axial mid-plane and a baffle); a) $n = 2.5$ 1/s; $x_{m.} = 1$ %; b) $n = 4.17$ 1/s; $x_{m.} = 1$ %; c) $n = 2.5$ 1/s; $x_m = 10$ %; d) $n = 4.17$ 1/s; $x_{m.} = 10$ %

Figs. 3a and 3b represent the particles distributions for a low mean concentration x_m = 1 %. For low agitator speeds n = 2.5 1/s (Fig. 3a), particles are very poorly distributed in the bulk of the liquid. Almost all the particles are collected on the free surface of the liquid phase and only small part of the particles is dispersed in the region of the impeller acting, i. e. within the range of the axial coordinate $0.5 < z/H < 0.9$. Practically, the particles do not reach the zone below the impeller and the bottom of the vessel. Immersion of the floating particles improves with the increase of the agitator speed n (Fig. 3b, n = 4.17 1/s). Distributions of the particles concentration shown in Fig. 3b are more uniform compared to the concentration profiles presented in Fig. 3a.

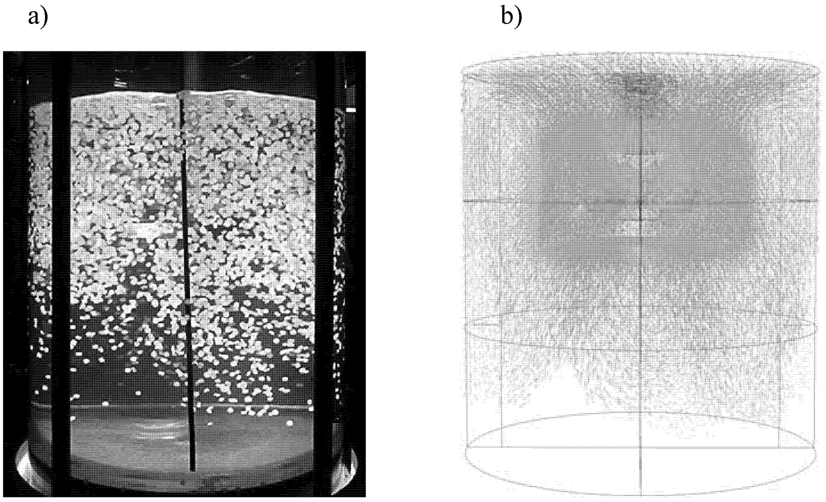


Fig. 4. Views of the particles concentration distributions; n = 2.5 1/s; $x_{m.}$ *= 1 %; a) experimental results (Mackiewicz (2008); b) numerically obtained results*

Axial distributions of the floating particles in the liquid agitated, obtained for the mean solids concentration x_m = 10 % and agitator speed n = 2. 5 1/s and n = 4.17 1/s, respectively, are given in Figs. 3c and 3d. Tendency for the axial profiles of the particles concentration is similar as for the distributions shown in Figs. 3a and 3b. However, higher concentration of the particles causes that regions without the solids are greater below of the impeller in the agitated vessel. The profiles of the particles concentration became more homogenous with the increase of the agitator speeds n (Fig. 3d).

Visualizations for numerically obtained distributions of the floating particles in the liquid are compared in Fig. 4 with the experimental results of the suspending such particles, identified on photographs by Mackiewicz (2008). This comparison shows that numerical and experimental distributions of the particles concentration are very similar. The results of the numerical simulations performed by Mackiewicz (2008) for assumed k - ε model of turbulence were also compared with the our results obtained for the k - ω model. On the basis of this comparison it can be suggested that better agreement of the experimental and numerical results for production of the suspension of floating particles can be obtained using k - ω model of turbulence in the numerical simulations.

Conclusions

From the numerical simulations, which were carried out for the suspension of floating particles produced in the baffled agitated vessel with the up-pumping pitched blade turbine located at nonstandard position ($h = 0.67H$), it can be stated that within the range of the performed numerical computations

- FLUENT 6.2 CFD software is very useful to analyze hydrodynamics of the such system.
- Qualitatively compared distributions of the concentration of floating particles in the liquid agitated, obtained from the numerical simulations and the experiments, are very similar.

References

Etchells A.W., *Mixing of floating solids*, (2001), Plenary Lecture, 4th Intern. Symp. on Mixing in Industrial Processes ISMIP 4, Toulouse, France, 14 - 16.05.2001.

Kantarci N., Ulgen K.O., and Borak F., 2005, A study on hydrodynamics and heat transfer in a bubble column reactor with yeast and bacterial cell suspensions, Canadian J. Chem. Eng., 83, 4, 764-773.

Karcz J. and Mackiewicz B., 2006, Suspending of floating solids in an agitated vesse*l*, Chem. Proc. Eng., 27, 1517-1533.

Karcz J. and Mackiewicz B., 2007, An effect of particles wettability on the draw down of floating solids in a baffled agitated vessel equipped with a high-speed impeller, Chem. Proc. Eng., 28, 661-672.

Mackiewicz B., PhD Thesis, Szczecin University of Technology, 2008.

Montante G. and Bakker A., 2004, Stirring up the phases in tall tank, Fluent News, 13, 2, 8-9.

Murthy B.N., Ghadge R.S. and Joshi J.B., 2007, CFD simulation of gas-liquid-solid stirred reactor: Prediction of critical impeller speed for solid suspension, Chem. Eng. Sci., 62, 7184-7195, doi: 10.1016/j.ces.2007.07.005.

Ozcan – Taskin G. and Wei H., 2003, The effect of impeller-to-tank diameter ratio on draw down of solids, Chem. Eng. Sci., 58, 2011-2022.

Ozcan – Taskin G., 2006, Effect of scale on the draw down of solids, Chem. Eng. Sci., 61, 2871-2879.