**CHEMICAL ENGINEERING TRANSACTIONS** Volume 24, 2011
Editor Sauro Pierucci
Copyright © 2011, AIDIC Servizi S.r.l., ISBN 978-88-95608-15-0 ISSN 1974-9791
DOI: 10.3303/CET1124240



# CFD Simulation of Solid Liquid Suspensions in Baffled Stirred Vessels Below Complete Suspension Speed

Alessandro Tamburini*, Andrea Cipollina, Giorgio Micale

Dipartimento di Ingegneria Chimica, Gestionale, Informatica, Meccanica - Università di Palermo
Viale delle Scienze - Edificio 6, 90128 Palermo, Italy – a.tamburini@unipa.it

Suspension of solid particles into liquids within stirred vessels is a complex phenomenon encountered in several industrial applications. Most researchers have focused their attention on the assessment of the minimum impeller speed ($N_{js}$) able to guarantee the suspension of all particles. Actually, in a number of industrial cases an impeller speed $N$ lower than $N_{js}$ is chosen as typical operating condition (Oldshue, 1983) as the resulting energy savings well counterbalance the loss of active interfacial area. Therefore, the assessment of the amount of unsuspended particles at any given impeller speed represents the key to quantify the economical advantage/disadvantage of adopting an impeller speed $N < N_{js}$. The present work focuses on the prediction of the fraction of suspended particles at rotational speeds $N < N_{js}$ by means of *Computational Fluid Dynamics* (*CFD*) for the case of a dense solid-liquid suspension in a baffled tank stirred by a standard Rushton turbine. An Eulerian-Eulerian Multi Fluid Model coupled with a standard k-ε turbulence model was adopted. The Multiple Reference Frame (*MRF*) approach was employed to simulate the rotating impeller. Experimental data collected by using the *Pressure Gauge Technique* (Brucato et al., 1997) were employed for the validation of the CFD model. A very good agreement between experimental data and CFD results was found.

## 1. Format And Type Fonts

The choice of design parameters ensuring an adequate solid suspension within a stirred vessel is still an open problem for design engineers. Many industrial applications involving transport phenomena between particles and a liquid in a stirred tank require the contacting interfacial area to be maximized as well as the power consumption to be minimized. The well known compromise between these two aspects generally leads to the adoption of the minimum impeller speed required for the complete off-bottom suspension of the particles. By the way, in a number of industrial cases a value of $N < N_{js}$ is chosen, as the loss of available interfacial area between particles and liquid is reasonably counterbalanced by decreased mechanical power requirements. Therefore, the knowledge of the amount of the suspended solids at different impeller speeds (*suspension curves*) is a topic of primary importance. So far, a lot of research efforts have been devoted to the evaluation of $N_{js}$, but only few works have dealt with the estimation of suspension curves: Brucato et al. (1997) and Micale et al. (2002) proposed a novel technique, namely the *Pressure Gauge Technique* (*PGT*), to assess the

Please cite this article as: Tamburini A., Cipollina A. and Micale G., 2011, Cfd simulation of solid liquid suspensions in baffled stirred vessels below complete suspension speed, Chemical Engineering Transactions, 24, 1435-1440
DOI: 10.3303/CET1124240

percentage of suspended solids in a stirred tank, at any given impeller speed, by means of pressure measurements on the tank bottom. Although such experimental approach is simple and reliable, *Computational Fluid Dynamics* (*CFD*) simulation is less onerous and time consuming, while providing valuable and detailed information. Only a few works dealt with the CFD prediction of the minimum impeller speed for complete solid suspension ($N_{js}$) (Kee & Tan, 2002; Murthy et al., 2007) but none of them was devoted to the issue of suspension curves. Particularly, Kee & Tan (2002) proposed a CFD approach to predict $N_{js}$ based on the evaluation of the simulated profiles of solid volume fractions for the layer of cells adjacent to the vessel floor. Conversely, Murthy et al. (2007) prediction method was based on the estimation of suspension quality via the standard deviation definition proposed by Bohnet and Niesmak (1980). The present work focuses on CFD simulations of incomplete-to-complete suspension regimes in stirred tanks with the aim of predicting suspension curves. Experimental data were obtained by *PGT* and successively used for validation of the CFD approach. It is worth noting that, for the first time, a CFD model dealing with partial suspension conditions is developed and validated with relevant experimental data.

## 2. Experimental information

The experimental system consisted of a cylindrical flat bottomed baffled tank with vessel diameter T=0.19m and total liquid height H=T. A standard six-bladed Rushton turbine with D=T/2 was used and set at a distance from the vessel bottom equal to 1/3 of H. Deionised water and glass ballottini were employed for the cases detailed in Table 1.

*Table 1: Features of the employed particles*

| Diameter [μm] | Density $\rho$ [kg/m$^3$] | Concentration $r_{\beta\_av}$ [-] | Max. Packing Factor $r_{\beta\_packed}$ [-] |
|---|---|---|---|
| 212-250 | 2500 | 11.9 v/v | 0.6 |
| 212-250 | 2500 | 5.95 v/v | 0.6 |
| 500-600 | 2500 | 11.9 v/v | 0.6 |

Full details on *PGT* apparatus and fundamentals are available in literature (Brucato et al.,1997; Micale et al., 2002).

## 3. Model and methods

All CFD simulations were carried out by using the commercial code *CFX4.4* (*AEA technology*). The adopted numerical approach for the simulation of the two-phase system was the Eulerian-Eulerian Multi Fluid Model where the two phases form an interpenetrating continua. The relevant continuity and momentum balance equations are reported below

$$\frac{\partial}{\partial t}(r_i\rho_i)+\vec{\nabla}\cdot\left(r_i\rho_i\vec{U}_i\right)=0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(r_i\rho_i\vec{U}_i\right)+\vec{\nabla}\bullet\left\{r_i\left[\rho_i\vec{U}_i\otimes\vec{U}_i-\left(\mu_i+\mu_t\right)\left(\vec{\nabla}\vec{U}_i+\left(\vec{\nabla}\vec{U}_i\right)^T\right)\right]\right\}=r_i\left(\vec{B}-\vec{\nabla}P\right)+\vec{M}_{i,j} \tag{2}$$

where $r$ is volumetric fraction, $\rho$ the density and $U$ the mean velocity, $i$ indicates the liquid or solid phase, $g$ the gravity acceleration, $\mu$ the viscosity, $P$ the pressure (the solid and the liquid phases share the same pressure field) and $M$ is the momentum inter-phase transfer term (clearly $M_{i,j}$=-$M_{j,i}$.) In the momentum equation of solid phase the turbulent viscosity is not included. Usually, in literature the two phases are considered to share the same value of the turbulent viscosity (computed by means of the homogeneous k-ε turbulence model) and satisfactory results can be obtained in a number of cases. Conversely, for the case here presented, simulations were carried out at different impeller speeds even at impeller speeds much lower than $N_{js}$ when many particles lie on the bottom: in such conditions assuming that particles resting on the vessel bottom possess turbulent viscosity is not acceptable and would generate a strong underestimation of the amount of particles resting on the vessel bottom. Interactions between the two phases were directly modelled only by inter-phase drag force terms within the momentum equations (*two-way coupling*). For both phases:

$$\vec{M}_{i,j}=C_{i,j}\left(\vec{U}_j-\vec{U}_i\right)=\left[\frac{3}{4}\frac{C_D}{d_p}r_\beta\rho_\alpha\left|\vec{U}_\beta-\vec{U}_\alpha\right|\right]\left(\vec{U}_j-\vec{U}_i\right) \tag{3}$$

where $C_{i,j}$ is inter-phase drag coefficient, $C_D$ is drag coefficient and $d_p$ is particle mean diameter. The subscripts $\alpha$ and $\beta$ refer to liquid and solid phase respectively. Other inter-phase momentum exchange terms give generally a much lower effect than drag force and are consequently neglected (Tamburini et al., 2009). Particle drag coefficient $C_D$ was considered variable in each cell in relation to the slip velocity between phases in accordance with Clift et al. correlation (1978). Free-stream turbulence influence upon drag coefficient (Brucato et al., 1998) was accounted for as well . Relevant adopted equations can be found in Tamburini et al. (2009). The well known k-ε turbulence model was adopted (relevant equations are not reported for the sake of brevity), but it was used to account only for the turbulence of the continuum phase in accordance with the former momentum equations: k and ε transport equations were solved only for the liquid phase and they were not taken into account for the particle phase. Anyway, turbulence interactions between phases are accounted for in the drag force term, which includes the influence of free-stream turbulence on particle drag coefficient $C_D$. Fluid–particle interactions as well as particle–particle interactions were modelled by the adoption of an excess solid volume correction (*ESVC*) algorithm (Tamburini et al., 2009) which avoids the solid volumetric fraction to exceed the maximum packing value physically allowed. The SIMPLEC algorithm was adopted to couple pressure and velocity. The hybrid-upwind discretization scheme was used for convective terms. The grid chosen for simulations encompasses 53,760 cells distributed as 24 × 70 × 32 along the azimuthal, axial and radial direction respectively. Given the symmetry of the system, only half of the tank was discretized and two periodic boundaries were imposed along the azimuthal direction. The grid is finer in the proximity of the impeller where the largest gradients of the flow variables are expected. This coarse grid was used in order to keep low computational requirements. However, a finer grid 48×140×64 (azimuthal×axial×radial) was also used to check whether results were affected by grid-dependence. Altogether this second grid encompassed 430,080 cells, i.e. 8 times finer than the coarse grid. At the beginning of each simulation all particles were placed on the tank bottom with a initial volume fraction of $0.60^{v/v}$, corresponding to the experimentally assessed maximum packing ($r_{\beta\text{-packed}}$). Steady state simulations were carried out by means of the Multiple Reference Frame (MRF) approach. 8,000 iterations were found to be more than sufficient to allow variable residuals to settle to very low and negligible values for all the investigated cases.

### 3.1 Unsuspended Solid Criterion (*USC*)

In accordance with PGT fundamentals, solids can be considered suspended if they contribute to increase the mean density of the suspension by lifting and moving away from the tank bottom. When solids lay still on the bottom they show their maximum volumetric fraction $r_{\beta\text{-packed}}$. Clearly, it is highly improbable for particles to be suspended while preserving this local value of $r_\beta$ , in fact it is likely that particles are distributed during the suspension process, thus showing a $r_\beta$ lower than $r_{\beta\text{-packed}}$. Therefore the Unsuspended Solid Criterion here proposed computes as unsuspended particles all the particles present in the domain regions where $r_\beta = r_{\beta\text{-packed}}$.

## 4. Results and discussion

Experimental results (plotted in different graphs together with their relevant CFD predictions for comparison purposes) show that the weight fraction of suspended solids ($x_{susp}$) decreases both with increasing solid loading and particle diameter as expected. Fig. 1 shows the prediction of the suspension curve by the MRF simulations. Despite the simplicity of the suspension criterion, simulation results appear to be in fair agreement with experimental data. Simulation data present the typical “sigmoidal” trend and are close to the corresponding experimental data even if there is a slight overestimation, especially at intermediate impeller rotational speeds. The good quality of CFD predictions is also showed by the contour plots of $r_\beta$: at low agitation speed (157RPM) most particles lie on the bottom; at 258RPM, many of them are suspended but without reaching the highest part of the tank; finally, at 493RPM the suspension is very close to be complete, with particles well distributed within the vessel. Notably, also Sliding Grid simulations were performed and no significant differences with respect to the MRF predictions were found.

Even in the case concerning a lower solid loading, depicted in Fig.2a, simulations results follow quite well the experimental data. Predicted fractions of suspended solids are close enough to the experimental ones even if a moderate overestimation was found especially at lower impeller speeds. At very low impeller speeds almost all particles rest on the bottom; increasing the agitation speed, the suspension phenomenon occurs suddenly and vigorously still at low values of $N$. This initial condition is very delicate from a computational point of view, thus leading to a great difficulty in simulating it properly. However the numerical approach presented here was found capable of reliably predicting the suspension curves even at a low solid loading. The effect of particle diameter on solid suspension curves was also investigated. Larger particle diameter (and consequently higher impeller speed necessary to achieve complete suspension conditions) will affect more significantly the drag coefficient due to the liquid free-stream turbulence. Observing data in Fig.2b, it can be stated that the simulations predict very well the sigmoidal trend, but they overestimate the fraction of suspended solids especially in correspondence of the suspension inception.

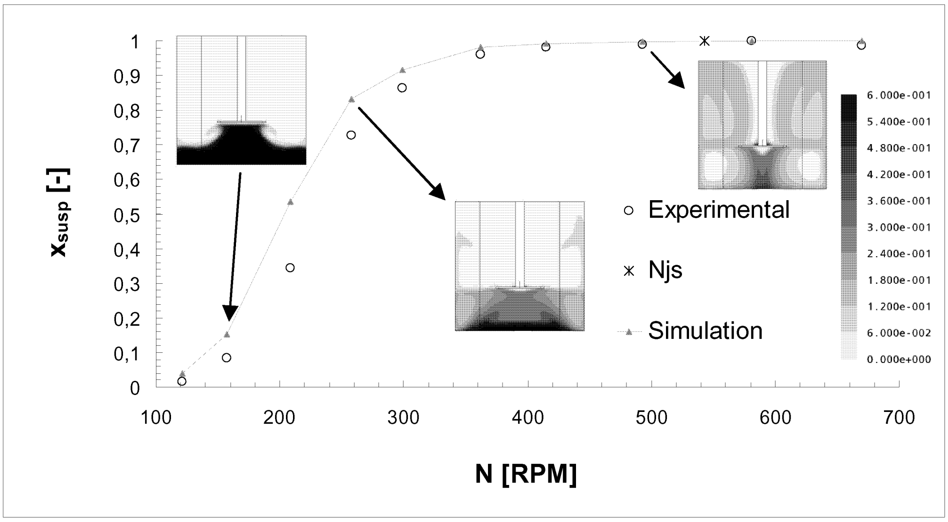


*Figure 1: Simulated vs experimental suspension curve and contours of solid volumetric fraction $r_\beta$ for the case of 212-250 μm ballottini with a solid loading of 11.9%$^{v/v}$.*

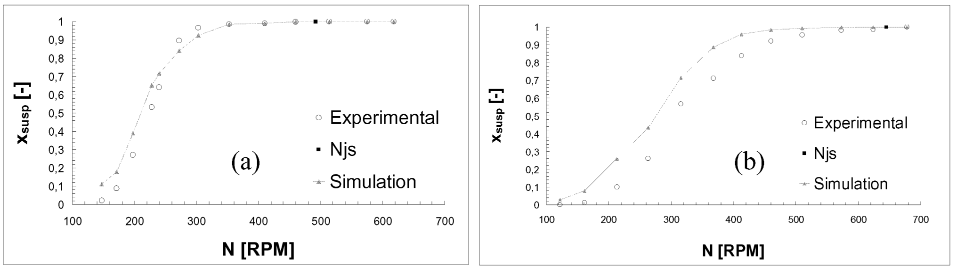


*Figure 2: MRF simulations versus experimental data: a)212-250 μm glass ballottini, $r_{\beta\_av}$ = 5.95%$^{v/v}$; b)500-600 μm glass ballottini, $r_{\beta\_av}$ = 11.9%$^{v/v}$*

Finally, a grid dependence check of computational results was performed for the case of 212-250 μm glass ballottini particles with mean solid concentration of 11.9%$^{v/v}$ and no significant differences were found.

## 5. Conclusions

CFD simulations of dense solid-liquid suspensions in a flat bottomed vessel stirred by a standard Rushton turbine were performed with the objective of predicting the percentage of unsuspended solids at different impeller speeds (covering partial to complete suspension regime). These simulations were carried out with the commercial code CFX4.4 by adopting the fully predictive Eulerian-Eulerian Multi Fluid Model along with the standard k-ε model in order to simulate the two-phase flow and the liquid phase turbulent flow, respectively. The Multiple Reference Frame (MRF) approach was used to simulate the impeller to baffles relative rotation. Inter-phase momentum exchange terms have been described through inter-phase drag forces. Effect of liquid free stream turbulence on drag coefficient was accounted for. Experimental data were purposely collected in order to validate simulation results. Influences of solid loading and particle diameter were investigated. Comparison between CFD predictions and experimental data showed a very good agreement, notwithstanding the simple modelling approach here adopted.

## References


Brucato A., Micale G. and Rizzuti L., 1997, Determination of the amount of unsuspended solid particles inside stirred tanks by means of pressure measurements. Rec. Progr. Genie des Proc 11, 3-10

Brucato, A., Grisafi, F. and Montante, G., 1998, Particle Drag Coefficients in Turbulent Fluids, Chemical Engineering Science 53, 3295-3314

Bohnet M. and Niesmak G., 1980, Distribution of solids in stirred suspensions. German Chemical Engineering 3, 57-65

Clift et al., Bubbles, 1978, Drops and particles, Academic Press, New York, San Francisco, London

Micale G., Grisafi F. and Brucato A., 2002, Assessment of particle suspension conditions in stirred vessels by means of pressure gauge technique. Trans IChemE, A, Chemical Engineering Research and Design 80, 893–902

Kee R.C.S. and Tan R.B.H., 2002, CFD simulation of solids suspension in mixing vessels. Canadian Journal of Chemical Engineering 80, 721-726

Murthy B.N., Ghadge R.S. and Joshi J.B., 2007, CFD simulations of gas-liquid-solid stirred reactor: prediction of critical impeller speed for solid suspension. Chemical Engineering Science 62, 7184-7195

Oldshue J. Y., 1983, in “Fluid Mixing Technology”, Chapter 5, McGraw-Hill, New York, NY

Tamburini A., Cipollina A., Micale G., Ciofalo M. and Brucato A., 2009, Dense solid–liquid off-bottom suspension dynamics: simulation and experiment. Chemical Engineering Research and Design 87, 587–597