Seventh International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
9-11 December 2009

PARTICLE VELOCITY AND CONCENTRATION PROFILES OF SAND – WATER SLURRY IN STIRRED TANK – MEASUREMENTS AND MODELLING

Jouni SYRJÄNEN[1], Sanna HAAVISTO[2], Antti KOPONEN[2] and Mikko MANNINEN[1]

[1] VTT Technical Research Centre of Finland, P.O. Box 1000, FI-02044 VTT, FINLAND
[2] VTT Technical Research Centre of Finland, Koivurannantie 1, P.O. Box 1603, FI-40101 Jyväskylä, FINLAND

ABSTRACT

Mixing of sand-water slurry at varying consistencies was studied in a laboratory scale cylindrical tank equipped with a 45° pitched blade turbine. The volume fractions of the solid phase investigated were 5% and 10%. Three-dimensional velocity profiles were measured utilizing ultrasound Doppler velocimetry along lines located circumferentially between two baffles of the tank .

Time dependent 3D CFD studies of the slurry flow in the tank were conducted with algebraic slip mixture model and full Eulerian multiphase model. Standard k-ε model was applied in turbulence modelling. Agreement of particle velocity components was generally good in the centre of the tank, while some deviation took place near the wall.

Keywords: CFD, multiphase, solid-liquid flow, stirred tank, ultrasound Doppler velocimetry

INTRODUCTION

Solid-liquid slurries are often found in industrial applications, for example in mineral processing, catalytic reactors, crystallization, polymerization, etc. The most frequently used reactor is a stirred tank, with one or more impellers and baffles to enhance the axial mixing. Two-phase mixing in a stirred reactor is a complicated process to model because of turbulent two phase flow with phase interactions (e.g Micheletti et al. 2003). Model validation is much needed, but also a demanding task.

In industrial applications, e.g. in flotation, the local solid concentrations are often high with volume fraction more than 10%. Because the measurement of velocity patterns and profiles of volume fractions is difficult in such high solids loads, earlier CFD model validation has most often been done at lower concentrations, typically around 1%. Recently, more attention has been paid on modelling also higher concentration slurries using Eulerian methods. Ochieng and Onyango (2008) compared different drag correlations by validating against experimental results for suspensions with volume fraction varying from 1% up to 20%. Kasat et al. (2008) used the Eulerian model for a 10% suspension. In particular, they studied the liquid phase mixing and the formation of a solid free layer on the top of the tank. Micale et al. (2004) examined the same phenomenon using both CFD and experimental observations in a slurry of about 10% solids volume fraction. Tamburini et al. (2009) carried out a similar validation and also compared different modifications of the drag function.

Measurements in high concentration slurries are scarce. Kasat et al. (2008) used Yamazaki's et al. (1986) experimental results for the concentration profile and Tamburini et al. (2009) compared to the measurements of Micheletti et al. (2003). The velocity profiles were not compared to experiments in any of the above studies.

In our earlier work (Haavisto et al., 2008) we employed the Ultrasound Doppler Velocimetry (UDV) to measure the particle velocity profiles in a 1% solid suspension. Good general agreement was obtained with CFD modelling using the algebraic slip mixture model and standard k-ε turbulence model. Here we extend the investigation, both UDV measurements and CFD simulations, to higher concentrations (5% and 10%). In addition, the full Eulerian multiphase model is compared to the algebraic slip model in 5% suspension.

EXPERIMENTAL

The schematic of experimental setup is shown in Figure 1. The mixing vessel is a plastic (PVC) flat-bottomed cylindrical tank of 0.59 m in diameter (T) fitted with three baffles (b=0.05 m) located symmetrically along the tank periphery. The slurry height in the tank is equal to the tank diameter providing a total volume of approximately 160 litres. A downward-pumping impeller with three 45° pitched blades and diameter (D) of 0.2 m is positioned at a clearance (C) of 0.1 m from the tank bottom. The width of the blades is 0.028 m.

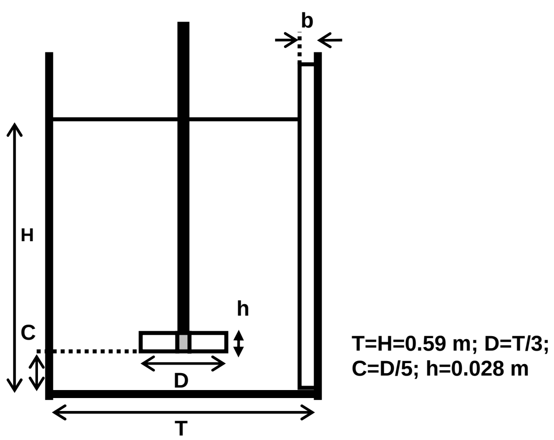

Figure 1: Schematic diagram of the tank geometry.

The impeller is rotated at speeds of 165 and 200 rpm when solid volume fractions were 5% and 10%, respectively. The density of the sand is 2100 kg/m^3 and its size distribution is presented in Haavisto et al. (2008). No significant particle breakage is assumed to take place during the mixing of the slurry.

The 3D velocity profiles in the tank are measured using a pulsed ultrasound Doppler velocimeter DOP2000, manufactured by Signal-Processing S.A., Switzerland, equipped with a custom made submersible probe. The measurement set-up and operating principle are described in Haavisto et al. (2008). The measurements are recorded along vertical lines on a plane located circumferentially midway between two baffles, shown in Figure 2. The velocity profiles are acquired piece-wise, starting from the tank surface down to the bottom. In each of the 8 – 10 measurement positions along a line, 6000 individual profiles are recorded.

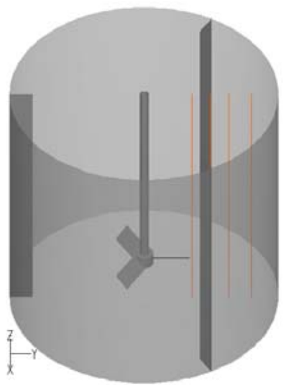

Figure 2: Geometry of the mixing tank. The radial measurement locations are illustrated with orange vertical lines.

CFD MODELLING

Modelling approach

In our previous study (Haavisto et al. 2008), the algebraic slip mixture model, described in detail by Manninen et al. (1996), was found suitable to modelling of sand – water slurry with 1% volumetric solid concentration and short particle relaxation time. In the present work, the application of the model to simulating higher particle concentrations in the same configuration was investigated. In addition, full Eulerian two-phase model was tested for comparison in 5% slurry case.

Turbulence was modelled using k-ε model with wall functions. A standard single particle model was used for particle drag and turbulent dispersion of particles was included in the model (Haavisto et al, 2008). The particle size used in the single particle size simulations was the Sauter mean diameter of the sand used in the experiments. In the mixture model simulation, the Ishii-Zuber model (Ishii and Zuber, 1979) was used for the mixture viscosity. Fluent version 6.3 software was used in the CFD simulations. One third of the vessel was included in the model for symmetric reasons. One impeller blade and one baffle were included in the computational domain. They were described as infinitely thin surfaces in the computational grid. The mesh which was also used in our earlier simulation (Haavisto et al. 2008) consisted of 40,000 cells. Time-dependent sliding mesh approach was used to model the impeller. The rotating part of the grid contained the impeller blade, and the stationary part contained the baffle.

Numerical Solution

Second-order accurate space discretization was used in the solution of the convective terms in the momentum, turbulence and particle phase volume fraction equations of the simulations. The standard pressure discretization of Fluent 6.3 was used in the momentum equation. The pressure-velocity coupling was solved with the SIMPLE algorithm, and a second-order time discretization scheme was used.

The time step size was chosen to correspond to a grid movement of 2.5° in the final stage of the calculation. This led to relatively small time steps and rather long computing times. The time step size ensures that the maximum Courant number in the computational domain is less than unity. The slip velocities and the flow velocities in the upper part of the tank would have allowed a larger time step.

In addition to monitoring residuals, volume integrals of velocity magnitude, dissipation of turbulent kinetic energy, and turbulent viscosity were tracked. The solution was considered converged to a semi-steady state when these values remained constant. No obvious convergence difficulties were noticed with the models used in the present study.

Simulated Cases

The volume fractions of sand in the simulations were 5% and 10%. The corresponding impeller rotational speeds were 165 rpm and 200 rpm, respectively. They implied tip velocities 1.72 m/s and 2.08 m/s, as well as impeller Reynolds numbers 110,000 and 130,000, thus making the flow fully turbulent. The particle size in the simulations was 114 μm.

RESULTS

Particle velocities and volume fractions in 5% slurry

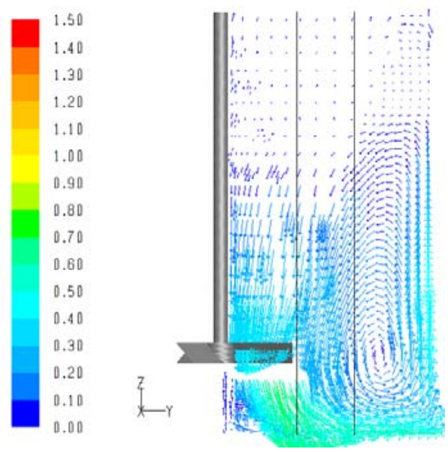

Figure 3: Instantaneous particle velocity vectors (m/s) calculated with mixture model. Radial measurement locations r=105 mm and r=185 mm are marked with vertical lines. 5% solid volume fraction.

The simulated instantaneous flow fields of the 5% slurry on a plane located circumferentially between two baffles are illustrated in Figures 3 and 5. The mixture model predicts a circulation loop which extends vertically up to three fourths of the tank height, while the one predicted by the Eulerian model is located in the lower half of the vessel.

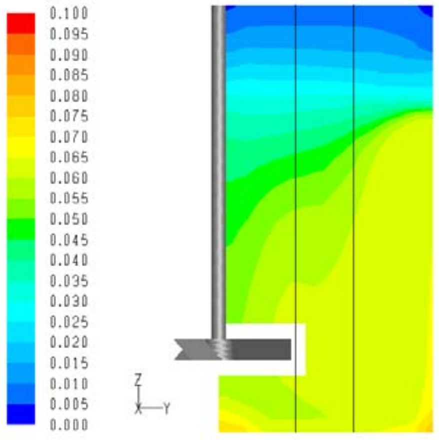

Figure 4: Time-averaged solid volume fraction calculated with mixture model. 5% solid volume fraction.

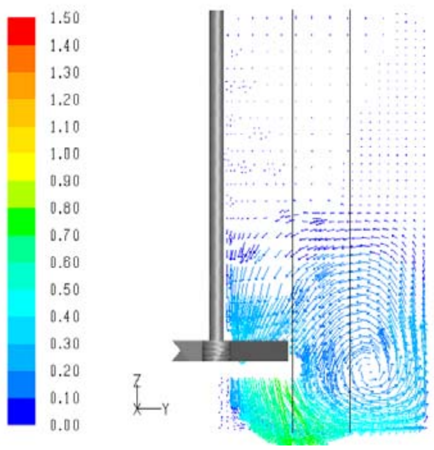

Figure 5: Instantaneous particle velocity vectors (m/s) calculated with Eulerian model. Radial measurement locations r=105 mm and r=185 mm are marked with vertical lines. 5% solid volume fraction.

Corresponding time averaged simulated particle volume fractions are shown in Figures 4 and 6. The low concentration layer in the upper part of the vessel is considerably larger in the Eulerian calculation. Volume containing solid concentration 1% or less comprises about 28% of the tank in the Eulerian prediction. In the mixture model simulation, it is only about 7% of the tank volume. The time averaged solid volume fraction is about 6% according to the mixture model calculation in the lower part of the tank. The Eulerian prediction is above 8%, as illustrated in Figure 7. Possible reasons for these differences are the explicit mixture viscosity used in the algebraic slip mixture model calculation, and the differences in the implementations of k-ε turbulence model in the mixture and Eulerian multiphase models in Fluent 6.3.

The predicted particle tangential, radial and axial velocity components with 5% particle volume fraction are compared with the experimental values along measuring lines on radial locations r=105 mm and r=185 mm in Figures 8 and 9. The radial velocity calculated with the mixture model agrees quite well with the measured in the location closer to the vessel centre but somewhat deviates from it in the location closer to the wall.

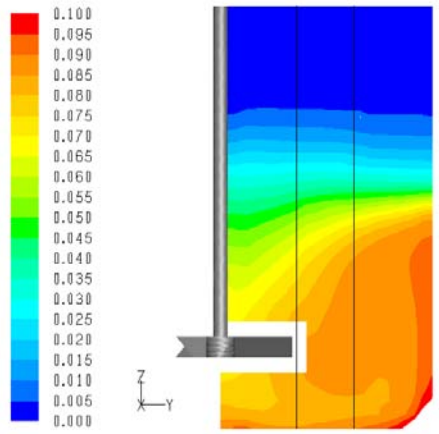

Figure 6: Time-averaged solid volume fraction calculated with Eulerian model. 5% solid volume fraction.

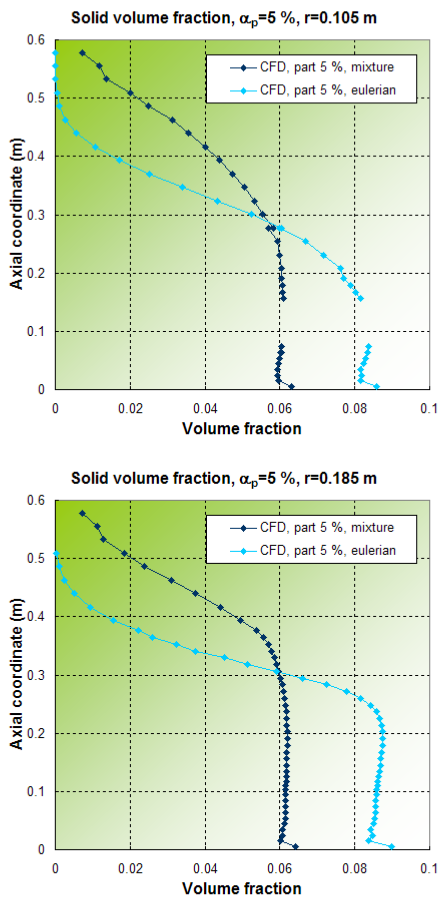

Figure 7: Time-averaged particle volume fractions on radial locations r=105 mm and r=185 mm. 5% solid volume fraction.

The agreement between the measured tangential component with the mixture model prediction is good in the centre of the tank, but somewhat poorer closer to the wall. The measurements suggest strong rotation above the vertical location of the impeller while the predicted maximum values of tangential velocity are found in the impeller discharge flow.

Similarly, the axial flow predicted by the mixture model is in good agreement with the measurement in the centre region of the vessel. The measured axial velocity component is directed more downwards than the calculated in the lower part of the vessel near the wall.

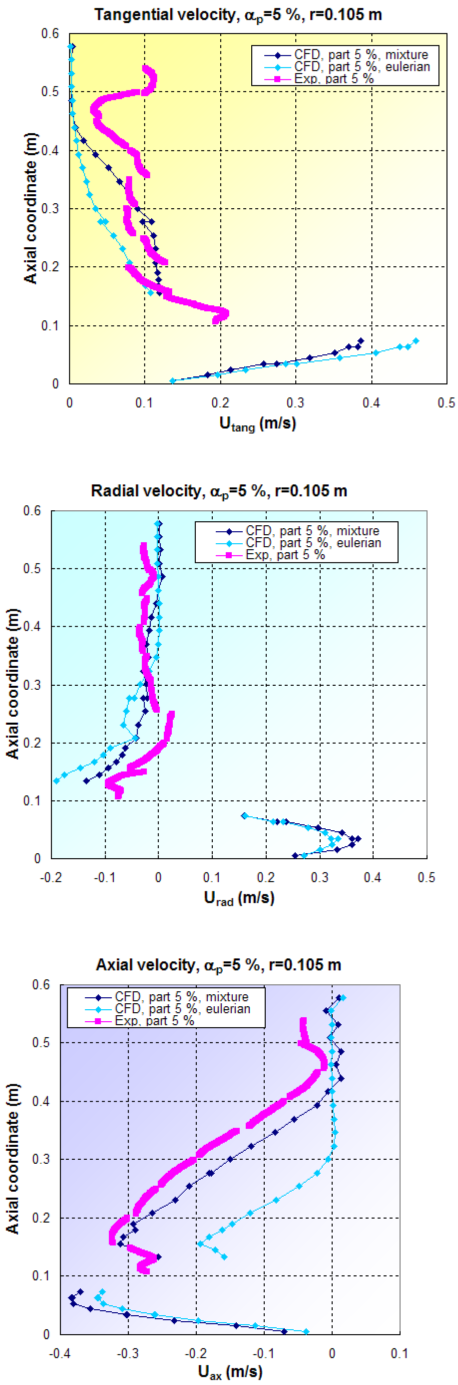

Figure 8: Time-averaged particle velocity components on radial location r=105 mm. 5% solid volume fraction.

In the Eulerian simulation of the 5% slurry, the smaller circulation pattern is reflected also in the velocity component profiles. They deviate clearly from mixture model simulation and experimental results. This concerns especially the axial particle velocity component, which indicates the height of the circulation loop and the extent of the low solids concentration zone in the upper part of the vessel.

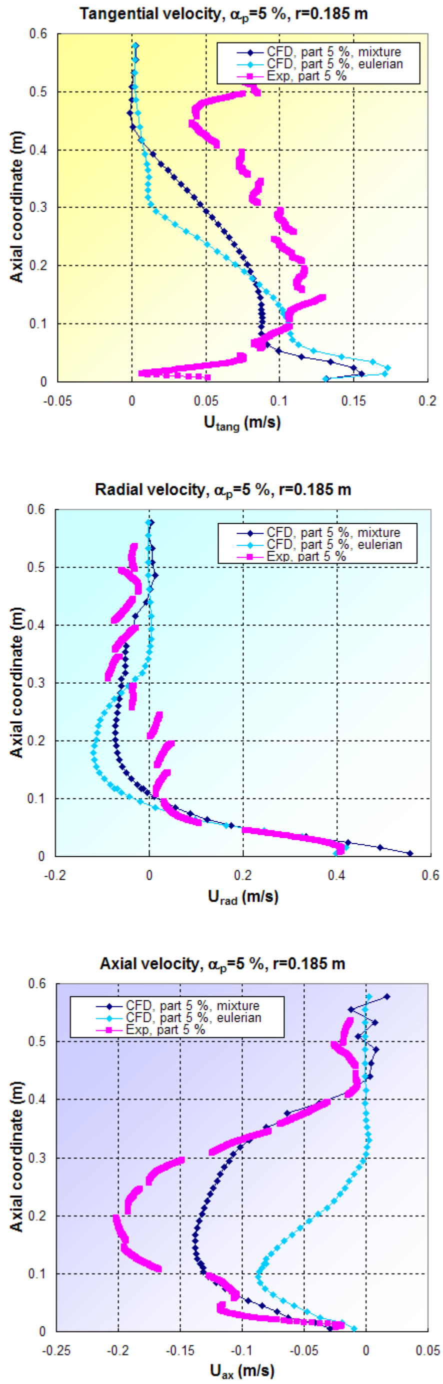

Figure 9: Time-averaged particle velocity components on radial location r=185 mm. 5% solid volume fraction.

Particle velocities in 10% slurry

The measurements in the 10% sand volume fraction experiment were made along three vertical lines with radial locations of 150 mm, 200 mm and 250 mm. The results along radial locations r=150 mm and r=250 mm are shown in Figures 10 and 11.

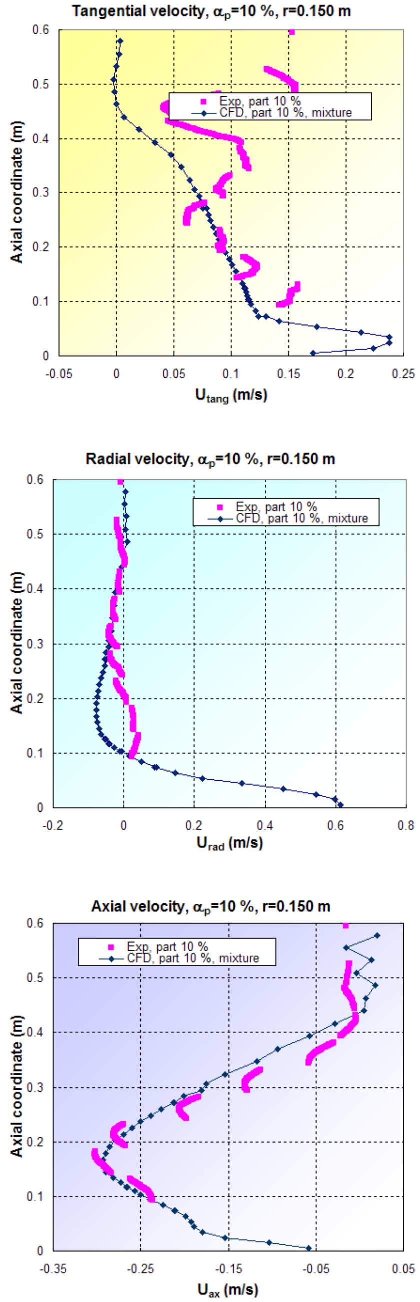

Figure 10: Time-averaged particle velocity components on radial location r=150 mm. 10% solid volume fraction.

The measured particle radial velocity towards the wall is somewhat higher than the calculated velocity just above the impeller. The measured tangential flow seems not to be very regular, while the predicted profile is rather smooth with maximum value in the discharge flow in the centre area of the tank. Closer to the wall, the predicted tangential velocity evens out vertically in the lower part of the vessel. The simulated axial velocity component agrees well with the measured in the centre of the tank, but deviates from it in the lower part of the tank in the vicinity of the wall.

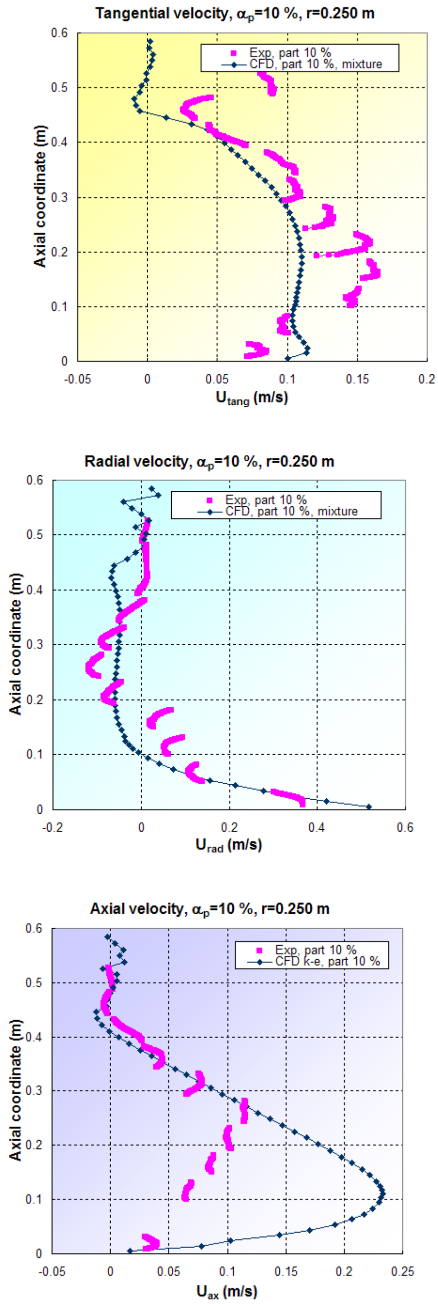

Figure 11: Time-averaged particle velocity components on radial location r=250 mm. 10% solid volume fraction.

CONCLUSION

Investigation of the applicability of ultrasound Doppler velocimetry measurement method and the algebraic slip mixture model simulation approach were extended to 5% and 10% consistency sand–water slurries after successfully applying them earlier in mixing vessel with 1% solid concentration. The agreement between the simulated and the measured particle velocities was found to be relatively good in the central region of the vessel. Near the wall, deviation of the results was observed with increasing solid concentration. Eulerian multiphase model was tested with parameters corresponding closely to those used with algebraic slip mixture model. Results obtained with it deviated from measurements more than mixture model predictions.

The simulations showed that the algebraic slip mixture model approach is applicable also in denser liquid-solid suspensions. The model can be further improved by refining the particle-fluid and particle-particle interaction models in dense suspensions. The UDV method was successfully applied to measuring particle velocity profiles in these suspensions.

ACKNOWLEDGEMENT

The authors gratefully acknowledge Tekes and industrial partners for financial support.

REFERENCES

MICHELETTI, M., NIKIFORAKI, L., LEE, K.C., and YIANNESKIS, M., (2003), "Particle concentration and mixing characteristics of moderate-to-dense solid-liquid suspensions", *Ind. Eng. Chem. Res*., **42**, 6236-6249.

HAAVISTO, S., SYRJÄNEN, J., KOPONEN, A. and MANNINEN, M., (2008), "UDV measurements and CFD simulation of two-phase flow in a stirred vessel", *Proc. 6th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries*, Trondheim, Norway, June 10-12.

KASAT, R,. KHOPKAR, A.R., RANADE, V.V., and PANDIT, A.B., (2008), "CFD simulation of liquid-phase mixing in solid-liquid stirred reactor", *Chem. Eng. Sci*., **63**, 3877-3885.

MICALE, G., GRISAFI, F., RIZZUTI, L., and BRUCATO, A., (2004) "CFD simulation of particle suspension height in stirred vessels", *Trans. IChemE, Part A, Chem. Eng. Res. Des*., **82(A9)**, 1204–1213.

OCHIENG, A. and ONYANGO, M.S., (2008), "Drag models, solids concentration and velocity distribution in a stirred tank", *Powder Technology*, **181**, 1–8.

TAMBURINIA, A, CIPOLLINAA, A. MICALEA, G., CIOFALOB, M., and BRUCATOA, A., (2009), "Dense solid–liquid off-bottom suspension dynamics: Simulation and experiment", *Chem. Eng. Res. Des*., **87,** 587–597.

YAMAZAKI, H., TOJO, K., and MIYANAMI, K., (1986), "Concentration profiles of solids suspended in a stirred tank", *Powder Technology*, **48**, 205-216.

MANNINEN, M., TAIVASSALO, V. and KALLIO, S., (1996), ”On the mixture model for multiphase flow”, VTT Publications 288, Technical Research Centre of Finland.

FLUENT 6.3 USERS GUIDE (2006), Fluent Inc.

ISHII, M. and ZUBER, N., (1979), ”Drag coefficient and relative veloctity in bubbly, droplet and particulate flow”, *AIChE J*., **25**, 843-854.

