Advanced Powder Technology 23 (2012) 445–453

Contents lists available at SciVerse ScienceDirect

Advanced Powder Technology

journal homepage: www.elsevier.com/locate/apt

Original Research Paper

CFD simulation of solid–liquid stirred tanks

Divyamaan Wadnerkar, Ranjeet P. Utikar *,1, Moses O. Tade, Vishnu K. Pareek

Department of Chemical Engineering, Curtin University, Perth 6102, Australia

ARTICLE INFO

Article history:
Received 31 October 2011
Received in revised form 6 March 2012
Accepted 17 March 2012
Available online 25 April 2012

Keywords:
Solid–liquid suspension
Stirred tanks
Hydrodynamic study
Drag models
Homogeneity
Cloud height
CFD

ABSTRACT

Solid liquid stirred tanks are commonly used in the minerals industry for operations like concentration, leaching, adsorption, effluent treatment, etc. Computational Fluid Dynamics (CFD) is increasingly being used to predict the hydrodynamics and performance of these systems. Accounting for the solid–liquid interaction is critical for accurate predictions of these systems. Therefore, a careful selection of models for turbulence and drag is required. In this study, the effect of drag model was studied. The Eulerian–Eulerian multiphase model is used to simulate the solid suspension in stirred tanks. Multiple reference frame (MRF) approach is used to simulate the impeller rotation in a fully baffled tank. Simulations are conducted using commercial CFD solver ANSYS Fluent 12.1. The CFD simulations are conducted for concentration 1% and 7% v/v and the impeller speeds above the "just suspension speed". It is observed that high turbulence can increase the drag coefficient as high as forty times when compared with a still fluid. The drag force was modified to account for the increase in drag at high turbulent intensities. The modified drag is a function of particle diameter to Kolmogorov length scale ratio, which, on a volume averaged basis, was found to be around 13 in the cases simulated. The modified drag law was found to be useful to simulate the low solids holdup in stirred tanks. The predictions in terms of velocity profiles and the solids distribution are found to be in reasonable agreement with the literature experimental data. Turbulent kinetic energy, homogeneity and cloud height in the stirred tanks are studied and discussed in the paper. The presence of solids resulted in dampening of turbulence and the maximum deviation was observed in the impeller plane. The cloud height and homogeneity were found to increase with an increase in impeller speed. The work provides an insight into the solid liquid flow in stirred tanks.

1. Introduction

Solid–liquid mixing systems are amongst the common operations used in the field of chemical and mineral industry. The main purpose of mixing is the contact between the solid and liquid phase for facilitating mass transfer. In industrial processes effective mixing is necessary at both micro and macro level for adequate performance. At the micro level, micromixing governs the chemical and mass transfer reactions. Micromixing is facilitated by mixing at macro level. Numerous factors such as the just suspension speed, critical suspension speed, solids distribution, etc. dictate the mixing performance. CFD has proved to be a useful tool in analyzing the impact of these factors on the flow characteristics of such systems [1–7]. Proper evaluation of interphase drag is essential for accurate predictions using the CFD model. In this study four different drag models are analysed and their validity is checked by comparing the results of CFD simulations at low concentrations of solid with the experimental data available in the literature [8]. The behaviour of turbulence kinetic energy, suspension quality and cloud height are extensively discussed.

2. Literature review

Micale et al. [5] used Settling Velocity Model (SVM) and Multi fluid Model (MFM) approaches to analyse the particle distribution in stirred tanks. In SVM, it is assumed that the particles are transported as a passive scalar or molecular species but with a superimposed sedimentation flow, whereas in MFM, momentum balances are solved for both phases. Computationally intensive MFM was found to be better than SVM, but for both the models it was necessary to take into account the increase in drag with the increasing turbulence. Micale et al. [4] simulated the solids suspension of 9.6% and 20% volume fractions using the MFM approach and sliding grid (SG) approach using the Schillar Nauman drag model. Schillar Nauman is applicable on spherical particles in an infinite stagnant fluid and accounts for the inertial effect on the drag force acting on it. It provided satisfactory results at low impeller speed.

Derksen [9] conducted Eulerian–Lagrangian simulations to study the velocity field, turbulence, solid distribution and particle-impeller and particle–particle collision and frequencies

* Corresponding author. Tel.: +61 892669837; fax: +61 892662681.
E-mail address: r.utikar@curtin.edu.au (R.P. Utikar).
[1] Present address.

0921-8831/$ - see front matter
http://dx.doi.org/10.1016/j.apt.2012.03.007