Minerals Engineering 83 (2015) 156–167

Contents lists available at ScienceDirect

Minerals Engineering

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

Numerical and experimental investigation of single phase flow characteristics in stirred tanks using Rushton turbine and flotation impeller

Manjunath Basavarajappa [a,*], Teri Draper [b], Pal Toth [b], Terry A. Ring [b], Sanja Miskovic [a,1]

[a] Department of Metallurgical Engineering, University of Utah, Salt Lake City, UT 84112, USA
[b] Department of Chemical Engineering, University of Utah, Salt Lake City, UT 84112, USA

ARTICLE INFO

Article history:
Received 12 March 2015
Revised 21 July 2015
Accepted 24 August 2015
Available online 8 September 2015

Keywords:
CFD
Turbulence
Flotation impeller
PIV
Flow transition

ABSTRACT

In this study, computational fluid dynamics (CFD) simulations are used to investigate turbulent single phase flow characteristics in lab-scale stirred tanks with different geometric variations. Water at standard conditions is used as operating fluid. Rushton turbine (*RT*) and flotation impeller (*FI*) are used to agitate the fluid leading to turbulent flows in the tank. For *FI*, impeller diameter, d, is varied and three sizes corresponding to d values of 75, 100, and 150 mm are considered. Additionally, for 75 and 100 mm *FI*, off-bottom clearance, C, is varied from 100 ($D/3$) to 60 mm ($D/5$). The impeller based Reynolds number, *Re*, ranged from 29,000 to 120,000. CFD results are compared with LDA data from the literature for *RT* and in-house PIV data for *FI*. CFD predictions for *FI* are found to match experimental measurements satisfactorily with accurate prediction of flow transition at lower C. The normalized flow properties are observed to be invariant with *Re* for both impellers in fully turbulent regime. Mean flow characteristics for *FI* suggests that the flow is characterized by strong radial and tangential velocities close to impeller with peak values along disc level. Turbulence kinetic energy profiles close to impeller are characterized by two peaks suggesting development of trailing vortex which is further verified using swirling strength visualization. For *FI* with diameter equal to 100 mm, flow transition in which mean flow changes from radial flow (double loop) to axial-type (single loop) flow is observed when C is reduced. Both PIV measurements and CFD simulation are able to predict this transition accurately. Using both torque on rotating parts and volume averaged dissipation rate of turbulence kinetic energy, power numbers are calculated for both impellers. The axial-type flow at smaller clearance is marked by significant drop in power number value.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

Stirred tanks are extensively used in chemical, pharmaceutical, oil and gas, and minerals and metallurgical industries or blending, suspending, contacting, and dispersing applications. The flows generated in the stirred tanks are predominantly turbulent due to high impeller rotation speeds used to achieve necessary process conditions. Radial impellers are mainly used for dispersion and mixing applications and for processes that require high values of turbulence kinetic energy, k, and turbulence energy dissipation rate, ϵ (Joshi et al., 2011a). Various factors like tank size, impeller shape, impeller size, number of impeller blades, number of blades, and off-bottom clearance affect the flow in stirred tanks. Recently, Joshi et al. (2011a,b) reviewed CFD results reported in the literature for single and multiphase flows in stirred tanks using radial and axial impellers. Furthermore, Joshi et al. (2011a,b) compared results predicted by different turbulence models with experimental laser Doppler anemometry (LDA) measurements. Joshi et al. (2011b) have reviewed all the widely used modeling approaches in the literature and summarized the shortcomings associated with each of them. Based on their comparison of CFD predictions and experimental measurements, they recommend using large eddy simulation (LES) to obtain accurate predictions of turbulent quantities in the impeller region. However, LES is still very expensive for industrial size tanks at high Reynolds numbers and requires modeling of filtered scales that are not completely resolved.

For the radial impellers, the off-bottom clearance plays an important role in determining the type of flow pattern that is

* Corresponding author.
E-mail addresses: manjunath.basavarajappa@utah.edu (M. Basavarajappa), sanja.miskovic@utah.edu (S. Miskovic).
[1] Principal corresponding author.

http://dx.doi.org/10.1016/j.mineng.2015.08.018
0892-6875/© 2015 Elsevier Ltd. All rights reserved.

developed. Montante et al. (1999, 2001a,b) have investigated the effect of changing clearance for Rushton turbine and noticed the mean flow to transition from double-loop to single-loop type as the clearance is suitably reduced. Zhipeng et al. (2011a) have conducted particle image velocimetry (PIV) measurements and LES of stirred tank flows by varying impeller sizes, *Re*, and clearance. Consistent with the observations of Montante et al. (2001a), transition of flow from double to single-loop type at low clearance is also reported by Montante et al. (2001a) and Zhipeng et al. (2011a). Zhipeng et al. (2011a) varied the impeller diameter from one-third to one-half of tank diameter, *D*, at constant clearance of 0.15 times tank height, *H*, and *Re* equal to 64,000. Zhipeng et al. (2011a) reported the critical clearance at which flow transitions to strongly depend on impeller diameter. Furthermore, Zhipeng et al. (2011a) recommended using a smaller sized impeller for single-loop flow. Galletti et al. (2004) have made experimental measurements of Reynolds stresses in the impeller region using 3-D LDA. They found the highest levels of turbulence anisotropy not only close to impeller but also close to vessel bottom and near the centers of the flow re-circulation loops.

Recently, Murthy et al. (2007) have performed numerical simulations to assess different turbulence models in stirred tanks agitated by both axial and radial impellers. They found LES to predict the flow most accurately and Reynolds stress model (RSM) to perform well in the re-circulation region but under-predict k in the impeller region. More recently, Singh et al. (2011) performed CFD simulation of Rushton turbine driven stirred tank at *Re* equal to 28,830. Singh et al. (2011) reported that the two equation turbulence models do not predict the secondary vortex motions based on swirling strength isosurfaces visualization. Furthermore, they reported the two equation models to incorrectly predict the location of peak values for both k and ϵ. Of all the turbulence models considered, Singh et al. (2011) recommend using shear stress transport (SST) model with curvature correction, which is a combination of $k-\epsilon$ and $k-\omega$ models. Gimbun et al. (2012) argue that $k-\epsilon$ model is able to predict both radial and tangential velocities well except in the close vicinity of trailing vortices where the flow is extremely anisotropic. However, authors observed that $k-\epsilon$ under-predicted k close to impeller while detached eddy simulation (DES) provided a better prediction on a sufficiently fine grid. Recently, Basavarajappa and Miskovic (2013) numerically investigated turbulent single phase flow developed by flotation specific impeller but provided no direct validation of their approach.

In the current work, steady state CFD simulations of turbulent single phase flows are carried out in lab-scale stirred tanks. The objective of this work is to use CFD model to analyze flow generated by a generic flotation type impeller and validate the model against experimental measurements. Rushton turbine and flotation impeller are used to agitate the fluid, and the impeller rotation is modeled using multiple reference frames (MRF) approach. Validation of numerical results is achieved by comparing CFD results with experimental measurements from the literature for *RT*. In-house 2D PIV experiments are performed for flotation impeller and mean x and y velocity components are compared with CFD results. The geometry of the *FI* used in CFD simulations and experiments matched to ensure comparison of identical systems.

The design of flotation impeller is entirely conceived by authors based on literature review, process characteristics, and experience, and the design is representative of flotation impellers employed in commercial flotation machines. Reynolds number in the range of 29,000–120,000 in the fully turbulent flow regime is used. Furthermore, for the flotation impeller study, impeller size, and clearance are varied to study their effect on averaged flow pattern. Turbulence in the flow is modeled using realizable $k-\epsilon$ model, which is better suited for rotating flows (ANSYS FLUENT Theory Guide Release 14.5, 2013). In addition to geometrical variations, RSM turbulence model is used to study the effect of turbulence model on predictions. Based on the analysis of velocity and turbulence characteristics, average flow features produced by flotation impeller are summarized.

2. Tank/cell configuration and impeller geometry

2.1. CFD simulations

A cylindrical tank of diameter, *D*, and tank height, *H*, both equal to 300 mm is used. The tank is closed at the top to avoid interaction with air which is a common practice employed for single phase simulations (Ng and Yianneskis, 2000). Four equally spaced baffles of width, $B = D/10$, are used to contain fluid rigid-body rotation and improve mixing efficiency. A schematic of the stirred tank fitted with flotation impeller is shown in Fig. 2. A six blade Rushton turbine (*RT*) of diameter, *d*, equal to 100 mm is used. A circular shaft of diameter equal to 20 mm ($D/15$) supports the *RT*. The clearance, *C*, defined as the distance from the tank floor to *RT* disc is kept constant at 100 mm ($D/3$). To avoid using full names of impellers tested, a concise notation with impeller name followed by its diameter is used in rest of the paper. Detailed schematic of $RT-100$ is shown in Fig. 1(a). For the flotation impeller (*FI*), same tank, shaft, and baffle dimensions are used. *FI* has six blades supported from the disc. For *FI* with d equal to 100 mm, blades are parallel to tank wall for 25 mm ($a = d/4$ for other sizes) and taper inwards towards shaft at an angle of 45°. The vertical length of inclined part h is equal to 20 mm ($d/5$ for other sizes). A schematic of *FI* is shown in Fig. 1(b) with design parameters a and h. To study the effect of blade size, three blade sizes with disc to tank diameter ratios, namely $d/D = 1/4$, 1/3, and 1/2 corresponding to d values of 75, 100, and 150 mm, respectively, are used. The other geometrical dimensions are proportionally varied for different sized impellers. The impellers are concisely named $FI-75$, $FI-100$, and $FI-150$. In addition to impeller size, off-bottom clearance for $FI-75$ and $FI-100$ is also varied by decreasing it from $D/3$ to $D/5$.

All the simulations presented in this work are for single phase flow and the working fluid used is water at 20° (293 K) and fluid is initially at atmospheric pressure. The *Re* of the flow is obtained using the impeller diameter as characteristic length scale, $Re = N\rho d^2/\mu$, where, N is impeller speed in revolutions per second, ρ is fluid density in kg/m^3, d is impeller diameter in m, and μ is dynamic viscosity of fluid in kg/(s.m). All the flows considered have *Re* equal to 29,000 and over, therefore, the flows fall under fully turbulent regime. Even though the flows are transient, the most important (mean) characteristics for turbulent flows can be studied by assuming steady state conditions.

2.2. PIV experiments

PIV experiments are conducted in a clear plexiglass tank of diameter equal to 300 mm. The PIV system used in this work is manufactured by LaVision. The measurements are made on surface of approximately 116 mm × 116 mm. More than 700 image pairs are processed to obtain mean velocity and turbulent data. $FI-100$ impeller is used in the experiments at two C values of 100 and 60 mm. The shaft and the impeller mounting mechanism used in the experiments is different from the numerical geometry described earlier. The thickness of the impeller disc used in experiments is 13 mm and different attachment mechanism is used to secure the impeller to the shaft. The *Re* based on the impeller rotation speed is calculated to be 40,375.

Download English Version:

https://daneshyari.com/en/article/232837

Download Persian Version:

https://daneshyari.com/article/232837

Daneshyari.com