

CFD Simulation of Solid-Liquid Stirred Tank with Rushton Turbine and Propeller Impeller

M. H. Pour, V. M. Nansa, M. Saberi, A. M. Ghanadi, A. Aghayari, and M. Mirzajanzadeh

Abstract—Stirred tanks have applications in many chemical processes where mixing is important for the overall performance of the system. In present work 5%v of the tank is filled by solid particles with diameter of 700 μ m that Rushton Turbine and Propeller impeller is used for stirring. An Eulerian-Eulerian Multi Fluid Model coupled and for modeling rotating of impeller, moving reference frame (MRF) technique was used and standard-k- ϵ model was selected for turbulence. Flow field, radial velocity and axial distribution of solid for both of impellers was investigation and comparison. Comparisons of simulation results between Rushton Turbine and propeller impeller shows that final quality of solid-liquid slurry in different rotating speed for propeller impeller is better than the Rushton Turbine.

Keywords—CFD, Particle Velocity, Propeller Impeller, Rushton Turbine.

I. INTRODUCTION

SOLID-LIQUID slurry are agitated using one or more impellers, are one of the most important unit operations in the chemical, biochemical, and mineral processing industries, because of its ability to provide excellent mixing between the phases. The flow pattern and turbulence prevailing in the tank ensures good heat and mass transfer properties for the system, apart from providing good solid suspension within the vessel. Despite its widespread use, the design and operation of these tanks and impeller design still remain a challenging problem because of the complexity of three dimensional circulating and turbulent multiphase flow encountered in the tanks. With the improvement in computational capabilities, computational fluid dynamics (CFD) has emerged as a viable option to study turbulent multiphase flows and gain insights on the hydrodynamic behavior of complex systems.

Mehdi Hashemi Pour is with the Department of chemical Engineering, Ahar Branch, Islamic Azad University, Ahar, Iran (e-mail: Me.Che79@yahoo.com).

Vahid Moghaddam Nansa is with the Department of Chemical Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran (corresponding author to provide phone: +98 912 193 4781; e-mail: v.moghaddam@ymail.com)

Mansour saberi is with the Department of Chemical Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran (e-mail: saberi1213@yahoo.com)

Asghar Mirzazadeh Ghanadi is with the Department of Chemical Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran (e-mail: asgharmirzazadeh@yahoo.com)

Ali Aghayari is with the Faculty of Chemical Engineering, Sahand University of Technology, Sahand, Iran (e-mail: apadideh9@gmail.com)

Mehrdad Mirzajanzadeh is with the Department of Chemical Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran (e-mail: mehrdad_1228@yahoo.com).

The optimum design and the efficiency of mixing operations are important parameters on product quality and production costs, so being aware of the different characteristics such as velocity distribution profiles and turbulence parameters, is very important to optimization of using the vessels. The flow motion in stirred tanks is 3-dimensional and complex. In the area surrounding the impeller, the flow is highly turbulent. In recent years, computational fluid dynamic (CFD) techniques increasingly used as a substitute for experiment to obtain the details flow field for a given set of fluid, impeller and tank geometries. There are many different CFD methods available to simulate the flow in stirred tanks. A detailed review of these methods was made by Brucato et al. [1]. The different methods can be divided into two distinct classes: one in which the whole tank is described with a fixed coordinate system and one in which a part of the coordinate system co-rotates with the impeller.

Micale et al. [2] 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. [3] simulated the solids suspension of 9.6% and 20% volume fractions using the MFM approach and sliding grid (SG) approach using the Schiller Nauman drag model. Schiller 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.

Ochieng and Lewis [4] simulated nickel solids loading of 1-20%w/w with impeller speeds between 200 and 700RPM using both steady and transient simulations and found out that transient simulations, although time consuming, are better for stirred tank simulations. The initial flow field was generated using the multiple reference frame (MRF) approach and then the simulations were carried out using SG. The Gidaspow model was used for the drag factor, which is a combination of the Wen and Yu model and the Ergun equation Ding and Gidaspow, [5]. Wen and Yu drag is appropriate for dilute systems and Ergun is used for dense packing. For the study of just suspended of solids using solids at the bottom of the tank as an initial condition, it provided satisfactory results.

Few researchers have compared the different design of impeller. Satio et al. [6] studied the performance of such impellers. Bakker et al. [7] studied the performance of impellers with a semicircular blade shape, the chemineer CD-6.

A comparative analysis of the fluid dynamic performance of the concave turbines and hydrofoil impellers was provided by Neinow [8].

Bakker et al [9] designed the new impeller BT-6 that has been optimized to take in to account the different flow conditions above and below the disc. The BT-6 is a radial flow turbine and asymmetric parabolic blade shape allows it to disperse more gas than any other radial flow turbine.

II. METHODOLOGY

The hydrodynamic study is simulated using Eulerian-Eulerian multiphase model. Each phase, in this model, is treated as an interpenetrating continuum represented by a volume fraction at each point of the system. The Reynolds averaged mass and momentum balance equations are solved for each of the phases. The governing equations are given below:

Continuity equation:

$$\frac{\partial}{\partial t} (\alpha_q \rho_q) + \nabla \cdot (\alpha_q \rho_q \mathbf{u}_q) = 0 \quad (1)$$

$$\frac{\partial}{\partial t} (\alpha_q \rho_q \mathbf{u}_q) + \nabla \cdot (\alpha_q \rho_q \mathbf{u}_q \mathbf{u}_q) = \alpha_q \nabla P + \nabla \cdot \tau_q + \alpha_q \rho_q \mathbf{g} + (F_{td} + F_q + F_{lift,q} + F_{vm,q}) + C \quad (2)$$

where q is 1 or 2 for primary or secondary phase respectively, α is volume fraction, ρ is density, \mathbf{u}_q is velocity vector, P is pressure and is shared by both the phases, τ_q is the stress tensor because of viscosity and velocity fluctuation, g is gravity, F_{td} is force due to turbulent dissipation, F_q is external force, $F_{lift,q}$ is lift force, F_{vm} is virtual mass force and F_{12} is interphase interaction force.

The stress-strain tensor is due to viscosity and Reynolds stresses that include the effect of turbulent fluctuation. Using the Boussinesq's eddy viscosity hypothesis the closure can be given to the above momentum transfer equation. The equation can be given as:

$$\tau_q = \alpha_q \mu_q (\nabla \mathbf{v}_q + \nabla \mathbf{v}_q^T) + \alpha_q (\lambda_q - \frac{2}{3} \mu_q) \nabla \cdot \mathbf{v}_q \mathbf{I} \quad (3)$$

where μ is the shear viscosity, λ is bulk viscosity and \mathbf{I} is the unit stress tensor.

III. CFD METHOD

3D computational fluid dynamic (CFD) simulations were carried out in order to model the behavior of stirred vessels with Propeller impeller and Rushton turbine for baffled configurations. The mesh geometry of the mixing tank was created using Gambit 2.3 and Fluent version 6.3 was used for

solving a set of nonlinear equations formed by discretization of the continuity, the particle mass balance and momentum equations. A computational grid consisting of three parts: an inner rotating cylindrical volume enclosing the turbine, a bottom of the tank that solid particles are there and an outer, stationary volume containing the rest of the tank. The structured grids, composed of non-uniformly distributed tetrahedral cells, were used in the three parts. That consisted of about 400000 tetrahedral nodes. The schematic diagram of the tank geometry is shown in Fig. 1 and impellers with tetrahedral mesh are shown in Fig. 2. The dimensions used are tank diameter, T=13cm and tank height, H=22.5cm. The tank has 3 baffles with the width of T/10(1.3cm) length of 22.5cm (Radius of curvature is 1.4° with thickness of 0.1cm). Distance of baffle from wall is 1.52cm. Shaft diameter is 0.8cm. The bottom clearance of the impellers was kept constant at T/4. The geometrical characteristics of the impellers are given in Table I. In this work study 5%v of the tank is filled with solid particles with diameter of 700 μ m. The properties of solid and liquid phase are presented in Table II. Rushton Turbine and Propeller impellers are used for stirring. An Eulerian-Eulerian Multi Fluid Model coupled and for modeling rotating of impeller, moving reference frame (MRF) technique was used and standard-k- ϵ model was selected for turbulency. Standard no-slip boundary condition was considered for all solid surfaces.

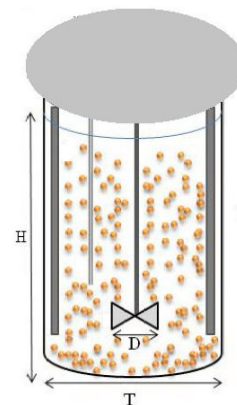


Fig. 1 Schematic diagram of the tank geometry

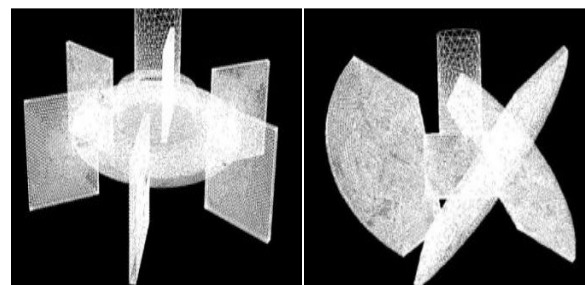


Fig. 2 (a)

Fig. 2 (b)

Fig. 2 (a) Rushton Turbine (b) Propeller

TABLE I

DESIGN PARAMETER AND DIAMETER AND DIMENSION OF IMPELLERS			
Impeller Type	Impeller diameter (cm)	Blade no	Blade Angle
Rushto Turbine	4.6	6.	90
Propeller	4.6	3	50

TABLE II

SPECIFICATION OF SOLID PARTICLES AND LIQUID PHASE			
Type / Shape	Specific gravity	Average size (d_p)	Solid loading (x_v)
Ion exchange resin/ Spherical	1.3	700 μm	5% vol.
Water	Temperature ($^{\circ}\text{C}$)	Density (kg/m ³)	Viscosity (pa.s)
	25	998.2	10^{-3}

IV. RESULT AND DISCUSSION

A. Flow Field

The velocity vector for solid and liquid phases plot of mean flow field located in the perpendicular plane crossing impeller center, X=0 plane is shown in Fig. 3 and Fig. 4. It can be observed that the mean velocity field of solid phase is similar to that of liquid, which means that the solid particles trace the liquid closely. A low velocity region persists away from the shaft at the top of the tank.

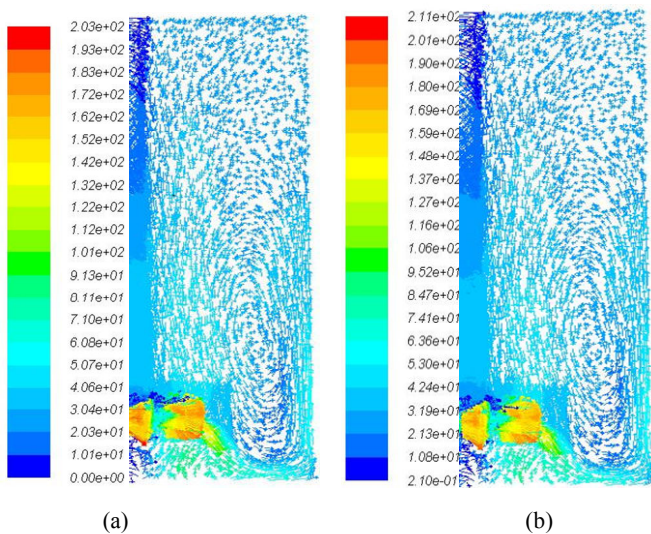


Fig. 3 Velocity field (m/s) along a cross-section of the tank through the middle of the tank for Rushton Turbine in N=600 (rpm) (a) solid phase (b) Liquid phase

The figure shows the orientation of the impeller discharge stream and ring vortices formed around the impeller disc. The one circulation flow pattern created in tank that for Rushton Turbine this loop created to upper to the surface of the tank and for Propeller impeller this loop created to lower part of the tank but both of them axially along the shaft returning to center of impeller and pumping by jet flow of impeller. As it shows the circulation pattern in Rushton Turbine is longer than Propeller but flow in tank with Propeller impeller is much turbulence than the Rushton Turbine.

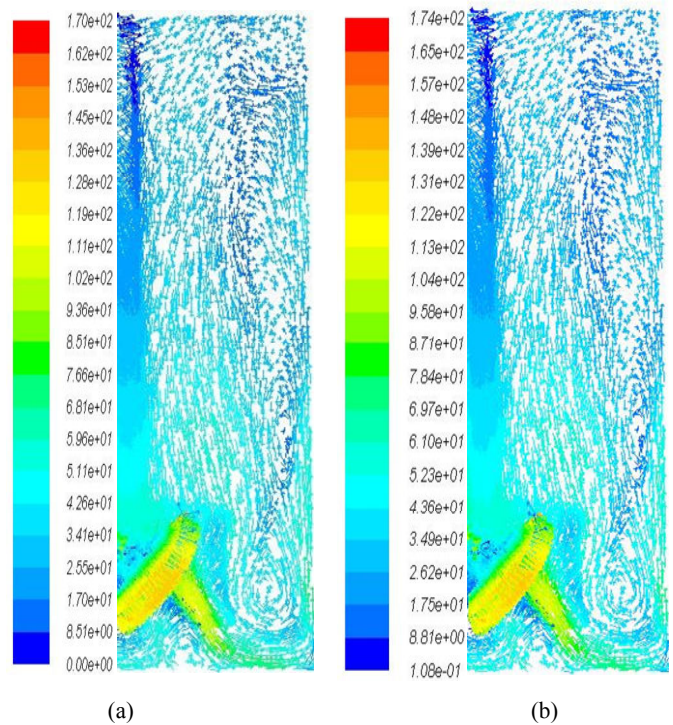


Fig. 4 Velocity field (m/s) along a cross-section of the tank through the middle of the tank for Propeller impeller in N=600 (rpm) (a) solid phase (b) Liquid phase

B. Particle Velocity

The radial profile of sand velocity for Propeller Impeller and Rushton Turbine in different location of tank with rotating speed of 600rpm are shown in Fig. 5. It is observed that impeller stream flows away from the impeller blades, and the velocity varies from dramatically in the axial direction. The velocity profiles in the impeller stream become flatter as the fluid moves away from the impeller. This is due to the entrainment of slow moving surrounding fluid into the impeller stream.

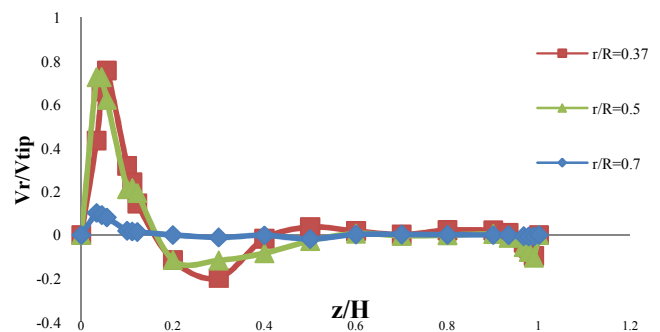


Fig. 5 Normalized radial velocity profile of solid concentration in different height of the tank for Propeller impeller at 600rpm

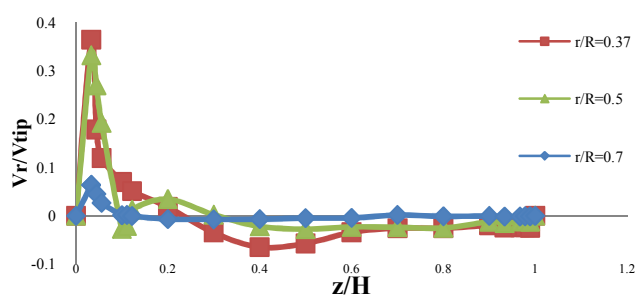


Fig. 6 Normalized radial velocity profile of solid concentration in different height of the tank Rushton Turbine at 600rpm

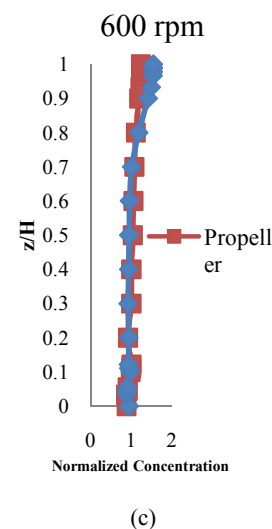


Fig. 7 Normalized axial concentration profile in different height of the tank, radial position of $r/R=0.37$ and different speed of impellers. (a) 200rpm (b) 400rpm (c) 600rpm

Fig. 5 and Fig. 6 shows that radial profile of solid particle for propeller impeller is similar to that of Rushton Turbine but value of normalized radial velocity in near to the impellers is different and this value is higher for propeller impeller than Rushton Turbine. Thus the jet flow of propeller impeller is bigger than the Rushton Turbine.

C. Solid Concentration

Fig. 7 shows the comparisons of axial distribution of solid volume fraction at the radial position of $r/R=0.37$ between Propeller impeller and Rushton Turbine in rotating speed of 200rpm, 400rpm and 600rpm. The simulation results observed that the distribution of the solid phase for both of impellers is much homogeneous in high rotating speed and the value of normalized concentration increasing with increase of rotation speed, but distribution of solid phase for propeller impeller is much homogeneous than the Rushton Turbine. Results show that the value of normalized concentration for Propeller impeller in each rotating speed is higher than the Rushton Turbine.

V. CONCLUSIONS

In present work, a computational fluid dynamics (CFD) model is developed to study the solid-liquid stirred tank with two types of impellers. Flow field, solid particle velocity and solid concentration distribution are simulated. Results show that flow field is different for Rushton Turbine and Propeller impeller, but radial profile for Rushton Turbine is similar to that Propeller impeller with different value. Investigation of solid concentration profile shows with an increase with rotating speed, dispersion rate and consequently the final quality of solid-liquid mixing system will rise and comparisons of axial distribution of solid volume fraction between Propeller impeller and Rushton Turbine shows that the propeller impeller is very good for using in solid-liquid systems at low or high rotating speed.

REFERENCES

- [1] Brucato, A., M. Ciofalo, F. Grisafi and G. Micale, "Numerical prediction of flow fields in based stirred vessels: A comparison of alternative modelling approaches", *Chem. Eng. Sci.* 53, 1998, pp. 3653-3684.
- [2] Micale, G., Montante, G., Grisafi, F., Brucato, A. & Godfrey, J. "CFD Simulation of Particle Distribution in Stirred Vessels". *Chemical Engineering Research and Design*, 78, 2000, pp. 435-444.
- [3] Micale, G., Grisafi, F., Rizzuti, L. & Brucato, A. "CFD Simulation of Particle Suspension Height in Stirred Vessels". *Chemical Engineering Research and Design*, 82, 2004, pp.1204-1213.
- [4] Ochieng, A. & Lewis, A. E. "CFD simulation of solids off-bottom suspension and load height". *Hydrometallurgy*, 82, 2006, pp. 1-12.
- [5] Ding, J. & Gidaspow, D. "A bubbling fluidization model using kinetic theory of granular flow". *AIChE Journal*, 36, 1990, pp. 523-538.
- [6] F.Satio, A.W.Nienow, S.Chatwin, I.P.T.Moore, "Power, gas dispersion and homogenisation characteristics of Scaba SRGT and Rushton turbine impellers", *J Chem Eng Japan*, 25(3), 1992, pp.281-287.
- [7] A.Bakker, K.J.Myers, and J.M.Smith, "How to disperse gases in liquids", *Chem Eng*, 101(12), 1994, pp.98-104.
- [8] A. W. Nienow, "Gas-liquid mixing studies: a comparison of Rushton turbines with some modern impellers", *Chem Eng Res Des*, 74, 1996, pp.417-423.
- [9] A.Bakker, "Impeller assembly with asymmetric Concave blades", *US Patent 5791780*, 1998.

