Numerical analysis on mixing performance of logarithmic spiral impeller

W X Xu¹,², F Xue¹*, W H Wang², S Q Yuan³ and Z M Ji²

¹ School of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China
² Suzhou Sinye Materials technology Co., Ltd. Suzhou 215151, China
³ National Research Centre of Pumps, Jiangsu University, Zhenjiang 212013, China

13775376699@163.com

Abstract. Mixing performance of the solid-liquid two phase system in a mixing tank equipped with a logarithmic spiral impeller is studied with fluid dynamics software FLUENT. Based on the kinetic theory of granular flow, the basic control equations of the E-E model are established. A multiple reference frame (MRF) method and the RNG k-ε turbulence model are chosen to simulate solid-liquid flow performance and results are compared with that of the four blade turbine (MK) and the six blade turbine (PY). Research shows that the logarithmic helicoidal impeller improves solid-liquid axial-flow performance, and the concentration, power consumption and the mixing time of it are better compared with the PY and MK impellers. For a wide range of mixing velocities, the concentration of the sludge particles is always lower in the upper position to impeller than in the lower position. With the increase of velocity, the power consumption raises gradually, the uniformity of solid particle distribution decreases and the mixing time reduces.

1. Introduction

The impeller is widely used in petrochemical, pharmaceutical, metallurgy, food, polymer, papermaking, environmental protection and other industrial fields. Different types of mixing impeller produce different flow fields. The concentration distribution of the mixing tank can be affected by the mixing process of the solid solution. Whether the mixing tank is evenly mixed and whether the mixing time is appropriate has a decisive influence on the safety and product quality of the whole mixing process. Therefore, it is of great significance to understand the distribution, power and mixing time size of different mixing impeller for the development and optimization of the mixing equipment.

In practice, the power consumption is large, the mix is uneven, and the dead space of mixing is easy to appear [1]. A new type of logarithmic spiral impeller is designed. The structure can effectively reduce the flow loss. The axial flow and radial mixing are combined organically. It is easier to obtain the three-dimensional circulation to mix the flow, and the efficiency is higher, and the mixing effect is better.

In this paper, the distribution and mixing properties of the flow field in the logarithmic spiral impeller by numerical simulation are studied. The research results are compared with the traditional axial flow impeller (MK) and radial flow impeller (PY), and the mixing characteristics of logarithmic spiral impeller are evaluated [2].
2. Physical Model

In order to study the mixing effect of a logarithmic spiral impeller, based on its structural characteristics, a rectangular mixing tank is selected. Its base is 300mm x 300mm, H=400mm. The logarithmic spiral impeller with d=100mm, n=250r/min is selected. The research is water and solid particles. The diameter of the solid particle is about 100 um, and its density is 3970 kg/m$^3$. The density of water is 1000 kg/m$^3$, and its viscosity is 1.005×10$^{-3}$ Pa·s. The logarithmic spiral impeller is located in the distance (C=1/20H) to the bottom. In the other hand, the traditional axial flow impeller (MK) and the radial flow impeller (PY) are located in the distance (C=1/3H) to the bottom respectively [3]. The three types of impellers are shown in figure 1 to figure 3.

![Figure 1. Logarithmic spiral impeller.](image)

![Figure 2. PY impeller.](image)

![Figure 3. MK impeller.](image)

3. Mathematical Model

Based on the mass conservation, the momentum conservation and the energy conservation, the control equations of fluid phase and particle phase in solid-liquid mixing flow model are described [4-6].

The fluid phase continuity equation:

$$\frac{\partial \alpha_f}{\partial t} + \nabla \cdot (\alpha_f \mathbf{v}_f) = 0$$  (1)

The particle phase continuity equation:
\[
\frac{\partial \alpha_f}{\partial t} + \nabla (\alpha_f v_f) = 0
\]  
\hspace*{1cm} (2)

The fluid phase momentum equation:

\[
\frac{\partial (\alpha_f \rho_f v_f)}{\partial t} + \nabla (\alpha_f \rho_f v_f v_f) = -\beta (v_f - v_p) - \alpha_f \nabla p + \alpha_f \rho_f g - F_{ad,f}
\]  
\hspace*{1cm} (3)

The particle phase momentum equation:

\[
\frac{\partial (\alpha_p \rho_p v_p)}{\partial t} + \nabla (\alpha_p \rho_p v_p v_p) = -\beta (v_p - v_f) - \alpha_p \nabla p + \alpha_p \rho_p g - F_{ad,p}
\]  
\hspace*{1cm} (4)

\(\alpha\) is the volume fraction, \(\rho\) is the density. The right parts of the above two equations are the drag force, the pressure gradient, the gravity and the additional forces [7-8].

The drag coefficient \(\beta\) is a function of the particle drag coefficient \(C_D\):

\[
\beta = C_D \frac{3\alpha_p \rho_f \left| v_f - v_p \right|}{4d_p} \alpha_f^{-1.8}
\]  
\hspace*{1cm} (5)

According to the classic Dallavalle correlation, the particle drag coefficient \(C_D\) is:

\[
C_D = \left[ 0.63 + \frac{4.8}{R_e} \right]^2, R_e = \frac{\alpha_f \rho_f \left| v_f - v_p \right| d_p}{\mu_f}
\]  
\hspace*{1cm} (6)

4. Numerical Simulation

4.1. Mesh Generation

The structured grids can be used when calculating the geometric structure of the area, but the unstructured grids can be used when the computational areas are more complex and irregular areas. In this paper, the pre-processor Gambit in FLUENT is applied. The mixed gridding technology is applied at gridding drawing process, and the gridding of small structure is diminished. The unstructured grids are used in the interior zone and the exterior zone, and the structured grids are used in the bottom zone (as shown in figure 4). Because the calculation results of the bottom zone and interior zone are affected by the number of the grid, the grids are refined in these two parts to make the calculation more accurate[9-10]. In this paper, the grid irrelevant tests were made by the effect of grid size on power number and deviation(as shown in figure 5). The number of the grids in flow field of three mixing impeller is 629562, 635841 and 618876 respectively.
4.2. Boundary Conditions

Boundary conditions setting: the tank wall, the mixing shaft, and the mixing impeller surface are defined as the wall boundary of the computational area. The liquid phase is defined as the non-slipping wall boundary, \( \nu_i=0, \ i=x,y,z \). And the particle phase is used as the zero shear stress wall boundary. The free liquid surface is set to a symmetric plane, so the gradient of the velocity and concentration of the surface is zero. Since the details of the flow in near-wall zone are not paid more attention, it is possible to use the standard wall function method to calculate the near-wall zone [11].

4.3. Simulation Methods

Based on the kinetic theory of granular flow, the basic control equations of the E-E model are established. A multiple reference frame (MRF) method and the RNG k-\( \varepsilon \) turbulence model are chosen to simulate solid-liquid flow performance. The pressure-velocity coupling algorithm is SIMPLEC, the momentum spreads is applied the second-order upwind difference scheme [12]. The turbulence kinetic energy and the turbulent kinetic energy dissipation rate are used in the first order upwind difference scheme. And the convergence residuals of the all variables are set to \( 10^{-4} \).

5. Simulation Results

5.1. Flow Characteristics

Under the action of the logarithmic spiral impeller, there are five flow areas in the mixing tank, it includes: the impeller, the discharge area of the impeller, the boundary area between the bottoms with the wall, the area under the impeller, and the main circulation area (as shown in figure 6). The fluid is jetted in radial with the help of the logarithmic spiral impeller. And the axial circulation flow is formed by the guide of the cyclic motion.
Take the mixing velocity of 250 r/min for example. There are four contour lines of the flow field in the maximum longitudinal section of the mixing tank. The positions of these lines are: Z=10mm (below the impeller), Z=20mm (near the impeller), Z=50mm (above the impeller), Z=150mm(Main circulation area). The mixing effect is determined by the velocity distribution of these positions [13]. The velocity distribution pattern is shown in figure 7. In the figure, R is the distance from the mixing axis in the tank. According to figure 7, the absolute velocity of the four different height sections is consistent with the change of radius. And the peaks of the velocity are all at R=50mm, which is near the largest diameter outer edge of the impeller. In addition to the negative pressure zone of the impeller, the velocity of the remaining flow field is bigger than 0.1m/s. It indicates that the mixing effect of the logarithmic spiral impeller can meet the design requirements.

5.2. Concentration Distribution
In order to understand the concentration distribution of the solid particles in the tank, there are three radial positions are selected on the plane of y=0 to observe the axial distribution of the solid volume concentration. The positions are L/R=0.3, 0.6 and 0.9. The horizontal coordinate C/Cavg is the ratio of the concentration of the sampling to the average concentration, and the vertical coordinate z/H is the axial position of the sampling point [14]. At the same time, in order to study the effects of the impeller velocity on the concentration distribution. There are five conditions are calculated respectively by velocity of 150r/min, 200r/min, 250r/min, 300r/min and 350r/min.

When the velocity is 150r/min, the top of the flow field has a clean layer, and the particles are piled up badly at the bottom as shown in figure 8. The particles of the bottom cannot float because of the low velocity. When the velocity is 200r/min, the suspension height of the particles is obviously increased for the higher velocity. The mixing effect is better than that of the 150r/min. When the velocity is 250r/min, the particles are filled with the whole flow field, but the concentration of the top is small. When the velocity is 300r/min, the uniformity of the concentration of the whole tank is increasing. When the velocity is 350r/min, the concentration is close to uniform field.
Through the concentration curve, we can know that: when the velocity is low, there are a clean layer at the top of the mixing tank and a stack layer on the bottom. The mixing effect of the particles improves with the increasing velocities. And the clean layer gradually reduces and disappears. Then the particles fill the tank. The local concentration of the particles shows strong in homogeneity in the axial direction. Fluid mixing performance could be improved by increasing mixing velocity.

5.3. Power Consumption
In the simulation process, the power of the three mixing impeller can be calculated by Monitoring the torque of the impeller as shown in the figure 9[15]. The figure shows that: with the increasing of the velocity, the power consumptions of the impellers are aggrandized. At the same velocity, the power consumption of the PY impeller is more than four times of the other two impellers’. And the power consumption of the logarithmic spiral impeller is be reduced by 20% compared with the MK impeller.

5.4. Mixing Time
In this paper, there are 26 different points are taken in different areas of the whole mixing tank. These selected points can fully reflect the mixing time of the whole flow field, and avoid the influence of the selected location on the calculation of mixing time. At the time of simulation, the concentration of these points is detected and the 95% rule is adopted [16]. That is: When the concentration of some point reaches the final concentration of 95%~105%, the time used is the mixing time of the point as shown in figure 9. It can be seen from figure 10 that the mixing time decreases with the increase of the mixing velocity. At the same velocity, the mixing time of the PY impeller is the shortest, and the logarithmic spiral impeller is the second, and the MK impeller is the longest.
6. Conclusion
With the guidance of the theories of solid-liquid interaction flows, the structure of the logarithmic spiral impeller is good at reducing the hydraulic losses and resolving those problems, such as higher consumption of energy and lower efficiency. There are two parts in this impeller, one is the axial diversion and the other is the radial mixing. Both the axial and radial flow can be obtained without the reflection of tank wall. The impeller has higher efficiency and better mixing effect.

Based on the kinetic theory of granular flow, the basic control equations of the E-E model are established. As there are some different opinions about the particle viscous force, solid pressure force and fluid-particle and particle-particle interactions, an interaction force, which is an additional force, is leaded into the E-E model. And the Eulerian two-phase model is simplified by the above consideration.

The solid-liquid suspension flow generated by logarithmic spiral impeller in the mixing tank is simulated. The velocity, the concentration, the power consumption and the mixing time of this kind impeller is compared with which in PY and MK impellers. For a wide range of mixing velocities, the concentration of sludge particles is always lower in the upper location to impeller than in the lower position. As the velocity of the impeller increases, the sludge particles distribution in the flow field becomes uniform. With the increase of velocity, the consumption of the power raises gradually, the uniformity of the particle distribution decreases and the mixing time reduces. For the mixing impellers (MK, logarithmic spiral and PY) with same diameters under various velocities, the PY impeller gives the best mixing result and MK impeller is the poorest. In addition, the PY impeller requires the highest power consumption, while the logarithmic spiral impeller saves the most.

Acknowledgments
This study is supported by Postdoctoral Research Project of Suzhou Sinye Materials technology Co., Ltd., Development and Demonstration Project of Jiangsu Province (BE2016341), National Natural Science Foundations of China (51676086), Key Technology R&D Program of Zhenjiang (NY20140048), Scientific Research Foundation of Jiangsu University (14JDG178).

References
[1] Montante G and Lee K C 2000 Liquid and solid particle mean flow and turbulence levels in a stirred vessel with low impeller clearance. Institution of Chemical Engineers Symposium Series. 146 305-16
[2] Wang F., Mao Z S and Wang Y 2006 Measurement of phase holdups inliquid-liquid-solid three-phase stirred tanks and CFD simulation. Chemical Engineering Science. 61 7535-50
[3] Ljungqvist M and Rasmuson A 2001 Numerical simulation of the two-phase flow in an axially stirred vessel. Trans IchemE. 79 533-46
[4] Špidla M, Mostek M and Sinevic V 2005 Experimental assessment and CFD simulations of local solid concentration profiles in a pilot-scale stirred tank. Chemical Papers. 59(6a) 386-93
[5] Angeline D., Line A and Morchain J 2008 LES and URANS simulations of hydrodynamics in mixing tank: Comparison to PIV experiments. Chemical Engineering Research and Design. 86 1322-30
[6] Xu Y and McGrath G 1996 CFD predictions of stirred tank flows. Trans Inst Chem Eng. 17 471-75
[7] Vashisth S and Nigam K D P 2009 Prediction of flow profiles and interfacial phenomena for two-phase flow in coiled tubes. Chemical Engineering and Processing. 48 452-63
[8] Zadghaffari R., Moghaddas J S and Revstedt J 2009 A mixing study in a double-rushton stirred tank. Computers and Chemical Engineering. 33 (7) 1240-46
[9] Panneerselvam R., Savithri S and Surender G D 2009 CFD simulation of hydrodynamics of gas-liquid-solid fluidised bed reactor. Chemical Engineering Science. 64 1119-35
[10] Zakrzewska B., Peryt S and Jaworski Z 2001 A study of transitional and turbulent boundary flows in a stirred tank. I. LDV measurements. Inzynieria Chemiczna i Procesowa. 22(3A)
427-43

[11] Virdung T and Rasmuson A 2008 Solid-liquid flow at dilute concentrations in an axially stirred vessel investigated using particle image velocimetry. Chemical Engineering Communications. 195(1) 18-34

[12] Inoue Y, Yano H and Yamashita S 2007 Experimental study on a three-dimensional wall jet flow field in a cross-section of developed region. Journal of Fluid Science and Technology. 2(3) 655-64

[13] Killgore K J, Maynord S T and Chan M D 2011 Evaluation of propeller-induced mortality on early life stages of selected fish species. North American Journal of Fisheries Management. 21(4) 947-55

[14] Chung K H K, Barigou M and Simmons M J H 2007 Reconstruction of 3-D flow field inside miniature stirred vessels using a 2-D PIV technique. Chemical Engineering Research and Design. 85(5A) 560-67

[15] Angst R and Kraume M 2016 Experimental investigations of stirred solid-liquid systems in three different scales: particle distribution and power consumption. Chemical Engineering Science 61(9) 2864-70

[16] Patil D J, Annaland M V S and Kuipers J A M 2015 Critical comparison of hydrodynamic models for gas-solid fluidized beds-part: bubbling gas-solid fluidized beds operated with a jet. Chemical Engineering Science. 60 57-72