Light Metals 2011 *Edited by: Stephen J. Lindsay*
TMS (The Minerals, Metals & Materials Society), 2011

Physical Simulation and Numerical Simulation of Mixing Performance in the Seed Precipitation Tank with a Improved Intermig Impeller

Zhang Ting-an[1], Liu yan[1], Wang shuchan[1], Zhao hongliang[1], Zhang chao[2], Zhao qiuyue[1], Lv guozhi[1], Dou zhihe[1]

(1 School of Materials and Metallurgy of Northeastern University, Key Laboratory of Ecological Utilization of Multi-metal Intergrown Ores of Education Ministry, Shenyang, 110004; 2 Shenyang Aluminum & Magnesium Engineering & Research Institute, Shenyang, 110001))

Keywords: Seed precipitation tank; Mechanical stirring; Numerical simulation; Solid-liquid phase flow

Abstract

A PC6D fiber optic reflection probe was used to measure local solid concentrations distributions in the seed precipitation tank with a new-style Improved Intermig impeller, and the commercial software FLUENT 12.0 on parallel computing graphic workstation was used to simulate numerically its flow fields. The physical simulation results show that at high rotation and viscosity, high-speed particles are more conducive to uniform distribution. When η=3.50cp, n=172rpm, the multiple relationships of the average value of the experiment and theory is 1.011 times. The numerical simulation results show that the too long or too short impeller off-bottom clearance height(C) is not conducive to the suspension of $Al(OH)_3$ particles. Enlarging the blade diameter is good for suspending of $Al(OH)_3$ particles. The blade diameter has a big influence on the stirring power, and the longer blade diameter needs more power consumption, but the C has little effect on the stirring power, which can be ignored. The physical simulation results compare well with numerical simulation.

At first, we want to find the best impeller off-bottom clearance height and impeller diameter just in single impeller experiments. And in next step, we will study the multiple impellers' performance based on the results of single impeller experiments.

Introduction

Technological revolutions have recently hit the alumina industry. As one of the key steps in the Bayer process for the production of alumina, the seed precipitation step has a great influence on alumina product output and quality, and also has an indirect effect on other processes[1]. So it is necessary to optimize and improve this step. Currently, the mechanically agitated precipitation tank has become the major equipment of seed precipitation step, because it has many advantages which include low power consumption, better mixing effect and less sediment; avoiding "Short-circuiting" of slurry; high reliability, etc[2]. But the application of mechanically agitated precipitation tanks was late in China, and we have less information on the flow field in the tank.

By comparing the power consumption of a traditional Intermig impeller and a pitched blade impeller in industrial tests, the former is much lower than the latter, but using the traditional Intermig impeller produced a large amount sediment[3]. So a new-style impeller, - an improved Intermig impeller - which could reduce the sediment significantly, was devised by the Shenyang Aluminum & Magnesium Engineering & Research Institute. Power consumption also declined from 200kW to 75kW. The difference between the traditional Intermig impeller and the improved impeller has been reported in the literature[4].

In order to fully understand the impact of the structure of the blade on the solid suspension characteristics and power consumption, the seed precipitation tank with Intermig impeller was studied in our laboratory delegated by the Shenyang Aluminum & Magnesium Engineering & Research Institute.

The study includes mixing uniformity, suspension of $Al(OH)_3$ particles and power consumption, etc. By this research we intended to improve the internal structure of the seed precipitation tank, solve the problem of sedimentation, and improve the yield and quality of alumina. This has important theoretical and practical significance for the alumina industry.

Research Method

<u>Cold water experiments</u>

The profile of local solid concentrations on the water-glass bead particles in liquid-solid system, were measured by using a PC6D fiber optic reflection probe and analyzed under different conditions. The PC6D particle concentration measuring instrument is manufactured by the Institute of Process Engineering, Chinese Academy of Sciences, the measuring range is 0~800g/L. The instrument uses an optical fiber as the measuring probe. The reflected light of materials at the fiber end comes back to the photo detector in the instrument through the same fiber and is converted into a voltage signal proportional to the concentration of materials. The average concentration of solid particles is obtained from the voltage signal.

The cold water experiment is based on the similarity principle. The ratio between water model and industrial seed tank is 1:33, and the experimental equipment was made with plexiglass Φ43.5cm × 52cm. Liquid height was 35cm and material system was glass beads-water (with sugar added to change the viscosity). **Table 1** shows the comparisons of physical parameters. Glass beads were used instead of Aluminum hydroxide and an amount of 3800g added in the cold water experiment. The syrup viscosity in the cold water experiment was consistent with the viscosity of the $NaAlO_2$ solution in the seed precipitation process.

Table I Comparisons of physical parameters		
Physical Parameters	Density /kg/m^3	Viscosity /cp
Sodium Aluminate Solution	1330	3.5
Running Water	1000	1.52
Syrup	1149	3.50
Aluminum Hydroxide	2430	Granularity 100μm
Glass Beads	2380	

Numerical simulation

Some research has been done on solid – liquid phases numerical simulation[5-7], that may verify the feasibility of study on seed precipitation tank by CFD.

(1) Grid Generation In order to reduce computing time and improve the convergence and stability of solution process, we should reduce the grid number and improve the mesh quality. So we took the following methods:

A: Ignored the thickness of baffle.

B: Changed grid from a tetrahedron into a polyhedron.

Figure 1 shows the grid model of the seed precipitation tank. The grid number was about ten million.

(2) Simulation Strategies From the literature about simulation of solid-liquid phase flow[8-11], we chose the simulation strategies are as follows:

Results and analysis of cold water experiment

Experimental measurement points were set to 7.5、9.5、11.5、13.5、15.5、17.5、19.5、21.5cm in radial distance (the center of slot as the 0 point that was indicated as l); 2、6、10、14、18、22、26、30cm in axial distance (the bottom of slot as the origin that was indicated as d).

Figure 2 showed the concentrations along the radial and axial distance. A uniform distribution of particles in the tank was assumed to be the theoretical concentration. When η=1.52cp、n=96rpm, the multiple relationships of the average value of the experiment and theory was 0.962 times; when η=1.52cp、n=172rpm, it was 1.340 times; when η=3.50cp、n=172rpm, it was 1.011 times. The results showed that the Experimental concentration was closer to theoretical concentration under this condition, which was more evenly mixed.

Turbulence model: Realizable k-ε; Multiphase model: Eulerian; Interphase drag: Gidaspow; Rotate model: Multi-Reference Frame; Numerical solution: SIMPLE algorithm for pressure velocity coupling; Discretized with the second-order upwind; All Residual converges to 10^{-3}.

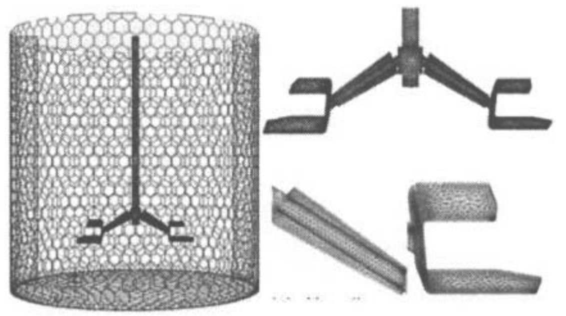


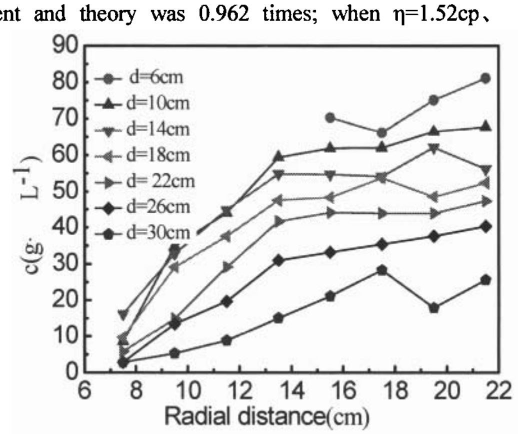
Fig.1 Grid model of the seed precipitation tank

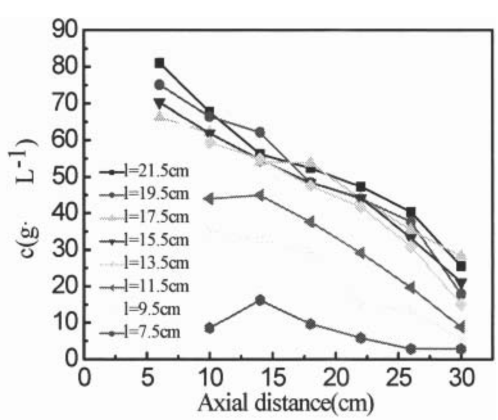
Results and analysis of numerical simulation

The numerical simulation research is for the seed precipitation process, the liquid-solid system of $NaAlO_2$ solution and $Al(OH)_3$ particles (for detail about material parameters see attached **Table 1**). At first, we validated the simulation strategy based on cold water experiments and then we found the best conditions by simulating the seed precipitation process.

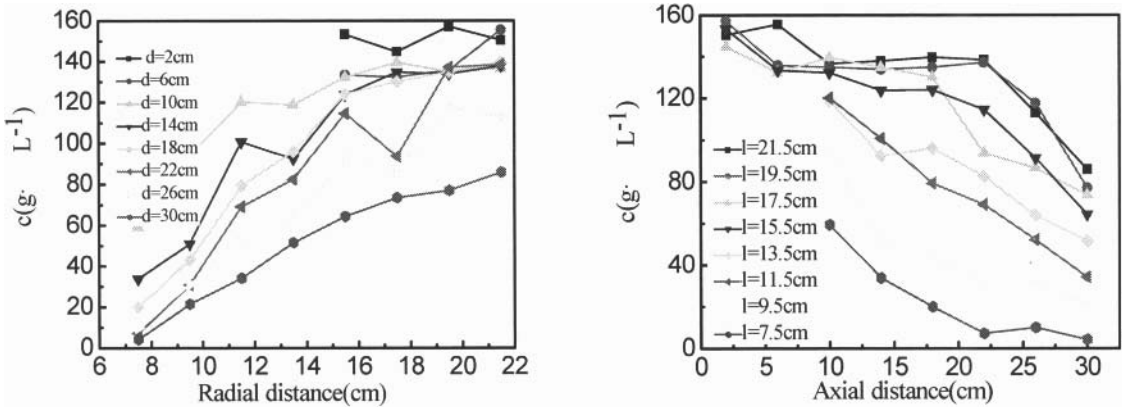
Simulation validation based on cold water experiments

In this section, the liquid-solid system was composed of water and glass beads. By comparing the results of water model experiment and numerical simulation on the solid particles concentration field, we got the following results:

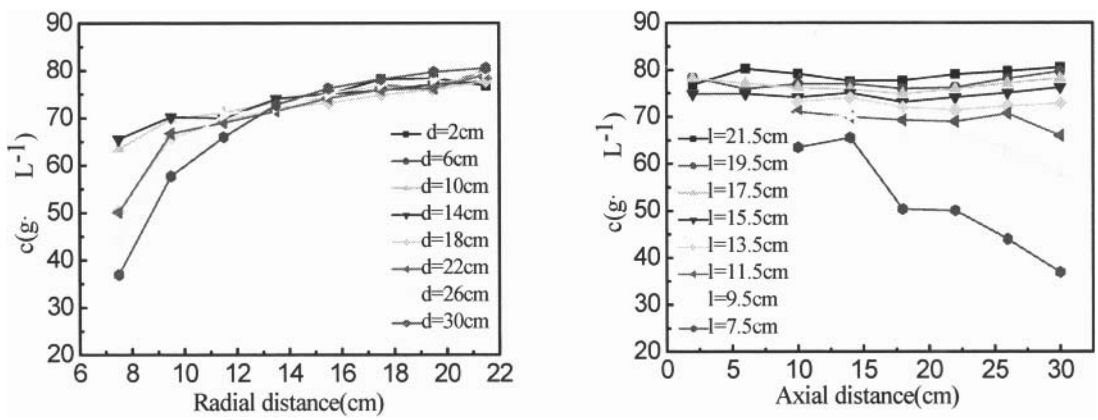




(a)η=1.52cp, n=96rpm

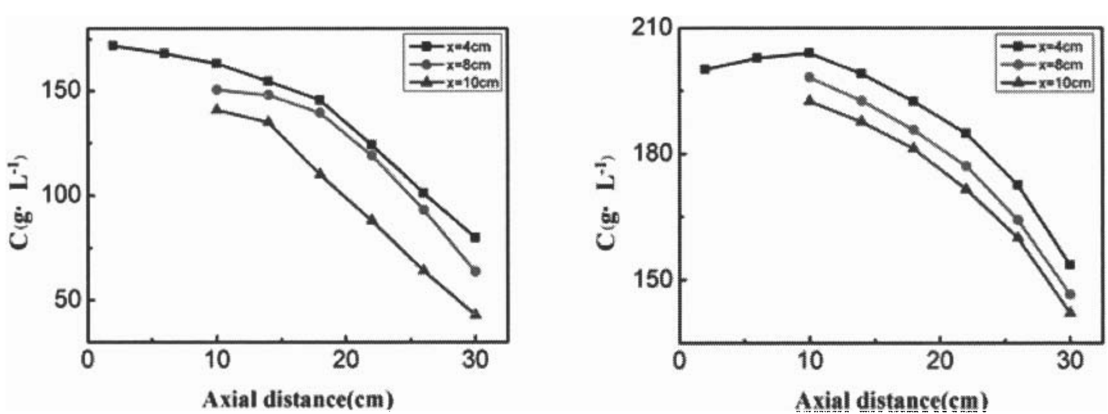


(b)η=1.52cp, n=172rpm



(c)η=3.50cp, n=172rpm

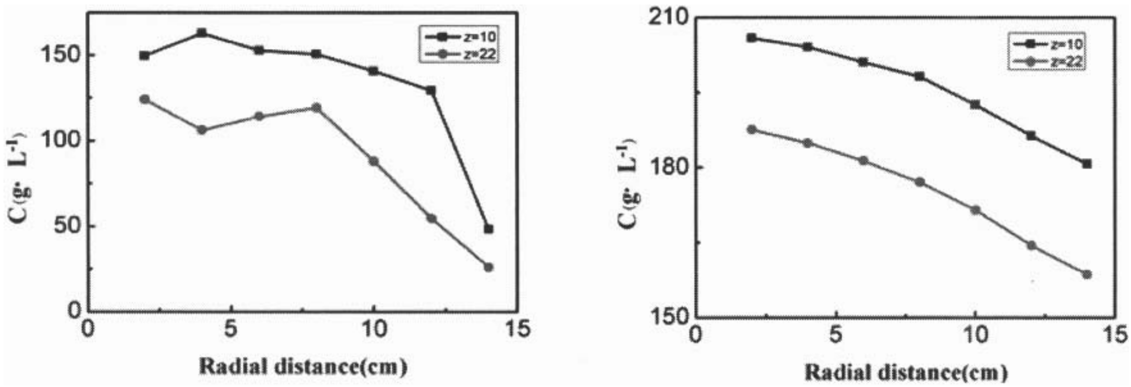
Fig.2 Concentrations along radial and axial distance by experiment



(a)Experiment results (b)Numerical simulation results

Fig.3 Axial distribution of solid particle concentration

(x is the distance between measuring points and tank wall in radial direction)



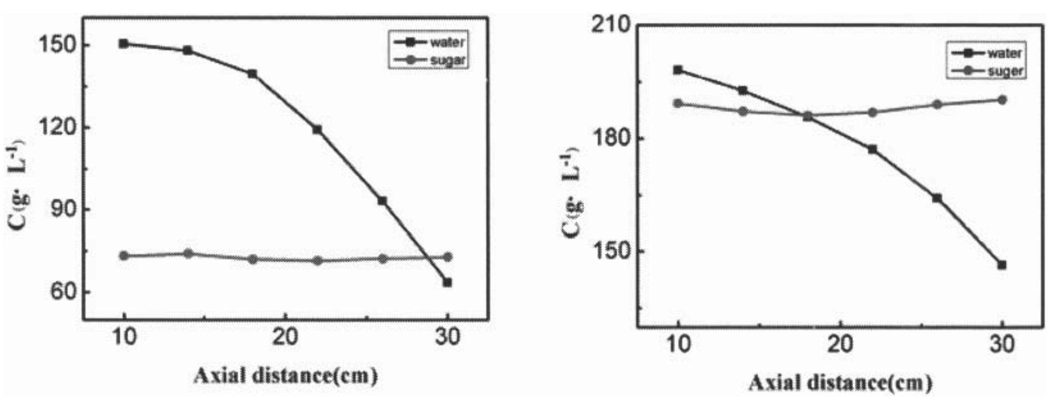
(a)Experiment results (b)Numerical simulation results

Fig.4 Radial distribution of solid particle concentration

(z is the distance between measuring points and tank bottom in radial direction)

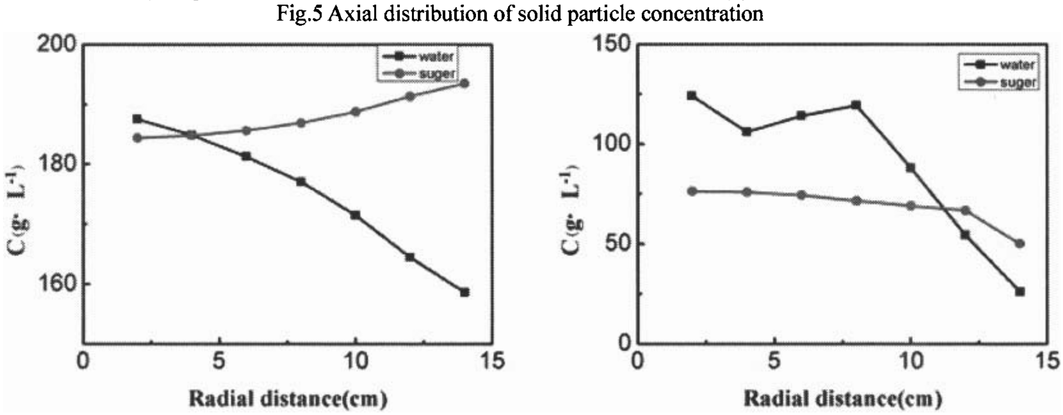
Figure 3 and Figure 4 showed that particle concentration decreased from the bottom to top surface in the axial direction (z), the particle concentration was higher closer to the tank wall in the radial direction (x). By comparing the results of the cold water experiment and numerical simulation, the latter value of particle concentration was slightly higher, but both of them had the same trend on particle concentration gradient. so the simulation strategy and results were credible.

Keeping other conditions the same, we increased the viscosity of the solution, and then we got the following results:



(a)Experiment results (b)Numerical simulation results

Fig.5 Axial distribution of solid particle concentration

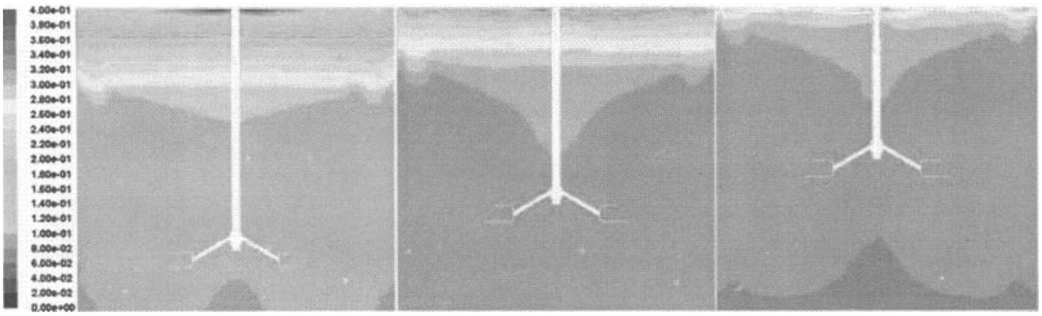


(a)Experiment results (b)Numerical simulation results

Fig.6 Radial distribution of solid particle concentration

Figure 5 and Figure 6 show that increasing solution viscosity makes let the particle concentration more uniform both in the axial and radial directions, and the simulation results were consistent with the experiment results.

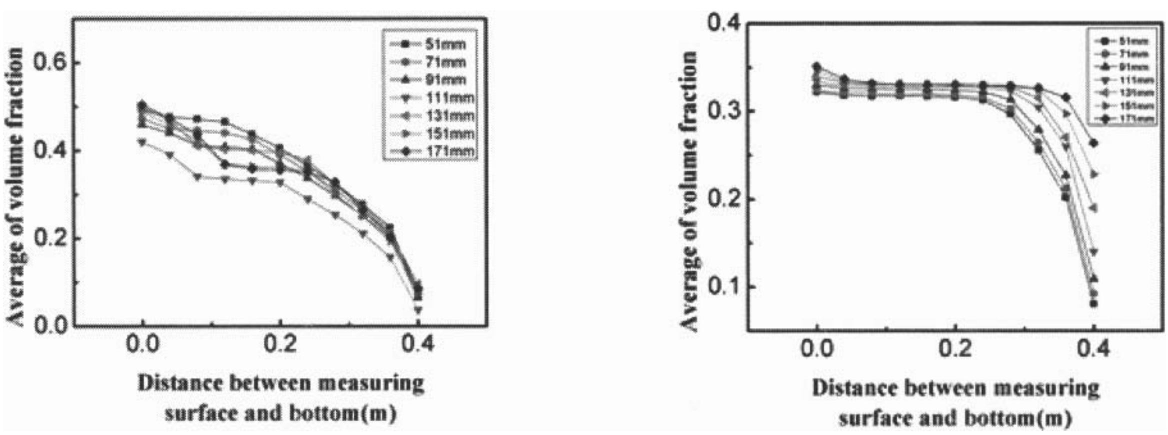
The effect of distance between the impeller and the bottom(C) on particle suspension and power consumption



(a)C=51mm (b)C=111mm (c)C=171mm

Fig.7 Concentration field with different C by numerical simulation, 150rpm

Figure 7 shows the distribution of particle volume fraction with different C. (y= 0 section). The results showed that too long or too short a value of C may lead to aggregation of $Al(OH)_3$ particles at the tank bottom. Figure 8 shows the average of particle volume fraction with high and low stirring speed.



(a)40rpm (b)150rpm

Fig.8 Average of particle volume fraction with different z sections by numerical simulation (C=51, 71, 91, 111, 131, 151, 171mm)

The results showed that C had little effect on the particle suspension at low stirring speed, while a large C may make the particle distribution more uniform in the whole tank at high stirring speeds.

The power consumption with different C at 150rpm can be seen in **Figure 9**. With the increase of C, the stirring power at first increased, and then began to stabilize, but the C had little effect on the stirring power. In this section the best value of C was 111mm.

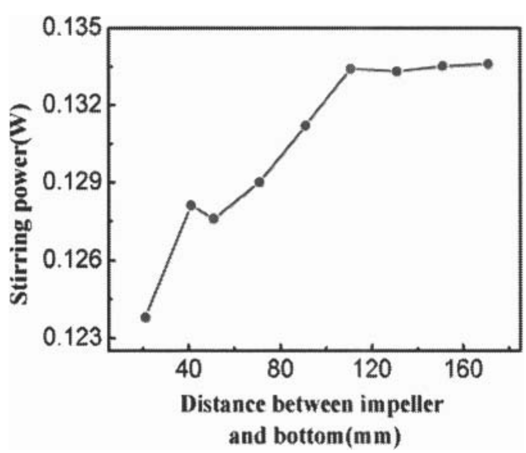
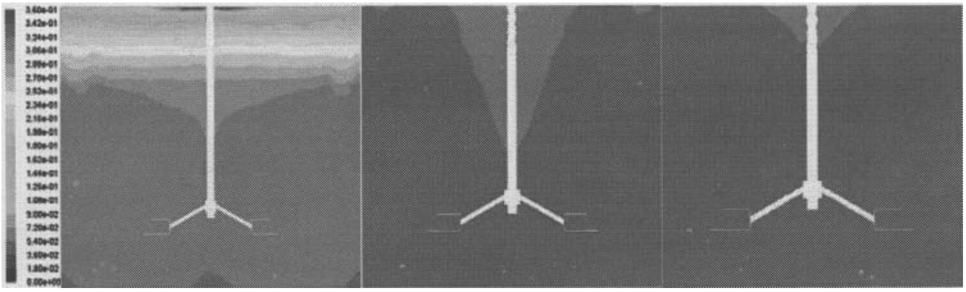


Fig.9 Power consumption with different C by numerical simulation, 150rpm

The effect of impeller diameter (D) on particle suspension and power consumption

Figure 10 shows the distribution of particle volume fraction with different D/T (T was the tank diameter, y= 0 section), and **Table 2** showed the power consumption under the same conditions.



(a)D/T=0.5 (b)D/T=0.6 (c)D/T=0.7

Fig.10 Concentration field with different D/T by numerical simulation, 150rpm

Table II Power consumption with different D/T by simulation(unit: W)			
C=71mm	D/T=0.5	D/T=0.6	D/T=0.7
40rpm	0.0472	0.127	0.258
150rpm	2.025	6.060	13.690

Figure 10 and Table II show that enlarging D may promote the suspension of $Al(OH)_3$ particles, but the power consumption increased substantially. So we should choose moderate D to get a better efficiency in particle suspension, with low power consumption. In this section the best value of D/T was 0.6.

Conclusion

(1) In terms of overall distribution, the particle concentration with radial distance increases, while the axial distance reduces. The high rotation and viscosity, high-speed particles are more conducive to uniform distribution. When η=3.50cp、n=172rpm, the multiple relationships of the average value of experiment and theory is 1.011 times. It shows that the experimental concentration is closer to theoretical concentration under this condition, indicating the particles are more evenly mixed.

(2) By comparing simulation and cold water experimental results, the feasibility of the CFD method is verified to simulate the flow and solid-liquid mixing characteristics.

(3) At a constant stirring speed, there is an optimum value of D and C which can get a better effect of particle suspension with low power consumption. In this study, the best values of C and D/T are 111mm (C/T=0.55) and 0.6 respectively.

Acknowledgement

The authors acknowledge the support of the National Natural Science Foundation of China (NO.50974035, 51074047).

References

1. Yang Zhongyu. *Alumina Production Technology*, (Beijing: Metallurgical Industry Press, 1993.10),133.

2. Zhang Chunyang, Fu Yan. *Alumina Plant Design*, (Beijing: Metallurgical Industry Press, 2008),115-141.

3. Wu Jianqiang, Dong Baocai, "Evaluation of Operating Results of Large-scale Precipitation Tanks with Mechanical Agitators", *Linght Metals*, 2000(03).

4. Wang Rui, "Numerical Simulation of Multi-Phase Flow in the Alumina Seed Precipitator" *PhD Thesis of Northeastern University, Shenyang, China*, 2009:29-30.

5. G. Montante, "Experiments and CFD Predictions of Solid Particle Distribution in a Vessel Agitated with Four Pitched Blade Turbines", *Chemical Engineering Research and Design*, 2000, 79(8):1006-1010.

6. R. J. Weetman, "Automated Sliding Mesh CFD Computations for Fluidfoil impellers", *9th Euro.Conf.On Mixing*, 1997:195-202.

7. I. Naude, C. Xuereb, J. Bertrand, "Direct Prediction of the Flows Induced By a Propeller in an Agitated Vessel Using an Unstructured Mesh", *Can Chem. Eng.*, 1998, 76:631-640.

8. Montante G, "Experiments and CFD Predictions of Solid Particle Distribution in a Vessel Agitated with Four Pitched Blade Turbines", *Chemical Engineering Research and Design*, 2000, 79(8):1006-1010.

9. Weetman R J, "Automated Sliding Mesh CFD Computations for Fluidfoil Impellers", *9th Euro. Conf. on Mixing*, 1997:195-202.

10. Naude I, Xuereb C, Bertrand J, "Direct Prediction of the Flows Induced By a Propeller in an Agitated Vessel Using an Unstructured Mesh", *Can Chem. Eng.*, 1998, 76:631-640.

11. Bakker A, Fasano J B, Myers K J, "Effects of Flow Pattern on the Solids Distribution in a Stirred Tank"(8th European Conference on Mixing, Cambridge, U.K), *IChemE Symposium*, 1994, 136(329):1-8.