Ninth International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2012

COMPUTATIONAL FLUID DYNAMIC (CFD) SIMULATION OF FLUID FLOW IN A MIXER-SETTLER EXTRACTOR FOR RARE EARTH METAL SEPARATION

Yang ZOU[1], Tingting MA[1,2], Yundong WANG[1*] , Jinsheng SUN[2] and Weiyang FEI[1]

[1]The State Key Laboratory of Chemical Engineering, Department of Chemical Engineering,
Tsinghua University, Beijing 100084, P. R. China
[2]School of Chemical Engineering and Technology, Tianjin University, Tianjin 300072, P. R. China
*Corresponding author, E-mail address: wangyd@tsinghua.edu.cn

ABSTRACT

Computational fluid dynamic (CFD) simulation is an effective research approach for understanding velocity fields used in process and equipment intensification. It was found that mixer-settler used in rare earth metal extraction process was unsatisfying in large amount of liquid backlogged, high energy consumption and long-time settling process. In this paper, CFD simulations and experiment validation were carried out to study the flow field in a mixer-settler for rare earth metal separation with analysis of the effect of stirring intensity and feed rate on mixing process. Results show that large area of low velocity appears at the corner of mixer and could not be dispelled by the increase of impeller speed. Flow rate has little impact on the flow field in mixer and the agitation of the impeller has low efficiency, especially at the top of mixer. Furthermore, the effect of internal components on flow field in the mixer was analysed, such as four baffles and a plate covered on the top of mixer. Results show that flow field changes not only in magnitude but also in flow pattern. Further researches need to be carried out to test the improvement of flow fields in mixer.

NOMENCLATURE

L	liquid flow rate (L/min)
N	impeller speed (r/min)
$\mathbf{u}$	velocity (m/s)
k	turbulence kinetic energy (m^2/s^2)
ε	dissipation rate (m^2/s^3)
X	X direction position of the mixer-settler (m)
Y	Y direction position of the mixer-settler (m)
Z	Z direction position of the mixer-settler (m)

INTRODUCTION

Computational fluid dynamic (CFD) simulation is an effective research approach for understanding velocity fields used in process and equipment intensification, especially after the fast development of computer hardware and software. In this paper, CFD simulation is focused on the velocity fields in the mixer-settler, which is the most widely used equipment in metal extraction with advantages of high unit efficiency, resilient operation, easy scale-up and controllability. For the sake of reducing the usage of land and costly feed, many new types of mixer-settler have been proposed recently (Michae, 2006, A & J, 2005, Hadjiev & Paulo, 2005, David, Alain & Claude, 1995). However, it was found in the plant that mixer-settler used in rare earth metal extraction process, which improved little during the last decades, was unsatisfying in large amount of liquid backlogged, high energy consumption and long-time settling process.

Few researchers focused on the flow field of mixer-settler during the decades for that a large number of researches have been done to investigate the flow field in stirred tank(Joshi, Nere, Rane, Murthy, Mathpati, Patwardhan & Ranade, 2011, ZhouWang & Shi, 2003, Rao, Fan, Wang & Fei, 2004), which has similar geometry structure, stirring condition and function compared to mixer-settler. However, mixer-settler has inlets, outlets and is operated continuously while the stirred tank is operated as batch. With increasing of axial velocity in the mixer, we can infer that research on flow field in mixer-settler is significant to improve the mixing and mass transfer efficiency in rare earth extraction.

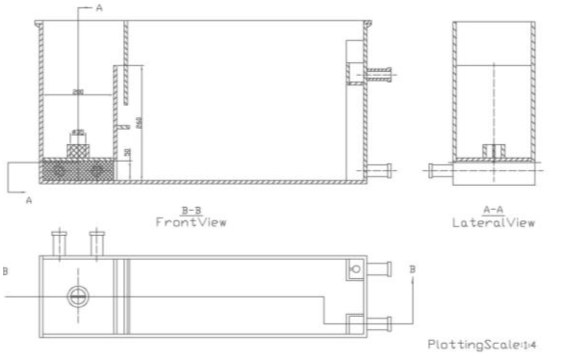


Figure 1: Three-view drawing of mixer-settler

A recent research studied the CFD steady state simulations of double liquid phases in the mixer, which tested the effect of different width of impellers, impeller speed inlet velocity and impeller diameter on consumption power, mixing and separation efficiency (ShabaniAlizadeh & Mazahery, 2011). As a matter of fact, the flow field of double liquid phases in a mixer-settler could not be static but changes every moment, especially the position of liquid-liquid interface. As the difficulty in CFD simulation of double liquid phases, single phase CFD simulation in mixer-settler, which is few reported in the literature, is still significant to analysis the flow field in mixer-settler.

In this paper, CFD simulations were carried out to study the single phase flow field in a mixer-settler for rare earth metal separation with analysis in the effect of stirring intensity and feed rate on mixing process. As a confirmation of CFD results, Particle Image Velocity (PIV) equipment, which showed a great performance in measuring the flow field in industrial contactor (Drumm Hlawitschka & Bart, 2011, Drumm & Bart, 2006, Rao, Fan, Wang & Fei, 2004), was used in the experiments.

Figure 2: Experiment setup and impeller

Furthermore, according to the results of stirred tank, the added baffles could increase the relative stirring velocity between the agitator and fluid and enhance the radial mixing, resulting in an improvement of mixing efficiency. In this paper, the effect of internal components on flow field in the mixer was analysed, such as four baffles and a plate covered on the top of mixer.

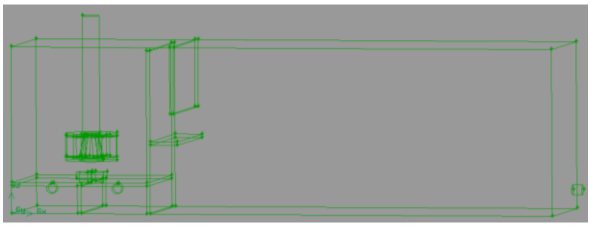


Figure 3: Geometry structure of the mixer-settler in CFD

EXPERIMENT SETUP

Experiments were carried out to validate the results of CFD simulation. Figure 1 shows the three-view drawing of a mixer-settler used in the research with dimensioning, which was made according to the industrial facility. Two water inlets were located at one side of the submersible chamber and two outlets at the end of the settler. The cubic mixer had a size of 200*200*200mm^3 and the settler was three times as long as the mixer. A weir was used in the settler to control the water level.

Figure 2 shows the whole experiment setup and the impeller. Water was injected from a tank into the submersible chamber through each inlet by a peristaltic pump, which had a maximum flow rate of 4.5L/min. Two flows of water got mixed in the mixer, where a vertical six-flat-blade disc turbine impeller was rotating to improve the mixing. The motor connected to the impeller was controlled by a transducer, which could adjust the agitation speed by changing frequency of the electricity. After mixing, water flowed into the settler and left through two outlets.

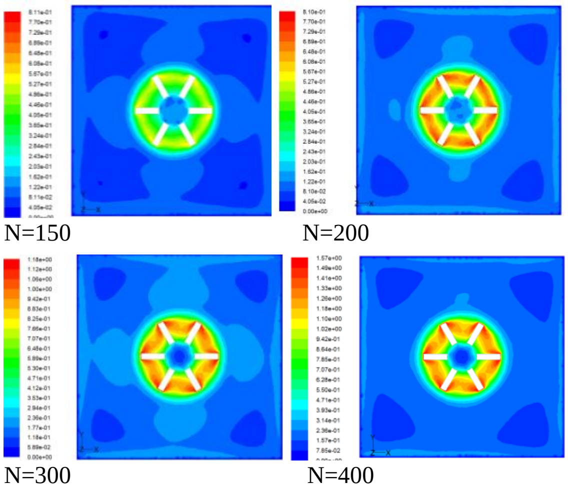


Figure 4: CFD results of velocity profile at section Z=65mm

PIV equipment, comprised of two cameras, a laser and a computer, was set to measure the flow field of a specified plane in the mixer. Trace particle used in the experiment was made of polystyrene, which was insoluble in water and had a density of 1.05g/cm^3 and a diameter of 10μm. As the rotating impeller reflected laser to any direction, which affected the cameras to distinguish luminous particles, section X=48mm and section Y=53mm were chosen as the measured plane, where impeller could not reach. Every measurement of PIV started at 10 minutes after the steady operating of the setup.

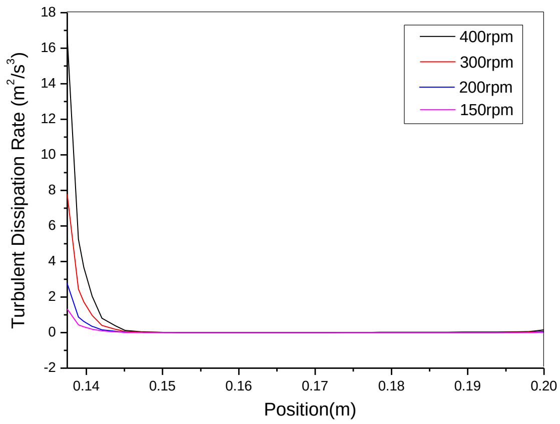


Figure 5: CFD results of turbulent dissipation rate at section Z=65mm versus X position

Moreover, four baffles with a width of 12mm and a plate were added to the mixer as a comparison. The plate, with a hole of 120mm in the centre, was covered on the top of the mixer and was supposed to increase the residence time and mass transfer efficiency.

MODEL DESCRIPTION

To further observe the flow pattern in the mixer settler, CFD simulations were carried out in the software Fluent

6.3. Figure 3 shows the geometry structure used in CFD simulations. This geometry, similar to the experiment setup, was generated by Gambit 2.4. The difference between the geometry and entity lay in the height of the setup. In the experiments, air occupied the upper of the mixer-settler and water occupied the underpart, where formed an almost steady air-liquid interface. As an approximation, only the underpart of the mixer-settler was chosen to simulate, which made the simulations easier.

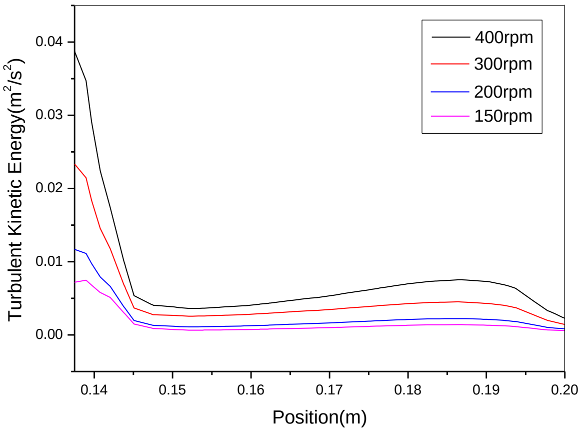


Figure 6: CFD results of turbulent kinetic energy at section Z=65mm versus X position

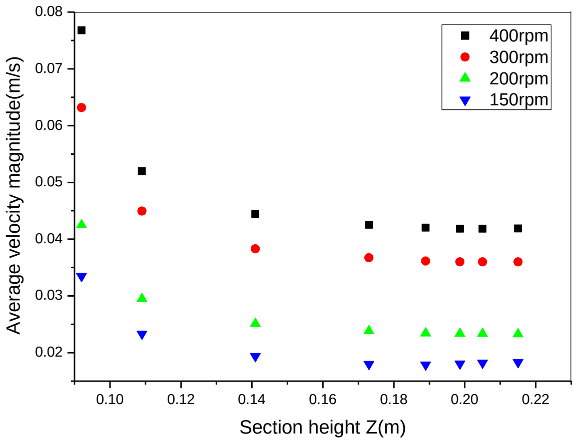
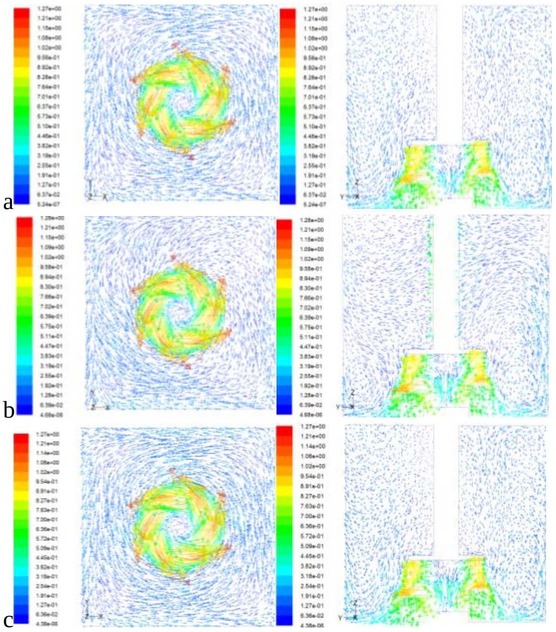


Figure 7: CFD results of average velocity magnitude at versus Z position

As the vertical six-flat-blade disc turbine impeller rotated in the mixer, the geometry was divided into two volumes, the inner volume containing the impeller and the static volume which was composed of the remaining part of the mixer-settler with two inlets, outlet and the whole settler. About 5.8 million unstructured T-grid and structured grid cells were used for meshing the geometry. No slip conditions were set for all stationary walls. The velocity of inlets could be given by the different volume flow. Zero pressure condition was specified at the outlet. Standard k-ε model was chosen as the turbulence model. Moving reference frame approach was adopted to calculate the whole geometry. In this approach, the fluid of the inner volume, which was rotating with the impeller, was simulated in relative velocity formulation and the fluid of static volume, which was moving at a lower relative velocity and absolute velocity, was solved by the continuity and momentum equations.

In all simulations, without consideration of mass transfer and chemical reactions, only one discretization scheme was used for the momentum, turbulent kinetic energy and energy dissipation. SIMPLE method was used for the pressure-velocity coupling. The simulations were supposed to have been converged when the residuals of continuity, velocity components and turbulence kinetic energy and energy dissipation rate fell down below 10^{-4}.
With parameters introduced above, CFD simulations were carried out under different stirring intensity and feed rate in mixing process.



section Z=65mm section X=100mm

Figure 8: CFD results of velocity fields at N=300r/min, different section and liquid flow rate: (a) L=3L/min; (b) L=4.5L/min; (c) L=6L/min

RESULTS

CFD simulations were carried out under different combination of impeller speed and inlet flow rate. Flow fields, kinetic energy and turbulent dissipation rate were compared under different conditions. Experiments were performed to validate the CFD simulations and internal components were added to change the flow field in the mixer, such as the baffles on the wall and a plate at the top of mixer.

Effects of impeller speed

Figure 4 shows the velocity profile of CFD results at section Z=65mm, the horizontal plane at the middle of impeller, at different impeller speed and liquid flow rate L=6L/min. Velocity profiles at X/Y section under different impeller speed are not illustrated here for that they are almost the same. A region of low velocity appears at the corner of mixer and could not be dispelled by increasing impeller speed. Slower movement in this region causes less mass transfer between liquids and reduces the unit extraction efficiency.
Figure 5 shows the turbulent dissipation rate condition of CFD results at section Z=65mm, from the edge of impeller (X=137.5mm) to the wall (X=200mm). Dissipation rate decreases quickly around the edge of impeller and comes to an almost steady value close to 0. Furthermore, a

stronger agitation engenders a faster decrease of dissipation rate.
Figure 6 shows the turbulent kinetic energy condition of CFD results at Z=65mm, from the edge of impeller (X=137.5mm) to wall (X=200mm). Kinetic energy decreases quickly around the edge of impeller and then becomes almost steady. A stronger agitation engenders a faster decrease of kinetic energy.

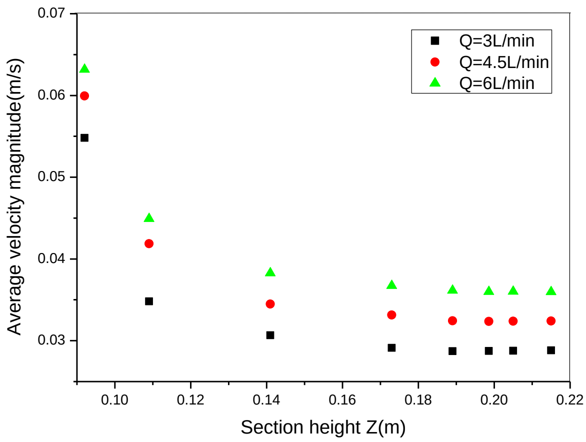


Figure 9: CFD results of average velocity magnitude versus section height Z

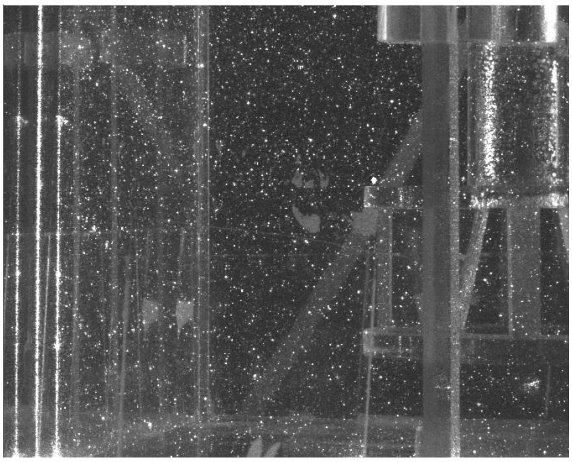


Figure 10: Original photo captured by camera of PIV equipment

Figure 7 shows the average velocity magnitude condition of CFD results from the top of impeller (Z=90mm) to the top of mixer (Z=220mm). Average velocity decreases quickly around the impeller and comes to a steady small value at the top of mixer. The fluid far away from the impeller flows at a low speed compared to that around impeller, which means low agitation efficiency. It is clear that geometry and operation parameter under the condition could not generate a homogeneous, fast and efficient flow field. Moreover, a stronger agitation engenders few increase of average velocity magnitude, especially when the impeller speed goes up to N=400r/min.

Effects of flow rate

Figure 8 shows the velocity fields of CFD results at certain impeller speed N=300r/min, different flow rate and section position. Obviously liquid flow rate hardly affects velocity fields around impeller, no matter the magnitude or the overall flow pattern. Moreover, strong rotational flow turns up throughout the whole mixer, which speeds up the fluid to leave mixer without sufficient mass transfer.

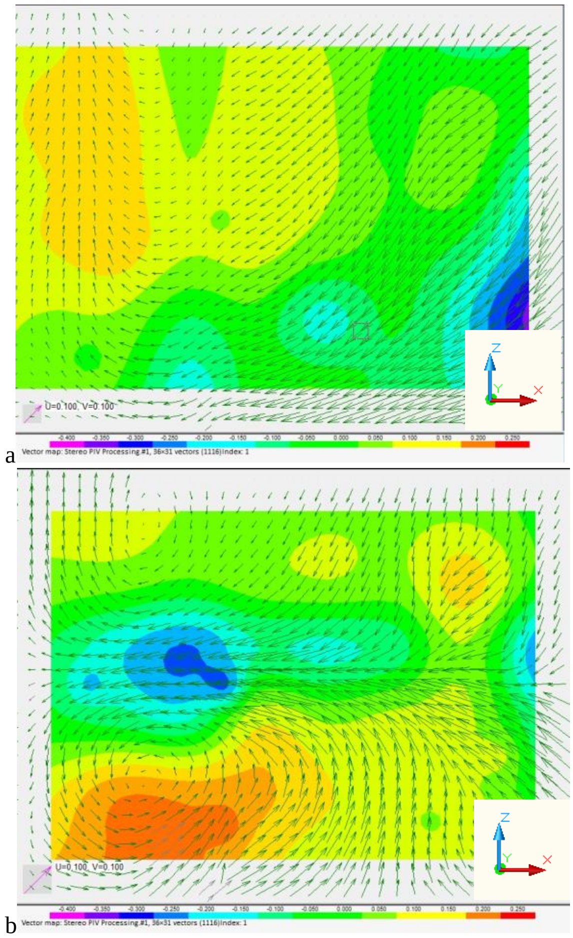


Figure 11: Effects of the addition of baffle and the plate on velocity profile at section Y=53mm: (a) no internal component; (b) with baffles and plate

Figure 9 shows the average velocity magnitude of CFD results from the top of impeller (Z=90mm) to the top of mixer (Z=220mm) at certain impeller speed N=300r/min and different flow rate. Average velocity decreases quickly around the impeller and comes to a steady value at the top of mixer, which is the same with results in figure 6. Furthermore, as the tangential velocity is much larger than axial velocity, a larger flow rate, which engenders a larger axial velocity, causes little increase of total velocity magnitude in the top of mixer.

Effects of internal components

Internal components could improves the flow fields rapidly according to the results of stirred tank. In the experiments, four baffles were located in the middle of each wall in the mixer to decrease rotational flow and a plate with a hole in the centre was set at the top of mixer to prevent the feed from leaving mixer immediately after agitation.
Figure 10 shows an original photo captured by camera of PIV at Y=53mm. Impeller was located at the right sight of the camera and the bright spot in the picture is trace particle, which reflects laser to the cameras. The velocity of the fluid was calculated by the movement of trace particle, in which two photos with an interval of 1000μs were captured. As the instantaneous flow fields were not

steady but changed in regular, 400 photos were captured by the cameras to measure the time average velocity.

Figure 11 shows the velocity profile after time average treatment of 400 pictures with and without internal components at section Y=53mm. The colourful background means the velocity at Y direction and the reference velocity of X/Z direction lies at the left bottom in the picture. With the addition of baffles and the plate, flow field changes a lot not only in magnitude but also in flow pattern. We can infer that the addition of proper internal components may improve the flow fields in mixer, which needs a further confirmation in the future.

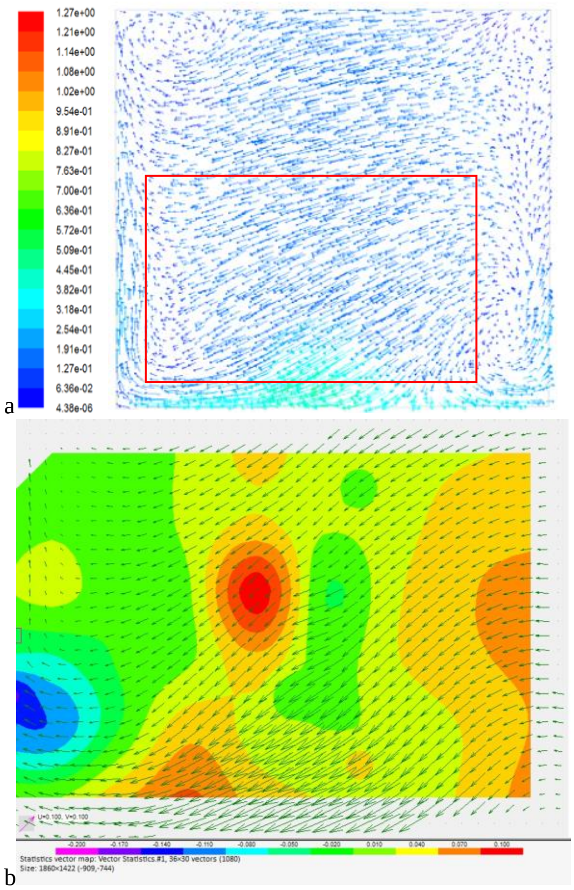


Figure 12: Comparison of velocity profile at section X=48: (a) simulation result; (b) experimental result

Validation of CFD simulation results

In order to confirm the CFD simulation results, experiments were carried out at N=300 and L=4.5L/min without internal components. The difference of velocity field between experiment and simulation at section X=48mm were investigated.

Figure 12(a) shows the velocity profile of simulation at section X=48mm. The velocity profile in red box was measured in PIV experiments, shown in figure 12(b). Judged by the flow pattern, they are almost the same. Furthermore, the largest velocity magnitude of experiment in the profile, which lies at the middle of the bottom, was 0.376m/s. This value agrees well with that in CFD results of 0.382-0.445m/s. In a word, It is obvious that the CFD simulation result has an good agreement with that in experiments.

CONCLUSION

With the validation of PIV measurement, CFD simulation is a reliable approach to study the flow fields in mixer-settler. According to the CFD simulation results, impeller used in industrial facility is operated with high energy dissipation and low agitation efficiency. The region of low velocity does not disappear as the increase of impeller speed, while larger energy dissipation occurs around the impeller. Flow rate has little impaction on the flow field in mixer. Internal components added in the mixer could change the flow field pattern largely, which is possible to find a proper structure that provides a more homogeneous and efficient flow field.

REFERENCES

A, G. M. and J, P. M., (2005), “Cylindrical mixer-settler apparatus and method”, *P.A.*, US20050041197.

DAVID, G., ALAIN, S. and CLAUDE, P. J., (1995), "Method and apparatus for mixing/separating two non-miscible liquids”, *P.A.*, FR19950013138.

DRUMM, C. and BART, H. J., (2006), “Hydrodynamics in a RDC extractor: Single and two-phase PIV measurements and CFD simulations”, *Chem. Eng. Technol.*, **29**, 1297-1302.

DRUMM, C., HLAWITSCHKA, M. W. and BART, H. J., (2011), “CFD simulations and particle image velocimetry measurements in an industrial scale rotating disc contactor”, *Aiche J.*, **57**, 10-26.

HADJIEV, D. and PAULO, J., (2005), “Extraction separation in mixer-settlers based on phase inversion”, *Sep. Purif. Technol.*, **43**, 257-262.

JOSHI, J. B., NERE, N. K., RANE, C. V., MURTHY, B. N., MATHPATI, C. S., PATWARDHAN, A. W., et al, (2011), “CFD simulation of stirred tanks: comparison of turbulence models. Part i: radial flow impellers”, *Can. J. Chem. Eng.*, **89**, 23-82.

MICHAE, G. B., (2006), “Solvent extraction method and apparatus”, *P.A.*, US20060354983.

RAO, Q., FAN, J., WANG, Y. and FEI, W., (2004), “DPIV measurement and CFD simulation of viscous fluid flow in stirred tank agitated by Rushton turbine”, *Huagong Xuebao/Journal of Chemical Industry and Engineering (China)*, **55**, 1374-1379.

SHABANI, M. O., ALIZADEH, M. and MAZAHERY, A., (2011), “Fluid flow characterization of liquid-liquid mixing in mixer-settler”, *Eng. Comput. -Germany*, **27**, 373-379.

ZHOU, G., WANG, Y. and SHI, L., (2003), “CFD study of mixing process in stirred tank”, *Huagong Xuebao/Journal of Chemical Industry and Engineering (China)*, **54**, 886-890.

