Numerical investigation on the gasoline mixture with side entering mechanical agitator

***X. X. Tang, Y. P. Qu, †S. Y. Chen, J. Y. Pan and D. D. Wan**

Key Laboratory of High-efficiency and Clean Mechanical Manufacture, School of Mechanical Engineering, Shandong University, China

*Presenting author:1245061506@qq.com
†Corresponding author: chensy66@sdu.edu.cn

Abstract

Employing the standard k-e turbulence model and sliding mesh technique, Fluent software was used to simulate the mixing flow peculiarity of the gasoline blending in the gasoline tank with installing side entering agitators. Simulation results show that high speed region is located in the vicinity of the Z=0.126 m plane paddle, where the speed can reach to 10.02 m/s. After continuous attenuation, the speed can reach to 0.35 m/s near the wall eventually, which meets the gasoline blending requirements according to the national standards. The stirring axis has a downward inclination with the horizontal plane in order to optimize the flow field in the middle and lower part of the tank further.

Keywords: Gasoline blending; Numerical simulation; Multiphase flow; Side entering agitator.

Introduction

As a main part of refinery production, gasoline from different refine processes should be blended together according with the national standards. The most widely used mixing equipment currently is the entering agitator installed in the gasoline tank. Especially in large storage tank, inputting a small power in the tank can get a relatively better effect [1]-[4]. The theoretical study of the flow field in the tank is very complicated and it is also very difficult to understand the distribution of the flow field in the tank by the test. In recent years, with the development of computational fluid dynamics (CFD) technology, numerical simulation has been widely used to study the characteristics of flow in gasoline tank, in which can shorten the research time and save funds. As a new means of scientific research, researchers pay more attention to the numerical investigation than before [5]-[6].

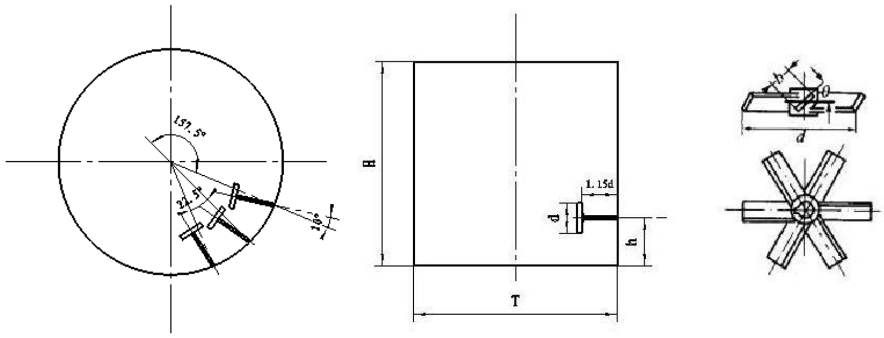
In the past 20 years, scholars have made further progress in the research of side entry mixing system. In 2004, Asghar [7] et al used the RNG k-e turbulence model to compute the flow field and mixing time of two different kinds of gasoline under the operation of single side entry agitator, of which the results are in agreement with the experimental results. In 2005, Janz [8] et al studied the agitating shaft’s angle of the side entering agitator. In 2007, Saeed [9] et al simulated the power and flow parameters distribution in the pulping mixing tank with a single side entry agitator in the flue gas desulfurization system of thermal power plant. In 2010, Gomez [10] et al carried out the numerical simulation on the side entry mixing tank under laminar flow condition and compared the results with PIV. The numerical simulation of the single-phase and multi-phase flow field in the side entry stirred reactor with different installation angle is carried out by Zhang [11] et al in 2013. In 2015, Liang [12] et al studied the liquid phase concentration distribution and the effect of particle diameter on the critical suspension speed of large double-layer side entry tank at different rotational speeds, and in the same year, Song [13] et al studied the stirring effect in the fermentation tank on the three-dimensional flow of solid-liquid (sludge and water) two-phase and obtained the velocity and concentration distribution in the tank.

In this paper, Fluent software was used to simulate the internal flow field of the mixing system so as to study the mixing peculiarity of multiphase gasoline ,which would have the guiding significance for the energy saving and emission reduction, the engineering application and the optimization design of the stirred reactor.

1 Calculation Model

1.1 *geometric model*

The engineering model adopted in this paper is based on the G408 gasoline blending tank which is owned by a Refining & Chemical Co., Ltd. Tank height is 14 m, the diameter is 22 m, nominal volume is 5000 m^3, safe height is 12.4 m, and the model is shown in Figure 1 (b). The stirrer is fixedly inserted into the angle-type lateral extension, whose arrangement is concentrated. Three agitators are installed as shown in Figure 1 (a). The angle of the axis of each stirrer is 22.5°, and the center line of the stirrer is deviated from the center line of the tank, whose angle is β. Three agitators are installed with a down angle. According to results of Liu [14], we takes the arrangement with the smallest power consumption, where the β is 10°. For the purpose of simplifying the model, we use the three-leaf propeller paddle type whose folding angle is 24°. In the case of ensuring the velocity of majority of the low-speed zones is not lower than the minimum mixing speed of 0.3 m/s, the similarity criterion of equal mixing power per unit volume is used for the side mixing model. Referring to agitator selection method from the "Mixing Equipment" [15], we determine the basic geometry of the model as follows: Inner diameter T is 1000 mm, height H is 1000 mm, and the arrangement of the blades is horizontal, lateral mounting height h is 126 mm, the diameter of the stirrer d is 100 mm, the diameter of the stirring shaft is 10 mm, the size of the hub is such as H15mm x 15mm, width of the blade is 15 mm, thickness is 2 mm, which is shown in Figure 1.

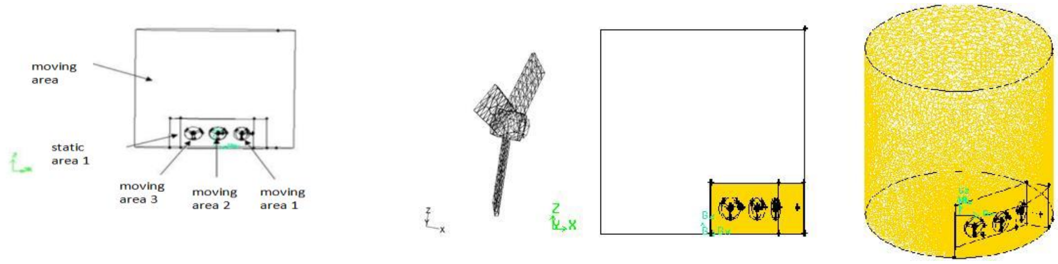


(a) (b) (c)

Figure 1. Geometric models: (a) Agitator arrangement orientation; (b) Blending tank model; (c) Schematic diagram of three leaf open turbine agitator

1.2 *Meshing and boundary conditions*

Gambit software is used to generate model grids [16]. As the agitator rotating, there are three moving and two static regions, which are shown in Figure 2 (a). Additionally, the three agitator positions are concentrated. Therefore, we block the model for meshing by using Tet/Hybrid scheme. The size of the boundary of the movement area is H 20 mm x 110 mm, the grids of motion region and the nearby static area are in local refinement, the total number of grid cells is 865817, as shown in Figure 2 (b).



（a） （b）

Figure 2． Grid division (a) Five regions of the mixing model; (b) Stirred tank area grid map

The boundary conditions for the side agitation model are set as follows, the top of the mixing tank is a free liquid surface, whose type is selected as Symmetry, and the relative motion of the stirring blade and the multiphase fluid in the tank is carried out by moving meshing method [17]. The rotational velocity of the three agitating blades is zero relative to the three corresponding moving regions, and the rotation axis remains the same. The model block interface is defined as the exchange wall Interface, and the remaining walls default to no slip standard boundary Wall.

1.3 *Simulation of working conditions and calculation strategies*

In the gasoline refining industry, a brand of gasoline is composed of the following four components [18], reforming gasoline, MTBE, catalytic gasoline and alkylated gasolines. So, there are four phases in the side agitation model, which is shown in Table 1. The density and viscosity of each in the table are measured at 20° C. In the initial state, the four components are assumed to fill the entire blending tank in a layered form， and under the action of gravity, the first phase of the maximum density (reformed gasoline) is located at the tank bottom. Figure 3 shows the initial stratification distribution of each phase. Reynolds number of the fourth phase [19] is,

$$\mathrm{Re} = \frac{\rho n d^2}{\mu} \qquad (1)$$

Where ρ is density, kg/m^3； n is the speed of agitator, r/s； d is the blade diameter， m； μ is fluid dynamic viscosity, Pa•s.

Table 1. Five kinds of physical parameters and distribution region in stirred tank model

components	density (kg/m ^{3})	viscosity （Pa•s）	Z distribution region （m）
Reformed gasoline	860.7	0.0004	0-0.22
MTBE	741.3	0.00036	0.22-0.32
Catalytic gasoline	733.2	0.00035	0.32-0.9
Alkylation	699.3	0.00034	0.9-1.0

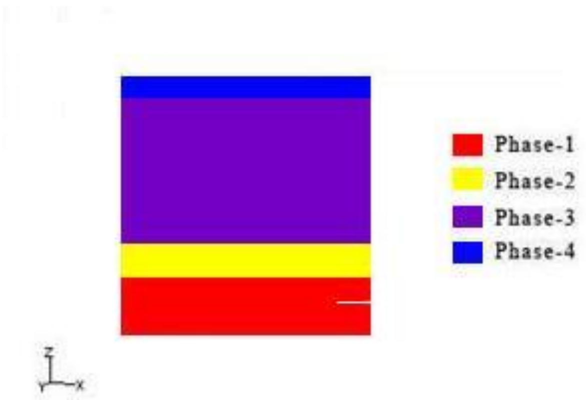


Figure 3. Initial distribution map of four phases

The Reynolds number of the region with the lowest density in the mixed flow field is 3.81x10^5. The mixing flow field is in turbulent state and the turbulence equation needs to be opened [20].

The steady-state implicit separation solver based on pressure is used for the side-mixing model. The pressure velocity coupling is solved by SIMPLE method [21], momentum, kinetic energy, dissipation rate is solved by Second Order Upwind scheme. The Eulerian mode is used in multiphase flow, the turbulence model is the standard k-e model where the convergence of all variables is $1x10^{-5}$. After the velocity converges, the unsteady solver is used to calculate the volume distribution function. The function is discretized by QUICK format and the rest is selected by default. The working pressure is set on the cylindrical surface at the top of the mixing tank, and the pressure value is 101325Pa [22]. As we open the gravity acceleration option, the direction is -Z.

According to GB/T 4756-98 "Gasoline liquid manual sampling method" [16], in the tank model, three points of A(0.3,0,0.1), B(0.3,0,0.5) and C(0.3,0,0.9) were selected as the density monitoring points, and the mixing time is,

$$t_{95\%} = \text{time for } \left| \frac{d - \bar{d}}{\bar{d}} \right| \leq 0.003 \qquad (2)$$

Where $\bar{d}$ is the equilibrium concentration (when mixing is complete), d is the concentration measured at the monitoring point.

We need real-time monitoring of mixed phase density changes. When the relative density value of A, B, C in the mixed phase reaches 0.003 in the vertical direction, the mixed gasoline is uniform.

2. Simulation results and discussions

With the start of the stirrer, the flow velocity is increasing. After the iteration of 800 time steps, the agitator has a very good flow field range. The area with larger velocity of flow passes through the direction of middle diameter of blending tank, then returns back when the fluid hits the wall. After returning to the vicinity of the stirring blade, the velocity is further increased to complete a process cycle as shown in Figure 4.

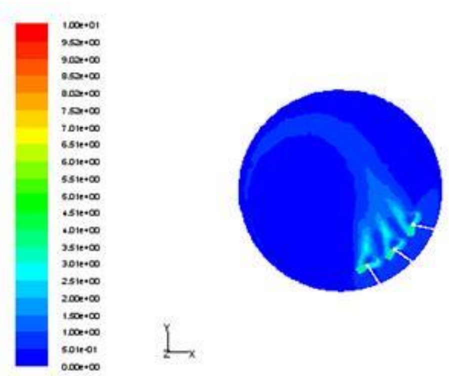


Figure 4. Velocity distribution of flow field at Z=0.126m

From the X-axis positive direction, the three agitators are in the clockwise rotation. From the Z-axis direction, the entire flow field will show a trend of counter-clockwise rotation. The velocity of the three agitators is superimposed on each other, and the liquid flows in the right direction of the opposite side. After the fluid hitting the tank wall, a vortex returned near the agitator, which is shown in Figure 5. As the centerline of the stirrer has a left angle to the centerline of the tank, the contact area between the reflux field and the stirring paddle is larger [23], further increasing the flow velocity and forming a new cycle. The mounting angle of the three sides is intertwined with the blade mounting angle, which expands the flow of the agitator. From Figure 5, the radial flow effect of the stirring blade is very good, which directly causes the axial flow of the mixing tank and the multi-phase mixing in the vertical direction, which is the main source of power to break the initial phase and stratification.

From Figure 6, the axial flow of the side agitator is very significant. The high velocity region of the mixing flow is located in the vicinity of the stirring blade on the Z=0.126 m plane, where the velocity can reach to 10.02 m/s. After continuous attenuation, it can reach to 0.35 m/s at the wall in order to meet the petrol blending speed requirements. The location of the low velocity zone on the four horizontal sections is not the same, and the velocity distribution is similar and uniform in the whole. There is no increase in the range of the low velocity zone because of the distance from the plane of the stirring shaft. On the contrary, compared with the Z=0.126 m plane, the flow field of the three region is relatively mild, where the

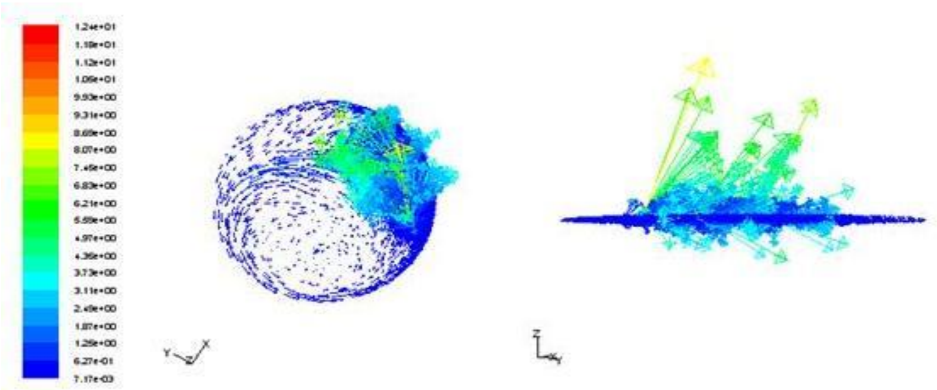


Figure 5. Velocity vector diagram of flow field at Z=0.126m

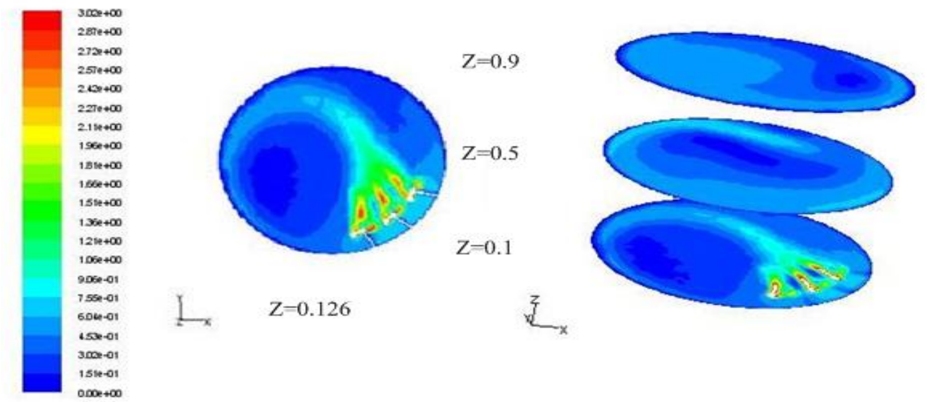


Figure 6. The velocity distribution on the four plane

velocity is large in the range of 0.30.6m/s, even which is smaller in a low velocity zone (deep blue), it is more conducive to the mix. It should be noted that a vortex will be formed, which determines that the low velocity region is not at the opposite tank wall but near the central area, as shown in Figure 7. This is very unfavorable for the gasoline blending, and the flow structure determines the center of the low-speed area relatively stable and not easy to eliminate, which will certainly affect the mixing time and be also a disadvantage of mechanical mixing.

For the three vortices on Z=0.1 m, Z=0.5 m and Z=0.9 m planes, it can be seen from Figure 7 that the central regions of the three vortices are low velocity zone. The velocity is scattered to the surrounding area and different in each direction (which is caused by the rotation of the vortex). Additionally, there is a speed gradient in the same direction. The region with the largest velocity in the whole plane is located in the Z=0.1 m plane, and the velocity in the Z=0.9 m plane is the smallest one.

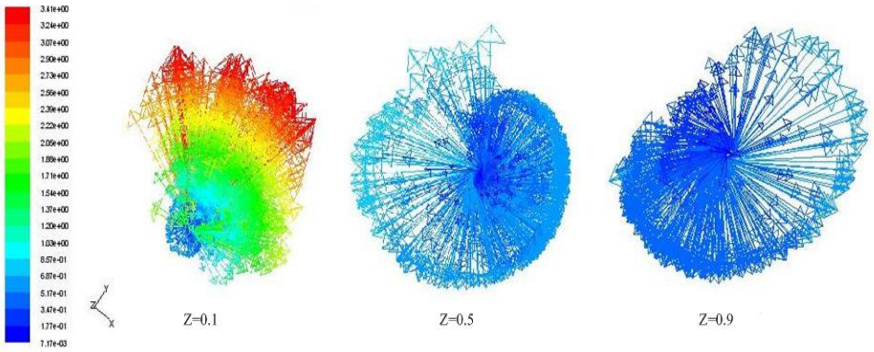


Figure 7. The velocity vector diagram on the three planes

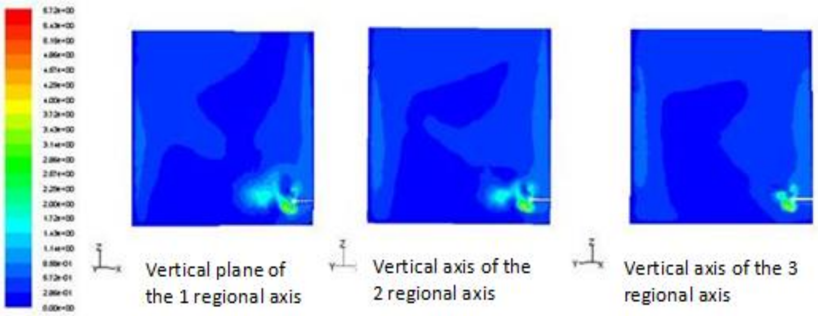


Figure 8. Velocity distribution in the vertical axis of three moving region

From Figure 8, the velocity field formed by three uniform impellers on the vertical plane of the axis is very similar and it is again verified that the vicinity of the central region of the transfer tank is a low velocity region. At the same time, the bottom of the tank is also a blind area for the mixing of the stirring blades. It is suggested that there should be a downward inclination angle between the stirring axis and the horizontal surface in the actual engineering installation, further optimizing the flow field in the lower and middle areas of the mixing tank [24].

The density values of three planes of Z=0.1 m, Z=0.5 m and Z=0.9 m obtained by Fluent are imported into Origin, and the density mixing time curve is obtained in Figure 9. The density of the three planes fluctuates violently between 10 s and 20 s after the stirring start, which is the result of axial flow in the flow field of the bleeding tank caused by radial flow. The upper layer of low density is brought into the lower region of the tank so that the density at the plane Z=0.1 m and Z=0.5 m decreases, corresponding to the upper zone density. With the continuous action of the stirring blade, the density gradually reaches to the equilibrium density of 755.9kg/m³ .On the Z=0.1 m plane, the mixed gasoline finally reaches the equilibrium density requirement, which is related to the effect of the low velocity zone near the stirring blade, resulting in a mixing time of 83.5 s.

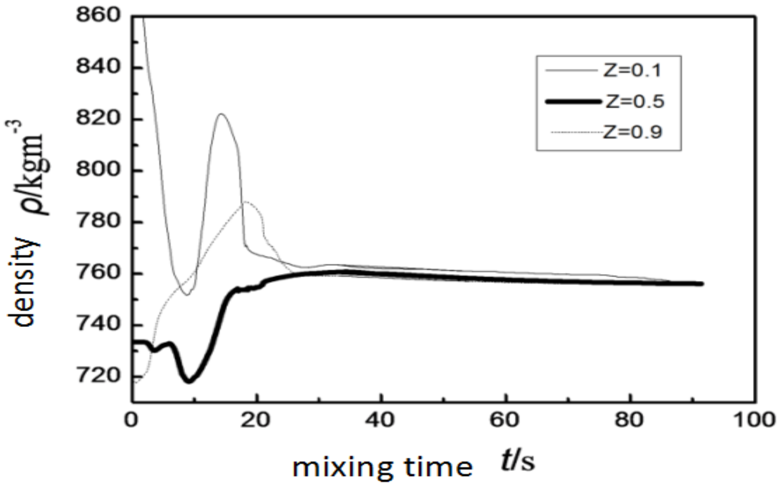


Figure 9. The mixed gasoline density profile and mixing time

3. Conclusion

In this paper, through the numerical simulation of the internal flow field of the side entering agitator system, the following conclusions are drawn:

(1) With the increase of stirring time, the flow velocity gradually increases, the radial flow affects the multi-phase mixing in axial and vertical direction, breaking the initial phase separation and then achieving gasoline blending. The velocity distribution of each section of the Z axis is similar, in the far zone, the stirring is relatively soft as the velocity range is 0.3~0.6 m/s. The range of the low velocity region is smaller, which is favorable for the mixing.

(2) The axial flow of the stirrer is very significant. The high velocity region of the mixed flow is located in the area near Z=0.126 m plane, where the speed can reach 10.02 m/s and then decay to 0.35 m/s at the wall to meet the speed requirements of gasoline blending.

(3)The bottom of the tank is a blind area for the mixing process. It is suggested that there should be a downward inclination angle between the stirring axis and the horizontal plane when the actual engineering installation is carried out. With the increase of stirring time, the mixing density gradually reach to the equilibrium density of 755.9kg /m^3, and the final mixing time is 83.5s.

Acknowledgement

This project is supported by Science and Technology Development Planning of Shandong Province (2014GGX108001), Key research and development program of

Shandong Province (2016GGX104018).

References

[1] Yu, Z. B. (2006) Comparison of Tank Mixing Ways in Petroleum Chemical Corporation. *Practical Scientific* **4**, 80-83.

[2] Cai, X. J. and Hu, L. Z. (2001) Research on the Structure of Side Inserted Agitator. *Practical Chemical Equipment* **30**, 45-46.

[3] Fang, J. and Sang, Z. F. (2009) Numerical Simulation of 3-D Flow Field in Side-entering Agitator. *China Petroleum Machinery* **37**, 30-34.

[4] Ma, Q.S. and Nie, Y.Q. (2003) Numerical Simulation of 3-D Flow Field in Side-entering Agitator. *Journal of Chemical Industry and Engineering* **54**, 612-618.

[5] Zhou, G. Z., Wang, Y. C. (2003) CFD Study of Mixing Process in Stirred Tank. *Journal of Chemical Industry and Engineering* **54**, 886-890.

[6] Guo, W. H. and Pan, J. Z. (2009) Application of Computational Fluid Dynamics to Flow Research and Structure Design of Stirred Tank. *Chemical Engineering* **37**, 20-23.

[7] Asghar, A. D. and Masoud, R. (2004) CFD Simulation of Homogenization in Large-scale Crude Gasoline Storage Tanks. *Journal of Petroleum Science and Engineering* **43**, 151-161.

[8] Janz, E. E., Fasano, J. and Myers, K. J. (2005) Different Solids Suspension Techniques in Flue Gas Desulfurization. *American Institute Of Chemical Engineers*, 4067-4068.

[9] Saeed, S., Ein MozaffariF. and Upreti, S. R. (2007) Using Computational Fluid Dynamics Modeling and Ultrasonic Doppler Velocimetry to Study-pulp Suspension Mixing. *Industrial and Engineering Chemistry Research* **46**, 2172-2179.

[10] GOMEZ, C., BENNINGTONC, P. J. and TAGHIPOUR, F. (2010) Investigation of the Flow Field in a Rectangular Vessel Equipped With a Side-entering Agitator. *Journal of Fluids Engineering* **132**, 1-13.

[11] Zhang, L. J. and Chen, G. G. (2013) Numerical Simulation of the Flow Field Characteristics of the Multi Phase Flow in a Stirred Tank. *Chinese Journal of Environmental Engineering* **4**, 1594-1600.

[12] Liang, J.Y. and Zhou, Y. J. (2015) Study on Solid Liquid Flow in Double Side Inlet Stirred Tank. *Light Industry Machinery* **33**, 35-38.

[13] Song, J. L. (2015) Numerical Simulation of Solid Liquid Two Phase Flow in a Fermentation Tank. *Energy Conservation* **5**, 22-25.

[14] Liu, G. Y. (2010) Numerical Simulation of Solid-liquid Suspension in Side Mixing. *Master's Degree Thesis of Shandong University*, 27-40.

[15] Wang, K. and Yu, J. (2003) Mixing equipment. *Chemical Industry Press*, 12-16.

[16] GB/T 4756-1998, (1998) Petroleum liquids-Manual sampling. *National standard press*, Beijing

[17] Xu, W. X. (2011) Analysis of the internal flow in a stirred tank bioreactor. *Hubei Agricultural Sciences* **13**, 2743-2745.

[18] Li, M. H. and Li, H. F. (2014) Study on Influencing Factors of Flow Field in Mixing Tank Bottom mixer. *Chemical Machinery* **6**:794-798.

[19] Cui, N. (2009) Simulation of Flow Field in Large Side Stirred Tank. *Shanghai Chemical Society* **2**.

[20] Chen, G. G., Zhang, L. J., Bo, Y. and Ye, X. C. (2012)Numerical Simulation of Influence of Blade Parameters on Flow Field and Power in Side-inlet Stirred Tank. *Journal of Beijing University of Chemical Technology* **3**, 29-34.

[21] Hou, H. G. (2010) Computational fluid dynamics simulation of gas-liquid flow in large stirred

fermentation tank. *Tianjin University*, China.

[22] Bi, J. W., Zhu, H. G., Shi, H. X., Li, Y. M., Rong, L. and Wang, T. (2010) CFD simulation and temperature field verification of mixing tank digester. *Journal of Agricultural Engineering* **10**, 283-289.

[23] Jiang, Y., Deng, L. K., Xu, G. H. and Lin, H. L. (2012) Simulation Study on Flow Field Characteristics of Side Stirred Fermentation Tank. *China Plant Engineering* **7**, 45-47.

[24] Ren, J. (2007) Simulation and Experimental Study on Flow Field and Dynamic Performance of Stirring Reactor. *Zhengzhou University, China*.