Numerical simulation of gasoline blending based on RJM system

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

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

*Presenting author: 1933651144@qq.com
†Corresponding author: chensy66@sdu.edu.cn

Abstract

In order to improve the mixing efficiency and reduce the mixing time, a plurality of nozzles with different installation angles should be selected in the process of gasoline blending. Based on the gasoline physical parameters requirements, nozzle jet model and five kinds of gasoline blending schemes are built to simulate the mixing flow field. Combining the standard $k-\varepsilon$ turbulence model and the sliding grid technique, the numerical simulation and process optimization is done with the commercial Fluent software. The attenuation curves of the nozzle axis radial velocity at the landscape orientation, of the dynamic pressure at the transverse jet center, the velocity distribution curve of the interface and the velocity contours of every mixing flow field are studied. The injection performance of the nozzles and mixed characteristics of the RJM system is finally worked out to get the density- mixing time rules.

Keywords: Gasoline blending; Numerical simulation; Swirling jet

0 Preface

With the development of science and technology, people pay more and more attention to energy conservation and environmental protection, which improves the various industries' quality requirements for petroleum products. Because of the limitation of the processing technology, many mono-component products can't meet the needs, so two or multi component products are mixed and stirred in different proportions. These different components can give full play to their excellent performance in order to meet the product quality requirements of the consumers. Therefore, oil blending is a necessary part of the oil production process [1]-[2].

The research work of jet mixing began in the 50s of last century. In the past 20 years, with the development of CFD technology, the research of jet mixing system has been further developed [3]-[7]. In 2002, A. W. Patwarhan took the jet velocity, the nozzle angle and the geometric dimensions as variables to predict the mixing time and the concentration distribution of jet mixing system, the total mixing time of simulation is consistent with the actual test result [8]. In 2004, Zughbi and Rakib studied the effects of jet angle and jet number on mixing time [9]. In 2006, Sun Wei simulated the core components in the rotary jet mixing system- the axial flow turbine, which was used to clean up the industrial oil sludge, and the author proved the influence factors and control methods of the power turbine's hydraulic performance [10]. In 2007, Rahimi and Parvareh [11] studied the mixing process of the crude oil storage tanks, which simultaneously installed the nozzles and the impellers inside, the conclusion was that the angle between the nozzle and the impeller had a significant effect on the mixing time. In 2008, Tian Yanli and others numerically simulated the three-dimensional flow field of the injection flow in the oil tank, they preliminarily studied the influence rule and the optimal combination of the nozzle structure, fluid properties, inlet conditions and the other factors [12]-[13]. In 2009, Parvareh and Rahimi et al proved that the nozzle position had a significant effect on the neutralization reaction rate in the mixing tank by means of experiments and numerical simulation [14]. Since 2010, Chen Songying [15]-[17] et al used the CFD software to simulate the flow field change of gasoline component, whose initial state was five-layer distributions under the action of rotary jet system, the simulation was established based on the three-dimensional gasoline components Mixture multiphase flow model, they investigated the change rule of density- mixing time under the different system speeds and nozzle angles. In recent years, more and more people work on the numerical simulation of rotary jet mixing system [18-21], which has a positive reference on the actual production.

Rotary jet mixing system (RJM) is a new attempt and application in the field of oil blending. The author designed the numerical calculation model of RJM system, simulated the flow field mixing of different nozzle angles and rotational speeds, got the flow field velocity contours, the axial dynamic pressure attenuation curve and the axial velocity and the range distribution curve. The author also obtained the density- mixing time curve of the specified cross-section and so on, got the change rules of dynamic characteristics, compared and analyzed the different mixing effect, conducted a preliminary analysis of optimization and comparison.

1 RJM system model

In order to research the dynamic characteristics of the gasoline blending, without considering the volatile oil and the electrostatic accumulation, the research was taken from the mixed flow field, found a jet nozzle model (Figure 1) to reflect some characteristics of the flow field in the actual operation system. Considering the important role of the jet shear force and the entrainment in the mixed flow, ignoring the influence of the pipeline in the system, the model is a simple open-loop system, with the import and the export boundary conditions. The flow field in the tank was the outflow field of RJM system. Because of the outlet boundary conditions of this model, that needed to increase the height of the tank model in order to avoid the effect to the phase proportion of the final mixture, ignored the free surface fluctuation effect on mixed flow at the top of gasoline blending tank [12].

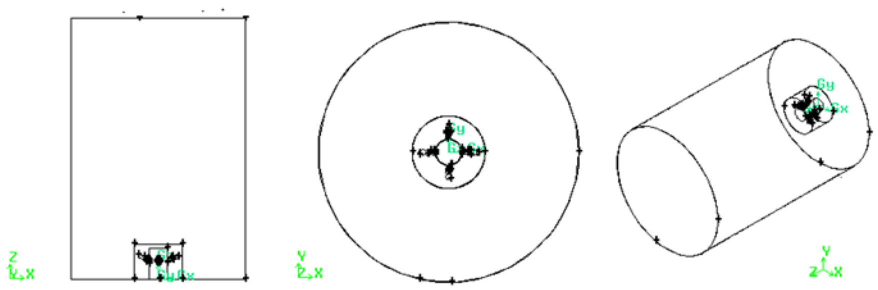


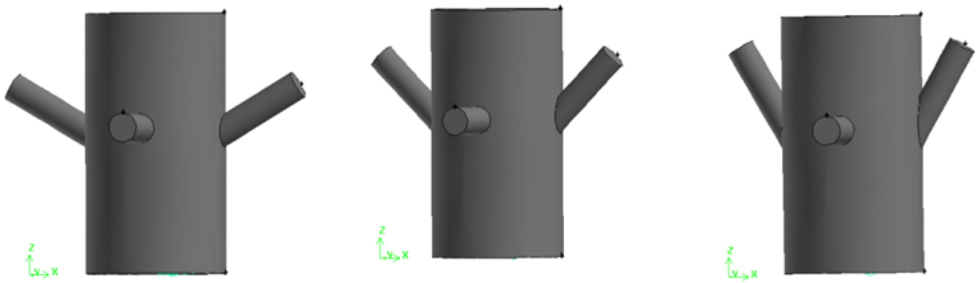
Figure 1. Nozzle jet model

2 Numerical simulation and optimization of mixed flow field

In order to improve the mixing efficiency and reduce the mixing time, it must have highly requirement of the flow field of the whole tank in the process of the gasoline blending. The author established five different kinds combination schemes $\emptyset 1\text{m} \times 1.5\text{m}$ of gasoline blending to do numerical simulation analysis and optimization of the flow field based on the blending tank and the real physical parameters of the gasoline ingredients.

2.1 RJM entity model

The inlet and outlet boundary conditions existed in the model of open loop system. In order to avoid the outlet boundary to destroy the various components of the gasoline, the height of the cylinder model should be increased properly, and an additional component was added as the main phase of the outlet boundary.



(a) $30°$ Angle of RJM system (b) $45°$ Angle of RJM system (c) $60°$ Angle of RJM system

Figure 2. Three RJM system appearances with different angles

In order to ensure a nozzle group arrangement with $180°$ distribution under the premise of constant horizontal position, five schemes are used in the different RJM models, the oblique installation angle of nozzles selected from $30°$, $45°$ and $60°$ (Figure 2) and the system speed selected from 0.1rad/s, 0.5 rad/s and 1 rad/s. In this paper, the author used the nozzle name code to distinguish the different schemes. For example, $0.1-45°$ nozzle is the model with a group of nozzles arranging at $180°$ in horizontal position, and the other group of oblique nozzles has $45°$ dip angle comparing with the horizontal plane (XY plane), and the speed of the main system is 0.1 rad/s. Following Table 1, the jet velocity at the exit of the five nozzles is the absolute velocity, and the velocity direction is perpendicular to the nozzle exit plane.

Table 1. Parameters of the five different schemes

Nozzle name	Jet velocity (m/s)	X direction angle (°)	Y direction angle (°)	RJM rotation rate (rad/s)
$0.1-45°$	3	0	45	0.1
$0.1-30°$	3	0	30	0.1
$0.1-60°$	3	0	60	0.1
$0.5-45°$	3	0	45	0.5
$1-45°$	3	0	45	1

2.2 Gridding and boundary conditions

Because the RJM system has a certain rotation speed, the magnitude and direction of the speed is constant, the author decided to use the sliding grid technique. The outer flow field of the whole system is divided into two parts, one is the moving region including the RJM system, and the other is the static region. The two regions are connected by the outer surface of the cylinder, and the type is defined as the Interface. The establishment of the interface was finished in the Fluent, so it can ensure the mutual transfer and exchange of each parameter of the flow field. The wall of the RJM system is set to the Moving Wall, and its rotational speed is 0 rad/s related to the moving region.

The size of the RJM system is different from the tank. Dividing the grid in block is to ensure the grid's quality. Using the Tet/Hybrid three-dimensional grid unit is to make full use of the established static and dynamic region. The grid sizes of two regions are set respectively to 0.0125mm and 0.015mm, which are divided totally into 580484 grids.

In the actual oil refining industry, a brand of gasoline was composed of the following four components: reformulated gasoline, MTBE, catalytic reforming gasoline and gasoline alkylate, plus the material of export spillover previously mentioned, so, this model was composed of five parts, as shown in Table 2.

Table 2. Physical parameters and distribution area of the blended gasoline components

Name	Density (kg/m^3)	Viscosity ($Pa \cdot s$)	Distribution by Z (m)
Reformulated 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
Gasoline alkylate	699.3	0.00034	0.9-1.0

The working pressure was set on the top of the blending tank at Z=1.5m plane, and the pressure value was the standard atmospheric pressure 101325Pa. Opened the gravity option, the size of the gravity was $9.8 m/s^2$, and the direction was the negative direction of the Z axis.

It should be noted that, in the initial state, all five components were assumed to be filled the whole tank in the form of stratification. Under the effect of gravity, the first phase with the maximum density (reforming gasoline) was located at the bottom of the tank, and the fifth material was distributed to the top. The Reynolds number at the nozzle exit (full of the first phase) was:

$$\mathrm{Re} = \frac{\rho v d_0}{\mu} = 6.14 \times 10^4 \tag{1}$$

The Reynolds number of the other regions with larger density was larger, so the mixed flow field was in a turbulent state. This model used the standard k-ε model, the boundary conditions can be summarized as: the four nozzles were set to VELOCITY INLET, the absolute speed was 3m/s, the direction was perpendicular to the nozzle section and pointed to the external flow field.

2.3 Numerical simulation results

Because the five models had the same arrangements with the transverse nozzles, the nozzle structures and exit velocities were also the same, they had the same axial velocity attenuation law and section velocity distribution law. Taking the $0.1-45°$ nozzle as an example, the injection performance of the nozzles was analyzed from the radial velocity attenuation curve of the nozzle axis and the center pressure attenuation curve of the nozzle jet.

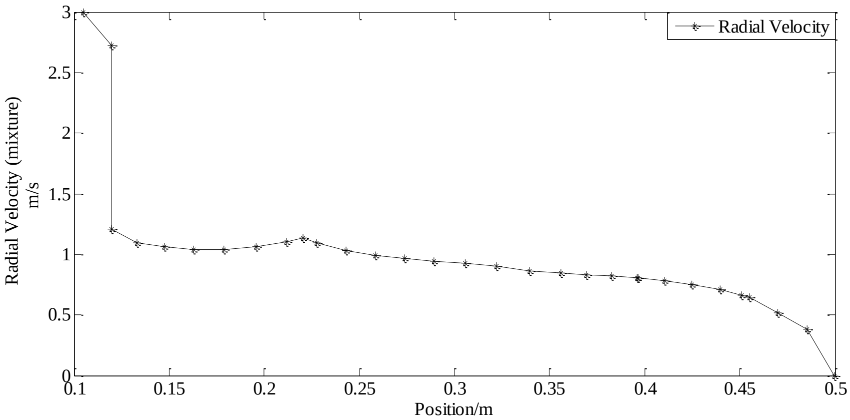


Figure 3. Radial velocity attenuation curve of the $0.1-45°$ nozzle axis

Figure 3 shows the radial velocity attenuation curve of the transverse nozzle of the system, which eliminates the influence of the axial rotation of the system. It can be seen that the horizontal nozzle in the most of flow field can maintain a moderate speed, about 1m/s, which is very important to ensure the uniformity of oil blending. In the vicinity of the wall, Y=0.5m, the axial velocity decay to about 0.35m/s, which is the lowest rate to ensure the operation of oil blending. Noted that the nozzle axis radial velocity in the vicinity of X=0.105m, the speed decays quickly, it is because that the position is the interface of dynamic and static area during the numerical simulation time, which had a greater resistance to the interface, on the other hand, the grid was in constant slip state, that affected the normal transmission of velocity field.

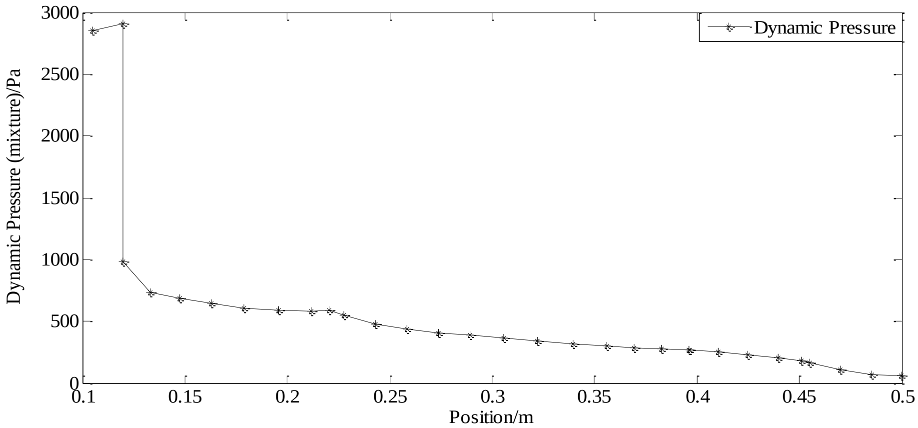


Figure 4. Center dynamic pressure attenuation curve of the $0.1-45°$ nozzle

Figure 4 shows the center dynamic pressure attenuation curve of the nozzle in this system. The jet dynamic pressure reached the maximum value at the nozzle exit, the jet dynamic pressure and total pressure were reduced after the mixture released from the nozzle, then it decayed to a small level near the wall after remaining a certain distance. Therefore, the shock force of the tank wall was small, which helped to reduce the accumulation of static electricity in the mixed flow field. The sharp drop of dynamic pressure at X=0.105m was also caused by the interface, the reason was similar to the speed decay.

The diffusion effect of the nozzle was studied by analyzing the distribution of the cross section velocity perpendicular to the axial direction of the nozzle jet.

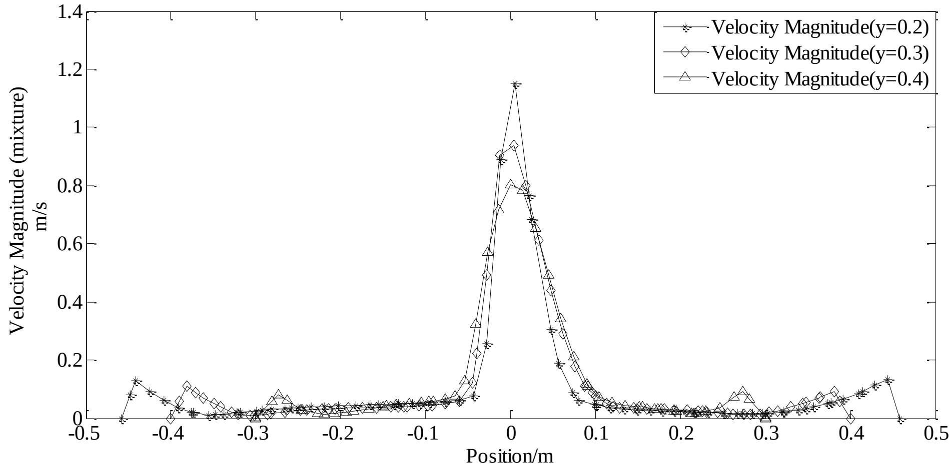


Figure 5. Velocity profile of the three ranges on the Z=0.1 plane

Figure 5 shows the mixing of the jet-flow to the same plane (Z=0.1). The velocity profile of the three different ranges of Y=0.2m, Y=0.3m and Y=0.4m plane can be seen, the maximum speed of the axis decreases with the increase of range, namely 1.15m/s, 0.94m/s and 0.8m/s. This is consistent with the axial velocity decay law of the nozzle as shown in Figure 3. The velocity distribution of the section has a self - mode, that is, the maximum velocity on the axis, the smaller the farther away from the axis, and the symmetrical distribution on the axis, the cross section velocity on different ranges is similar. The velocity in the decay process, appeared a couple symmetrical angle at the high speed on the both ends, this was due to the influence of the axial velocity of the flow field, and the velocity was mainly caused by the deflection of the flow direction at the end of the jet-flow.

(1) Simulation results of different nozzle angles at the same rotating speed

In this model, the numerical simulations of the mixing characteristics of the RJM system with three kinds of oblique nozzle inclination angle of $30°$, $45°$ and $60°$ were carried out. The results of simulation and analysis were as follows.

Figure 6, Figure 7, Figure 8 clearly show the core area and the main body of the jet-flow, as well as the velocity gradient in the process of oblique nozzle jet attenuation. For the RJM system with the $0.1-30°$ nozzle, the velocity was about 0.3m/s - 0.35m/s in the axial direction of the nozzle when the exit velocity was 3m/s, the velocity met the requirements of petroleum blending. In addition, the variations of the RJM system rotates counterclockwise along the Z axis and the jet-flow region under the effect of gravity can also be seen. For different nozzle inclination angles, the main affected areas of the jet-flow were also different. For the $0.1-30°$ nozzle, stir at the bottom of the tank was violent, conducive to a high density components upward movement to full mixture. For the $0.1-60°$ nozzle RJM system, the range of the oblique nozzle was far, focusing on the mixing in the upper and middle blending tank, relatively easy to cause the volatilization of the gasoline, while it was not conducive to the rapid mixing of the larger density phase at the bottom of the tank.

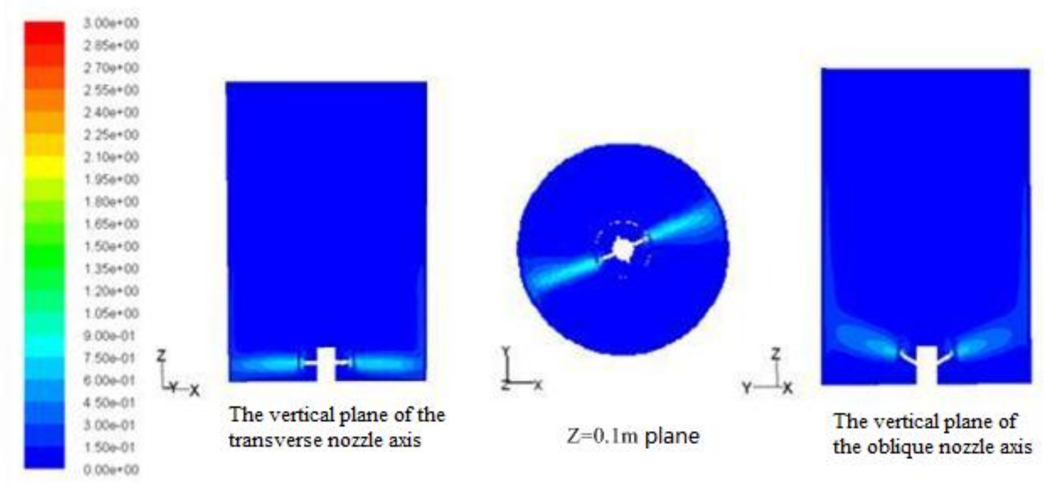


Figure 6. The velocity nephogram in the three plane mixed phase of the $0.1-30°$ nozzle RJM system at 460s

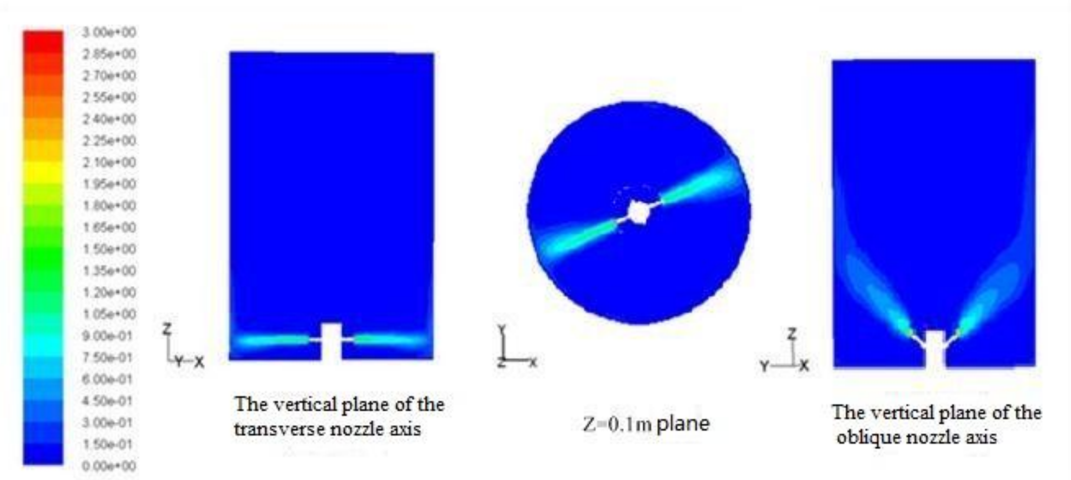


Figure 7. The velocity nephogram in the three plane mixed phase of the $0.1-45°$ nozzle RJM system at 460s

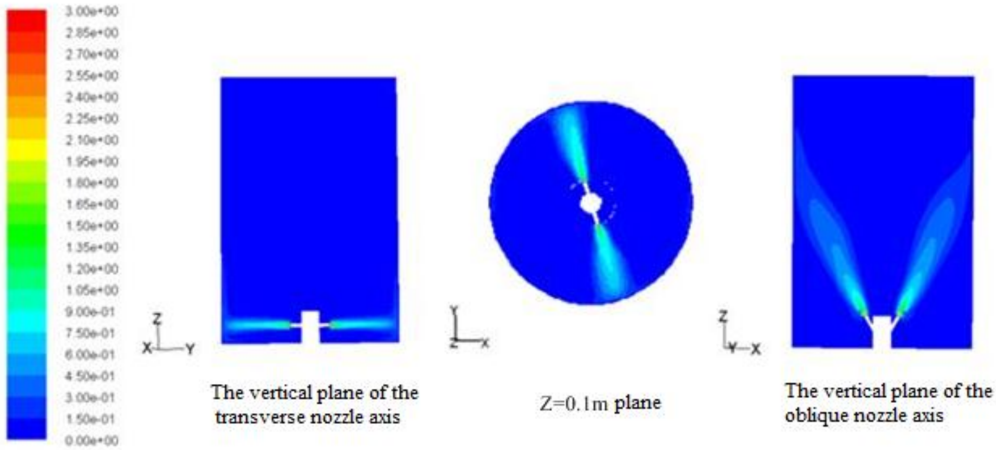


Figure 8. The velocity nephogram in the three plane mixed phase of the $0.1-60°$ nozzle RJM system at 460s

The axial flow in the mixed flow field was necessary to make the initial state of the multiphase flow in a stratified arrangement, which was mainly dependent on the jet flow of the oblique nozzle in the RJM system. In the Y=0 plane, the velocity distributions in the range X=0.2m, X=0.3m and X=0.4m were taken to investigate the jet influence of the mixing of the upper and lower fluid to the tank, as shown in Figures 9, 10 and 11. Figures 9, 10 and 11 illustrated the velocity distribution of an oblique nozzle in a straight line in the Z direction, the velocity profile of the nozzle was similar to that of the transverse nozzle, and there were also high speed and low speed zones. But the difference was that, due to the influence of oblique nozzle angle, for different values of X, high speed area appearing in the Z direction was not consistent, and with the increase of X value, high speed zone shifted to Z positive axis, maximum velocity also showed the attenuation trend. For example, for the $0.1-45°$ nozzle RJM system, the maximum speed of the X=0.2m section was in the vicinity of Z=0.25m, the maximum value was 0.7m/s, while the maximum speed of the X=0.4m section was in the vicinity of Z=0.45m, the maximum was attenuated to 0.46m/s.

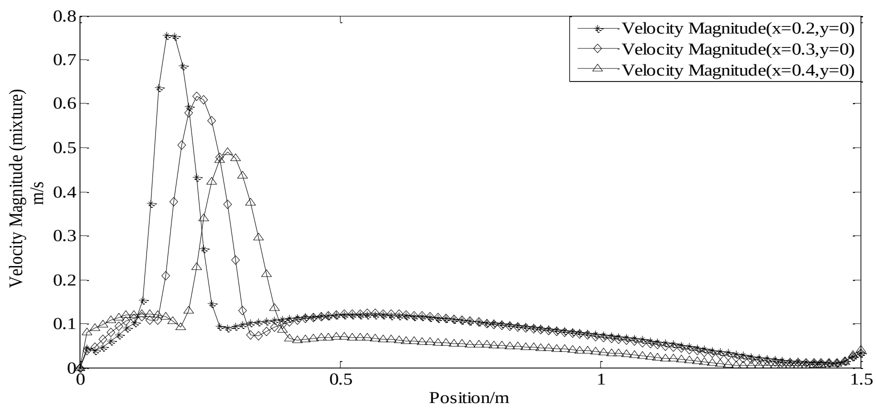


Figure 9. The velocity profile of the three range of the $0.1-30°$ nozzle RJM system

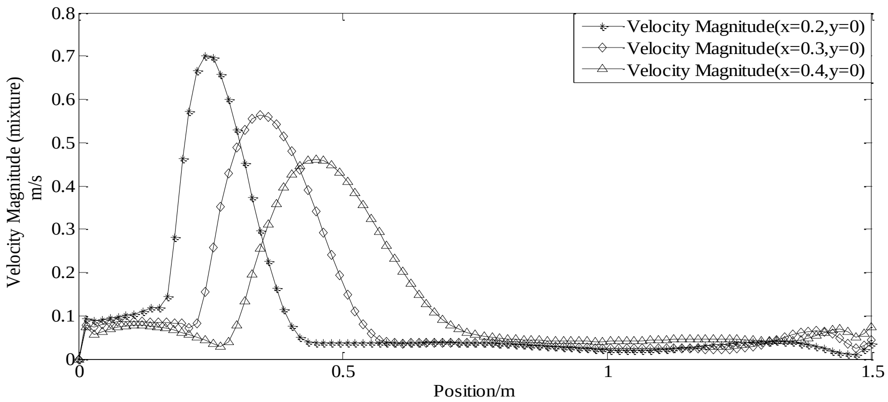


Figure 10. The velocity profile of the three range of the $0.1-45°$ nozzle RJM system

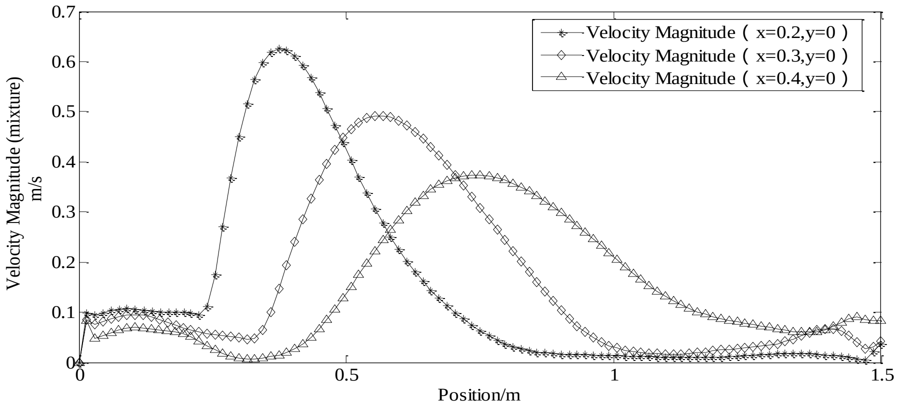


Figure 11. The velocity profile of the three range of the $0.1-60°$ nozzle RJM system

It was different from the transverse velocity distribution of the transverse nozzle (Figure 5), the above three diagrams also showed that the velocity distribution on the cross section was not symmetrical under gravity. In the flow field at the bottom of the tank, the velocity could be kept constant by the influence of the circumferential flow formed by the transverse nozzle. So in the [0, 0.2] region of the abscissa, a constant speed was independent of the range X. Similarly, affected by the outlet pressure, the upper oil tank area would develop a temporary speed recovery area, which was the direct result of the abnormal velocity distribution in Figure 11 at the X=0.4 cross section.

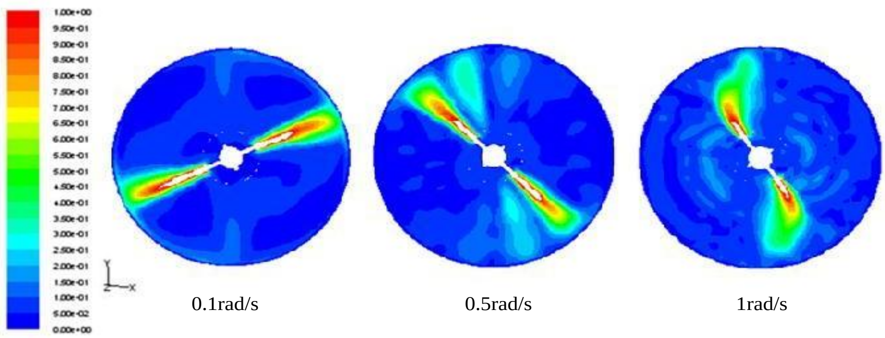


Figure 12. Velocity distribution of Z=0.1 section with different rotating speed of the nozzle RJM system

(2) Simulation results of the same nozzle inclination at different speeds

In order to explore the influence of the rotation speed of RJM system around the Z axis on the mixing effect, the mixing characteristics of RJM system with three different speeds (0.1rad/s, 0.5rad/s and 1rad/s) were simulated. The most direct effect on the mixing flow field was the change of circumferential velocity in the XY plane. As shown in Figure 12, the 0.1rad/s speed of the system will appear in the vicinity of the nozzle 0.1m/s - 0.15m/s low velocity correlation, but the overall impact on the flow field is very small, it can be regarded as static. As shown in Figure 12, the 0.1m/s - 0.15m/s low velocity correlation would appear in the vicinity of the transverse nozzle under the 0.1rad/s speed of the system, but the overall impact on the flow field was very small, that can be regarded as static. The system speed of the 0.5rad/s was obviously wider than that of the 0.1rad/s, and the range of the high speed area was larger, and it was obvious that the jet traces left by the transverse nozzle at the last moment. In the Z=0.1 plane, the mixed flow field of 1rad/s system was more variable, the relatively large rotation speed caused a circumferential velocity, the velocity was larger and larger along the radial direction, which was proportional to the distance from the point to the center of the circle. Finally, the direction of the jet was deflected along the opposite direction of rotation, and the terminal velocity of the whole flow field was up to 0.3m/s, that was beneficial to the mixing of multiphase fluids.

The above results do not show that with the increase of the rotating speed of the system, the mixing time will be shorter. This is because a mixture of multiphase flow mainly depends on the circumferential velocity of jet, the increasing rotation speed will significantly increase the circumferential velocity of flow, and that may format the vortex in the XY plane, which has little effect on mixing multiphase flow, sometimes plays an opposite role. On the other hand, the large circumferential velocity has a great influence on the range of the transverse nozzle, so, it is easy to form a dead zone in the tank wall, which affects the final mixing time.

3 Conclusion

This article based on the reduced gasoline tank model without considering the nozzle structure, ignoring the premise of volatile oil and electrostatic accumulation. Mixing model was established from the jet mixing angle, and the author studied on the gasoline blending dynamics by using the Fluent software, obtained the following conclusions:

(1) The transverse nozzle can maintain a moderate speed about 1m/s in most regions of the flow field, which was very important for ensuring the uniformity of oil blending. In the vicinity of the wall, the axial velocity was reduced to about 0.35m/s, which ensures the smooth operation of the lowest rate of oil blending. It can be seen that the RJM jet mixer can reduce the volatile gasoline at the same time it can ensure the uniform mixing of the most flow field.

(2) The jet dynamic pressure reached maximum at the nozzle outlet. When the mixed fluid left the nozzle, the jet dynamic pressure and total pressure were decreased, after maintaining a distance, finally were attenuated to a very small level near the wall. Therefore, the shock force of the tank wall was small, which helped to reduce the accumulation of static electricity in the mixed flow field.

(3) For the 30° nozzle RJM system, stir on the bottom of the tank was violent, which was conducive to the phase with larger density moving upwards to fully mix. For the 60° nozzle RJM system, the oblique nozzle had a long range, it focused on the upper mixed in the blending tank, which may easily cause the gasoline volatilization, while not conducive to rapidly mix of the high density phase at the bottom of the tank.

Acknowledgement

This project is supported by Science and Technology Development Planning of Shandong Province (2014GGX108001), Key research and development program of Shandong Province (2016GGX104018).

Reference

[1] Li, Z. X. and Xu, S. W. (1997) *Oil Storage and Transportation Design Manual*, Petroleum Industry Press, Beijing, China.

[2] Chang, S. R. and Liang, W. (1999) Progress and prospect of gasoline, *Natural Gas and Petroleum* **04**, 19-22.

[3] Ranade, J. J. (1996) Towards better mixing protocols by designing spatially periodic flows: the case of a jet mixer, *Chemical Engineering Science* **51**, 2637-2642.

[4] Masoud, R. and Arsalan, P. (2005) Experimental and CFD investigation on mixing in a semi-industrial stirred tank, *Chemical Engineering Journal* **115**, 82-92.

[5] Wang, X.P. (2008) Research on the jet mixing system used for oil tank anti-sedimentation, Shandong University, Jinan, China.

[6] Liu, L. Y., Meng, F. L., Wang, W. Q. and Song, B. (2015) Application and development of agitator in crude oil storage tank, *Contemporary Chemical Industry* **02**, 386-388+396.

[7] Wang, J. L. (2007) Numerical simulation and hydraulic characteristics of internal flow in hydraulic driven rotary jet mixer, Zhejiang University, Hangzhou, China.

[8] Patwardhan, A. W. (2002) CFD modeling of jet mixed tanks, *Chemical Engineering Science* **57**, 1307-1318.

[9] Zughbi, H. D. and Rakib. M. A. (2004) Mixing in a fluid jet agitated tank: effects of jet angel and elevation and number of jets, *Chemical Engineering Science* **59**, 829-842.

[10] Sun, W. (2006) The hydraulic performance study and structure optimization design of power turbine in industrial sludge jet mixing system, Zhejiang University, Hangzhou, China.

[11] Rahimi, M. and Parvareh, A. (2007) CFD study on mixing by coupled jet-impeller mixers in a large crude oil storage tank, *Computers and Chemical Engineering* **31**, 737-744.

[12] Tian, Y. L., Hang, L. Q. and Pan, D. J. (2008) Full numerical simulation analysis of rotary jet mixer, *Electromechanical Engineering* **25**, 86-90.

[13] Wang, L. Q., Lin, S. D. and Tian, Y. L. (2008) Study on jet performance of large flow nozzle based on CFD, *Fluid Machinery* **11**, 17-22.

[14] Parvareh, A., Rahimi, M. and Alsairafi, A. A. (2009) Experimental and CFD studies on the effect of the jet position on mixing performance, *Iranian Journal of Chemical Engineering* **6**, 3-10.

[15] Chen, S. Y., Xie, F. C. and Qu, Y. P. (2010) Research on the jet mixing system used for oil blending, *Journal of Engineering Thermo-physics* **31**, 229-232.

[16] Xie, F. C. (2011) Study on the dynamics peculiarity of gasoline blending in the process of rotary jet mixing, Shandong University, Jinan, China.

[17] Liu, C. W. (2015) Research on the jet stirring dynamics of lime slurry pond of wet flue gas desulfurization system, Shandong University, Jinan, China.

[18] Wang, M. R., Zhang, Y. F. and Cai, Y. B. (2014) Numerical simulation of flow field of omni-directional rotate jet mixer, *Chemical Industry and Engineering Progress* **S1**, 79-84.

[19] Yu, B. B., Yang, J. Y. and Liu, Q. (2015) Numerical simulation on rotary jet mixer of oil tank at different jet velocity, *Contemporary Chemical Industry* **06**, 1412-1414+1418.

[20] Fan, D. W. (2016) Numerical simulation on rotary jet mixer of large storage tank, *Oil-gas field Surface Engineering* **09**, 4-5+9.

[21] Hu, J. G. (2017) Study on CFD for coal powder slot anti-settling jet mixing system, *Shanghai Chemical Industry* **01**, 16-20.