Numerical simulation and optimization of cleaning bidirectional nozzle of large petroleum storage tank

J Lyu¹, J J Ye², Z G Lou¹ and J R Xu¹

¹School of Naval Architecture and Mechanical-electrical Engineering, Zhejiang Ocean University, Zhoushan 316022, China
²School of Petrochemical and Energetic Engineering, Zhejiang Ocean University, Zhoushan 316022, China
E-mail: lvjunn@163.com

Abstract. In the process of storing petroleum for a long time, the coagulability substance in the petroleum will form high viscosity petroleum sludge at the bottom of the tank, resulting in the loss of petroleum, the decrease of reserves, and the internal corrosion. The main components of petroleum sludge are petroleum and wax, which have certain recycling value. In this paper, the numerical simulation technology is used to simulate and optimize the flow field of the bidirectional nozzle in the large petroleum tank cleaning device. The shape of the nozzle with fast velocity, steady jet-flow and high efficiency, solves the problem of deposit and cleaning of petroleum sludge in petroleum tank, and provides a new idea for the future design of the mechanical cleaning device for the petroleum tank.

1. Introduction
During the storage process of crude petroleum, the inorganic impurities such as high melting point wax, asphaltene, colloid and sands of the entrainment will precipitate with water and form the petroleum mud at the bottom of the tank. If the mud is accumulated, the mud will cause the loss of the petroleum, the decrease of the reserves of the tank, the water and the salts in the mud will cause the continuous corrosion of the tank[1-3]. With the large-scale construction of petroleum tanks, it will have adverse effects on its long-term safety production. The petroleum content in the general storage tank is about 1%-2.2%[3] of the volume of the whole storage tank, and the period of the tank cleaning is generally 3-5 years, and the petroleum storage in China has reached $6 \times 10^7$ t at the end of the fifteen, and the petroleum sludge at the bottom of the tank is as high as $1-4.4 \times 10^4$ t[4] every year. At present, the petroleum sludge is mostly stored in the form of simple stacking or landfill, resulting in the waste of resources, and at the same time, it also has serious harm to the petrochemical field environment and human health. Therefore, it’s of great significance to solve the problem of depositing and cleaning petroleum sludge in petroleum tank, to improve the efficiency of petroleum storage, to save the cost of operation and to improve the ecological environment in the field.

The numerical simulation technology is used to simulate and optimize the flow field simulation of the two-way nozzle in the large petroleum tank cleaning device. The bidirectional nozzle structure model of the common storage tank is set up[5], and the 4 types of model modification are established. The influence of the nozzle shape[7-9] and the number of vertical tube on the jet flow of the two-way nozzle is studied, and the optimum 4 is to be optimized. The model with fast flow rate and stable jet-flow provides the basis for the design and development of large petroleum tank cleaning device.
2. Computational methods

2.1 Governing equations
The governing equations for turbulent flow in the present study are the continuity equations for mass conservation and unsteady Reynolds-averaged Navier-Stokes equations for momentum transport, as follows:

\[ \frac{\partial u_i}{\partial x_i} = 0, i = 1, 2, 3 \]  \hspace{1cm} (1)

\[ \rho \frac{\partial u_j}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_i} = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} - \rho u_i u_j \right), i, j = 1, 2, 3 \]  \hspace{1cm} (2)

Where \( i = 1, 2, 3 \) represents X,Y,Z direction respectively. \( -\rho u_i u_j \) is the Reynolds stress tensor, \( u \) is the averaged velocity, \( \rho \) is the density, in the present study.

There are several turbulence models employed to satisfy the governing equations in this similar study. The standard \( k - \varepsilon \) model are chosen for use in this study. Transport equations of \( k \) and \( \varepsilon \) in standard \( k - \varepsilon \) model as

\[ \frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + G_t + G_b - \rho \varepsilon - Y_m \]  \hspace{1cm} (3)

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon,1} \frac{\varepsilon}{k} (G_k + G_b) - C_{\varepsilon,2} \rho - S_\varepsilon \]  \hspace{1cm} (4)

\[ G_t = \mu_t \left( \frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_j} \right) \frac{\partial u_i}{\partial x_j} \]  \hspace{1cm} (5)

\( G_k \) is the turbulent kinetic energy production caused by an average velocity gradient, \( G_b \) is the turbulent kinetic energy production caused by buoyancy, \( Y_m \) represents the effect of fluctuating expansion of turbulent on dilatation dissipation in compressible turbulent, \( C_{\varepsilon,1}, C_{\varepsilon,2} \) and \( C_{\varepsilon,3} \) are constant, \( \sigma_k \) and \( \sigma_\varepsilon \) are turbulent Prandtl number of turbulent kinetic energy \( k \) and turbulent dissipation rate \( \varepsilon \). \( S_k \) and \( S_\varepsilon \) are the user-specified source term.

The free surface is modeled by using Volume-of-Fluid (VOF) approach and a high-resolution interface capturing scheme, as (6).

\[ \frac{\partial \alpha}{\partial t} + \vec{v} \cdot \nabla \alpha = 0 \]  \hspace{1cm} (6)

2.2 Physical models and computational domain
In this study, the basic dimensions of the nozzle model refer to the common tank cleaning two-way jet nozzle parameters, and its shape and structure are shown in the figure1. And the inner region of the tube and the efflux generating area of the outlet are selected as the computational domain. The tube wall was set as the wall surface, and the inlet is set as pressure boundary with pressure is 1Mpa. The pressure boundary is also set at outside the jet area. In terms of physical conditions, light crude petroleum with a density of 875kg/m^3 was selected as the liquid phase, and incompressible air was selected as the gas phase. The computational domain and the boundary conditions is shown as figure2.
2.3 Study on grid convergence
This study choose polyhedron meshing calculation domain, near the tube wall uses the prism layer encryption. In this study, 5 grids were divided, with the number of grids ranging from 0.2 million to 0.85 million. The average outlet speed was taken as the reference standard for grid dependency analysis. Its calculation results as shown in figure 3, the results difference is small but gradually increased along with the increase in grid average outlet speed and stable after the 0.41 million grid, so choose 0.41 million grid for subsequent calculations. The 0.41 million grid is shown in the figure 4.

3. Numerical results and discussion
The shape of nozzle and the number of nozzle were optimized, and the square mouth and circular mouth was taken respectively. The number of vertical tubes is also taken into account. A total of 4 different bidirectional nozzle models have been established as shown in figure 5. All nozzles have the same discharge area and length, its basic parameters are shown in the table 1.
Table 1. Types nozzle parameters.

| Type | The nozzle shape | The number of vertical tubes | Nozzle diameter (side length) | Inlet area |
|------|-----------------|-----------------------------|-------------------------------|------------|
| P1   | circular        | single                      | 17.6cm                       | 243cm²     |
| P2   | circular        | double                      | 12.4cm                       | 243cm²     |
| P3   | square          | single                      | 15.60cm                      | 243cm²     |
| P4   | square          | double                      | 11.03cm                      | 243cm²     |

With the same inlet pressure, the average velocity, average turbulent kinetic energy and mass flow at the jet outlet are respectively monitored. The calculated results are shown in the figure 6.

![Mass Flux](image1)
![Average Turbulent Energy](image2)
![Average Velocity](image3)

(a) Mass flux  (b) Average turbulent energy  (c) Average velocity

Fig. 6. Flow field parameters of outlet surface.

In the flow field of the four types of bidirectional nozzle, the mass flux of type P2, P4 is smaller than P3. P1, P3 mass flux is around 320 kg/s. That compared with the single nozzle, the double nozzle mass flux is smaller. And P2 type nozzle flow is far less than the rest of the three models. The average turbulent kinetic energy of model P1 and model P3 is greater than that of model P2 and model P4. This is because the structure of the two vertical tubes of the single vertical tubes has a similar "three-way" structure, which changes the direction of the fluid, which will result in more swirls and vortexes. Larger turbulent kinetic energy means more vortexes, which also means better scour cleaning ability. However, if the turbulent kinetic energy is large, cavitation will occur, resulting in jet diffusion, and the range is not far. Because the fluid is in the same nozzle for a long time, the internal wall of the tube has obvious effect on the steady flow of the fluid, so the turbulent kinetic energy is small. Fortunately, the turbulent kinetic energy of all shaped nozzles is at a small level, and no cavitation occurs during jet flow. Therefore, it is necessary to select P1 and P3 nozzles with high turbulent kinetic energy at the nozzle for further optimization.

In terms of speed, the discharge velocity of the four types of nozzle is relatively similar. In ideal case, with the same pressure, the same area of tube through the fluid volume should be the same, but in the actual cases, the wall of the boundary layer effect, spiral angle will affect the performance of the nozzle, 4 different types of nozzles have different outlet velocities, and the outlet velocities of P1 and
P1 nozzles are slightly higher than those of P2 and P4. It is shows that in terms of speed, single vertical tube is better than double vertical tube, because for double vertical tube, a stream surface area is greater than the single vertical tube, under the same pressure flow surface area means that the greater the area of low or even zero velocity boundary layer increases, the higher the energy consumption, the flow rate also will be more slow.

From cross section flow field distribution, the four types of nozzle tube flow field are different, their flow field parameters of cross section graph, as shown in figure7-9. Considering the double vertical tube flow is symmetrical and mutual interference, therefore, only the right part was shown in calculation domain.

Jet speed on single vertical tube is bigger than the double vertical tube, in single vertical tube, maximum speed area is at the downstream area of angle, but in the subsequent flow speed and had significantly lower. In the single vertical tube model, the fluid velocity is smaller in the tube, and the effect is better than that of the double nozzle model after the nozzle shrinkage and acceleration.

In the single vertical tube model, high pressure is maintained in the horizontal tube, while in the double vertical tube model, there is an obvious step-down section in the horizontal tube. This also indicates that this part of the fluid speed is fast, and the nozzle contraction section is not good for the acceleration and flow resistance of the fluid. In the four models, the connection between the horizontal tube and the vertical tube is under great pressure. In the actual design of the structure of the device, structural reinforcement is required here.

The volume fraction here represents the volume fraction of the liquid. When the volume fraction is 1, the position is filled with liquid, and when the volume fraction is 0, the place is filled with gas. It can be seen from figureure P3 models side there is an obvious direction of jet nozzle and the Y axis unparallel situation, this is caused by uneven speed of liquid in the nozzle, to show that the square nozzle contraction stableility fluid effect circular nozzle is better.

4. Conclusions
In this paper, the numerical simulation of bidirectional jet nozzle used in tank cleaning is carried out. In addition, the number of vertical tubes and the shape of nozzle were changed, and 4 different models of
nozzle models were formed. It was found in the numerical simulation of jet nozzle of different types of nozzle.

1) under the same pressure, the outlet velocity and mass flow of the single vertical tube are larger than that of the two vertical tubes, and the turbulent kinetic energy of the outlet is also larger than that of the two vertical tubes.

2) compared with the flow field diagram of medium cross-section, it is found that the internal flow velocity of single vertical tube model is not as uniform as that of double vertical tube.

3) the circular nozzle type has a stronger effect on fluid aggregation than the square nozzle type. If the square nozzle does not have a long horizontal tube, the slant of the jet flow will occur.

We can judge that P1 type is the most in four kinds of models to meet the requirements of tank washing work, in the process of this type of nozzle jet has a larger turbulent kinetic energy, and jet direction accuracy, fast speed, huge impact, is a good way to finish the work of cleaning the petroleum tank.

References
[1] Yunyun Bai. Application of rotary jet agitator in crude oil storage tank 2013 J. China New Technology and New Products, 25(3): 14.
[2] Liye Liu, Fanli Meng, Weiqiang Wang. The application and development of agitator in crude oil storage tank 2015 J. Contemporary Chemical Industry, 32(2): 386-388.
[3] Luliang Xu, Xiaofang Zhang, Yonggang Huang. Automatic cleaning and resource disposal of large oil tank sludge 2002 J. Petrochemical Environmental Protection, 25 (4): 26-30.
[4] Liangliang Xu. Reduction and recycling technology of oil tank bottom mud 2004 J. Petroleum and Natural Gas Chemical Industry, 36(5): 369-74.
[5] Qiuying Du. Numerical simulation of high pressure jet cleaning of tank for mechanical cleaning 2016 D. Daqing: Northeast Petroleum University.
[6] Zhian Deng, Xudong Ma, Haijing Shen. Numerical simulation of jet cleaning with different shapes of nozzles 2014 J. Petroleum Machinery, 42(4): 81-84.
[7] Lei Zhuang. Nozzle for high pressure water jet cleaning 2003 J. Cleaning World, 19(12): 28-31.
[8] Wenhui Zhou. Numerical simulation of internal and external flow field of high pressure water jet nozzle 2008 D. Lanzhou: Lanzhou University of Technology.