CHANG CHEN
SHIFU ZHANG
ZHI HAN
JIKUN GUO
QIXIN ZHANG
and
DONGMEI ZHANG

CFD numerical simulation of internal flow field in fuel tank during large-flow rapid refuelling process

With the gradual improvement of modern social life, more and more attentions are attracted to higher efficiency of refuelling process. However, large flow rapid refuelling process may result in unexpected turbulence and violating the safety standards. Since there are a few of existing literature researches the internal flow field in fuel tank during large flow rapid refuelling process, this paper carried out numerical simulation research for the gas-liquid two-phase flow in oil tank under the condition of vertical refuelling and tilting filling in that circumstance, and devoted to research characteristics of the flow field inside the fuel tank and the cause of the foam formation under different fuelling conditions by using CFD software. The result of this study was of certain reference value to solve the problems of large foam and turbulence in flow field during large flow rapid refuelling process.

Keywords: *Numerical simulation, VOF model, internal flow field, large-flow rapid refuelling process, CFD.*

1. Introdunction

Rapid-large flow refuelling technology focuses on improving fuel efficiency by optimizing the traditional refuelling devices and refuelling methods, such as enlarge the fuel gun caliber, improve the pump flow, and increase the fuel pressure, etc. The shortcomings of traditional refuelling technology such as small flow and poor efficiency are prominent and the evolving society has put forward higher demands to it. During the refuelling process, the diesel will produce foam simultaneously. And, it will have an impact on the safety usage of diesel oil. What is more, the greater flow of refuelling diesel, the more bubble will be produced which will reduce the fuel efficiency seriously.

The internal flow field in fuel tank during refuelling process has been of considerable interest to researchers in recent years, and a brief summary of some of the relevant researches is presented as follow. Lapin, A presented a two-dimensional computational fluid dynamics study of the two-phase gas-liquid flow in bubble columns and set up the physical model behind the simulation and typical results obtained in some two-phase gas-liquid flow situations (Lapin and Lübbert, 1994). Zhao, Allan G develop a hybrid modelling method to address the premature shut-off of the fuel dispenser nozzle during automotive fuel tank refuelling. The fuel tank system is divided into a number of subsystems each with its own specific model. What is more, he has developed a lumped parameter model to simulate the storage tank and vent tubes while a three-dimensional CFD model is employed to simulate the complex multiphase flow in the filler pipe (Zhao, 2003). Chen, Zuo Bing adopted VOF discrete phase model and donor-acceptor method to analyze the flow field in the fuel tank which is swirling by symmetry axis. They have studied and simulated the speed variation of the flow field in the tank (Chen, et al., 2005). Ma, F established a three-dimensional mathematical model of internal flow field in a soil bore-enlarged nozzle with the CFD software. Their research results show that the diffuse angle and divergent cone length of the nozzle influence the internal flow field more and the effect of the cylindrical segment length is relatively less (Ma and Zhang, 2006). Peng, Chengyun aimed at the motorcycle fuel tank forming process and studied on the strain/stress character and the influence of the major technologic parameter on its thickness variation (Peng, 2006). Chen, J. Q determined the proper mesh discretization scheme and applied the proper boundary conditions based on the Gambit software, then established the reasonable numerical simulation model for the gas-liquid two-phase flow during the refuelling process (Chen, et al., 2011). Mastroianni, M experimentally studied the refuelling of a transparent rectangular fuel tank fitted with a standard filler pipe and roll-over valve (Mastroianni, et al., 2011). Zhang, S. Q developed a numerical simulation method for nitrogen-enriched air (NEA) flow in multi-bay fuel tank according to the characteristic of multi-bay fuel tank (Zhang, et al., 2013). Chen, L numerically studied the internal flow

Messrs. Chang Chen, Shifu Zhang, Jikun Guo, Qixin Zhang, Dongmei Zhang, Army Logistics University of PLA, Chong Qing, 401311 and Zhi Han, Oil Representative Office of PLA, Beijing 100081, China. Corresponding author Chang Chen, E-mail: 408430734@qq.com

fields of the commonly used nozzle and a new main nozzle composed of two nozzle needles connected in series by means of a three-dimensional model implemented by the computational fluid dynamics technique (Chen, et al., 2015).

It must also be mentioned that much work so far has focused on the internal flow field in fuel tank under regular refuelling circumstance. Little is known, however, about the internal flow field in fuel tank during large flow rapid refuelling process which may be resulted in unexpected turbulence appearance and violating the normal standards. And we demonstrate through an extensive literature review that the existing researches are not capable of handling the specifics of problem in this study.

What is different from previous studies is that, this paper focuses on the gas-liquid two-phase flow in oil tank under the condition of vertical refuelling and tilting filling during large-flow rapid refuelling process, and devoted to research characteristics of the flow field inside the fuel tank and the cause of the foam formation under different fuelling conditions by using CFD software.

This paper proceeds as follow. Section 2 presents models and algorithms for numerical calculation. Section 3 carried out numerical simulation of flow in oil tank under the condition of vertical refuelling during large-flow rapid refuelling process. Section 4 numerically researched and analyzed the flow in oil tank under the condition of tilting refuelling during large-flow rapid refuelling process. Section 5 concludes the paper.

2. Models and algorithms

2.1 Turbulence model

The commonly used turbulence models include single equation model (Spalart-Allmaras), double equation model, Reynolds stress model and large eddy simulation (Winkler, et al., 2017). What is more, the two equation model includes standard k–ε model, RNG k–ε model and capable k–ε model. The standard k–ε model within the two equation simulation is adopted in this paper. As part of this model, equation of turbulent kinetic energy k and equation of turbulent dissipation rate ε must be solved first.

$$\rho\frac{dk}{dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k+G_b-\rho\varepsilon-Y_M \quad \text{... 1}$$

$$\rho\frac{d\varepsilon}{dt}=\frac{\partial}{\partial x_i}\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial k}{\partial x_i}+C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k+C_{3\varepsilon}G_b\right)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad \text{... 2}$$

In the above formula, G_k represents the turbulent kinetic energy which caused by average velocity gradient, G_b represents the turbulent kinetic energy which caused by buoyancy. Y_M shows the effect of compressible turbulence on the overall dissipation rate. $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$, are empirical constants. σ_k and σ_ε are prandtl numbers of turbulent kinetic energy k and turbulent dissipation rate ε respectively. Constant coefficient are listed in Table 1.

Table 1. Constant coefficient of Standard k–ε model

$C_{1\varepsilon}$	$C_{2\varepsilon}$	$C_{3\varepsilon}$	σ_k	σ_ε
1.44	1.92	0.09	1.0	1.3

2.2 Volume of fluid model

During the refuelling process, there are two kinds of phases in the tank, one is liquid diesel, and another is air. Since two phases are not harmonious, there is a contact surface between them. Therefore, the volume of fluid model in the multiphase flow model is suitable for numerical calculation. The volume of fluid model (VOF) is a surface tracking method under a fixed Eulerian grid in which different fluid components share a set of momentum equations and are recorded in each calculation unit in the full flow field (Potham, et al., 2017).

α_q represents the volume fraction of q_{th} phase,

$$\begin{cases}\alpha_q=0, & \textit{the } q_{th} \textit{ phase does} \text{ not } \textit{exit in unit}\\ \alpha_q=1, & \textit{only } q_{th} \textit{ phase exit in unit}\\ 0<\alpha_q<1, & \text{multi}-\text{phases } \textit{exit in unit including } q_{th} \textit{ phase}\end{cases} \quad \text{...3}$$

What is more,

$$\sum_{q=1}^{n}\alpha_q=1 \quad \text{... 4}$$

The control equations for the multi-phase flow model are as follows.

Continuity equation:

$$\frac{\partial}{\partial t}\left(\alpha_q\rho_q\right)+\nabla\cdot\left(\alpha_q\rho_q\vec{v}_q\right)=0 \quad \text{... 5}$$

In formula, ρ_q reprents the physical density of q_{th} phase, $\vec{v}_q$ reprents the velocity of q_{th} phase.

Momentum conservation equation:

$$\frac{\partial}{\partial t}\left(\rho\vec{v}\right)+\nabla\cdot\left(\rho\vec{v}\vec{v}\right)=-\nabla\rho+\nabla\cdot\left[\mu\left(\nabla\vec{v}+\nabla\vec{v}^T\right)\right]+\rho\vec{g}+\vec{F} \quad \text{...6}$$

Energy equation:

$$\frac{\partial}{\partial t}\left(\rho E\right)+\nabla\cdot\left[\vec{v}\left(\rho E+p\right)\right]=\nabla\cdot\left(k_{eff}\nabla T\right)+S_n \quad \text{... 7}$$

$$E=\frac{\sum_{q=1}^{n}\alpha_q\rho_q E_q}{\sum_{q=1}^{n}\alpha_q\rho_q} \quad \text{... 8}$$

In formula (7), (8), k_{eff} reprents effective heat transfer coefficient, E_q represents energy of each phases.

3. Numerical simulation of the flow field under the condition of vertical refuelling during large-flow rapid refuelling process

3.1 The establishment of geometric model

It is necessary to establish appropriate geometric model and mathematical model to simulate the internal flow field of the fuel tank during the refuelling process, which determines the correctness of the numerical simulation results. The establishment of the model must follow two major principles: first, the authenticity of the model; second, the feasibility of mathematical calculations. The authenticity requirements of the established geometric model is tried to reflect the characteristics of the object so as to guide the practical application effectively. The feasibility of mathematical calculation is that the model must be simplified properly, so that the model can be effectively calculated and analyzed by simulation tools. The three-dimensional geometric model is established as in Fig.1. The model dimensions are 700mm × 700mm × 800mm, the diameter of tank port is $\phi = 80\text{mm}$ and the height is h = 80mm. Considering the complexity of the internal structure of the refuelling gun, the fuel gun structure is simplified as a 200 mm long pipe, which placed vertically in the tank mouth.

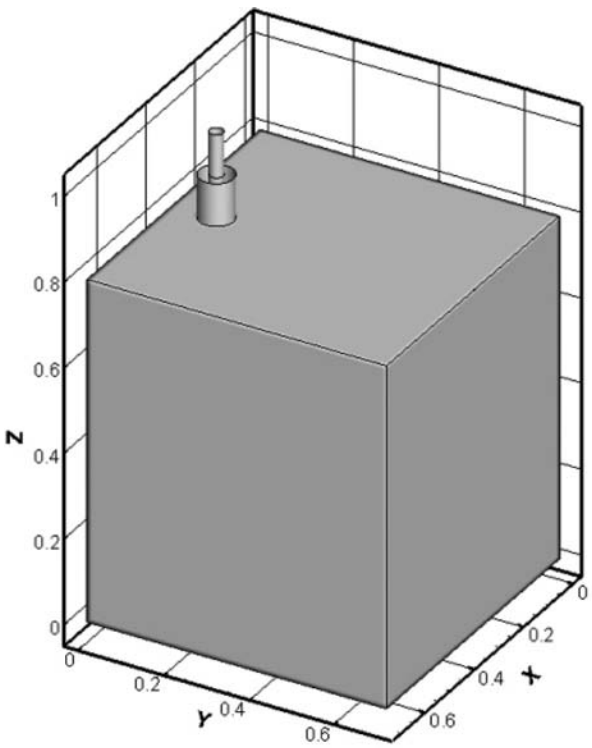


Fig.1. Geometric model

3.2 Grid partition

Fluent Soft have developed the Gambit preprocessor which has flexible interface and highly automated grid generation function. In this paper, we used this tool to divide the grid for guaranteeing the high quality of the grid. And, the meshing results are shown in Fig.2.

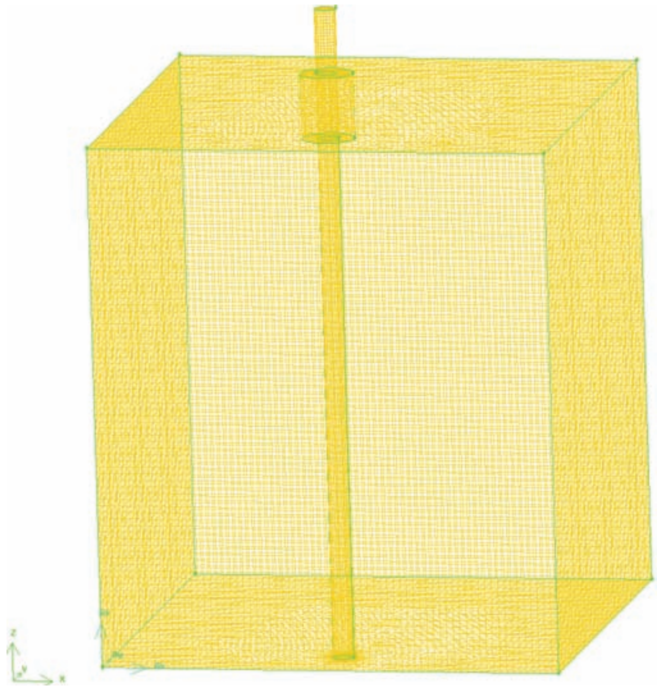


Fig.2. Mesh grids

3.3 Boundary conditions

According to the liquid flow inside the tank, the boundary condition selects the velocity inlet and the pressure outlet. The upper end of the fuel gun is the velocity inlet, and the annular area on the upper end of the tank mouth is the pressure outlet. In addition to the entrance and exit, the remaining boundaries are set to fix wall conditions. The calculated operating conditions for this numerical simulation are shown in Table 2.

3.4 Solver and parameter setting

Starting the Fluent solver, the “standard two-equation” turbulence model is selected with the single-3d calculation accuracy, and the pressure-based unsteady implicit solver is adopted. The two-phase fluid material are set as diesel and air, and the diesel density is modified to 830 kg/m^3. The gravity effect is taken into account, and other operating condition keeps the default setting. At the entrance of the oil tank, the diesel volume fraction is set to 1, and the diesel speed is set to 3.73m/s. At the outlet of the oil tank, the pressure is set to 1 standard atmospheric pressure. What is more, the convergence factor of the solver is set as follows: pressure equals 0.3, density equal 1, volume force equals 1, momentum equals 0.2, turbulence kinetic energy equals 0.8, turbulent dissipation rate equals 0.8, and turbulent viscosity equals 1. The pressure velocity coupling is selected by the

Table 2. Computational conditions

Refuelling gun caliber (mm)	Filling flow (L/min)	Refuelling gun type	Filling angle	Entrance velocity (m/s)	Outlet pressure (Pa)
32	180	Straight, regular	vertical	3.73	0

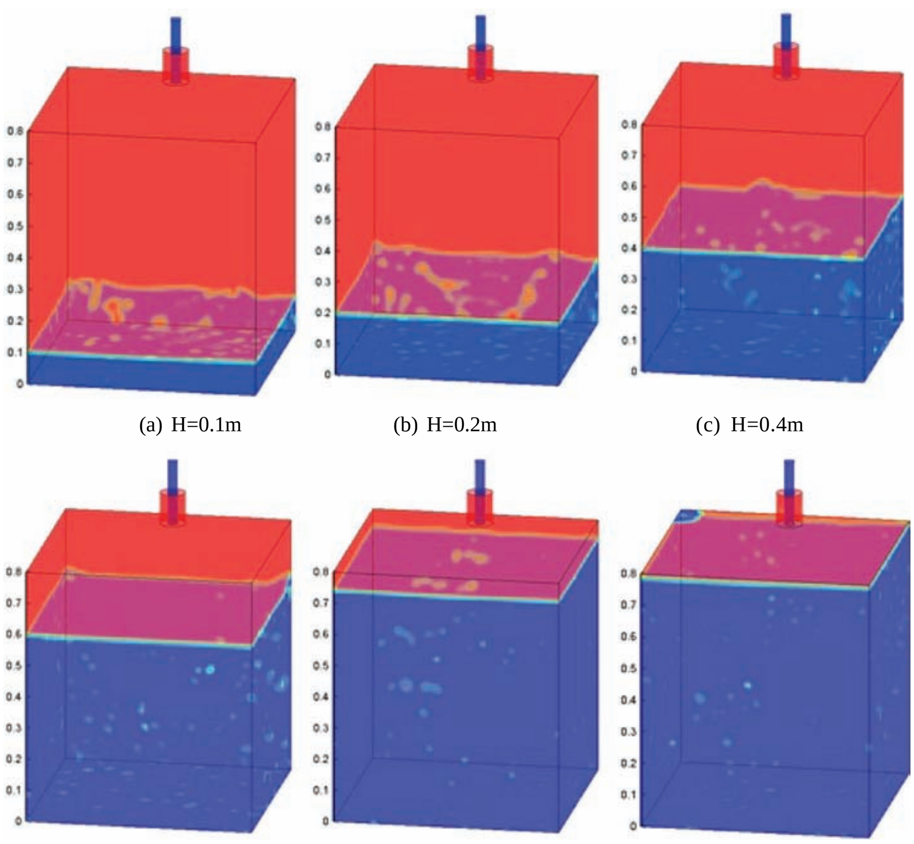
"PISO" algorithm. In order to accelerate the convergence of calculation, the discretization scheme of momentum equation and turbulent kinetic energy equation is chosen as "first order upwind" format. All residual convergence accuracy is defined as 0.001; the initial value of the velocity at the entrance of the oil tank is taken as the global initial value. Start the iteration calculation and save the data file automatically every 0.5s. The interphase mass and heat transfer are not considered during the calculation.

3.5 Results and analysis

When the calculation is complete, the six stages that the level of the liquid reaches the fuel tank 0.1m, 0.2m, 0.4m, 0.6m, 0.75m, 0.8m scale were selected as the study subjects. And the three-dimensional contours of oil and gas volume fraction (Fig.3), velocity contours of tangent plane of oil flow (Fig.4), velocity vectors of oil and gas volume fraction of the tangent plane of oil flow (Fig.5), and contours of oil and gas volume fraction of the tangent plane of oil flow (Fig.6) were obtained during the refuelling process.

In Fig.3, the red materials indicate gas phase (air), blue materials indicate liquid phase (diesel), the transition colour materials from red to blue can be considered as gas-liquid mixture phase (bubble). In the early stages as shown in Fig.3(a)-(b), the amount of bubbles in the diesel fuel is small, but the volume is larger and the diesel flow field is more turbulent. As the refuelling process progresses, the volume of the bubbles inside the diesel fuel becomes smaller in the medium term as shown in Fig.3(c)-(d). At the same time, it can be seen that the bubbles are widely distributed throughout the diesel oil and can be considered as turbulence in the diesel oil at this stage. At the end stage as shown in Fig.3(e)-(f), the number of bubbles in the diesel oil is significantly reduced and mainly concentrated in the upper half of the diesel layer. The lower half of the flow field are relatively stable. It can be considered that the turbulent area has been moved up with the liquid level, and the stable flow area appears in the lower part of the diesel oil layer.

In Fig.4, the flow rate decreases from red to blue, and it can be seen that the maximum velocity in the flow field appears in the position where the oil column is in contact with the diesel oil surface. And as the liquid level increases, the flow rate at the oil column and diesel oil surface contact position is gradually reduced. In Fig.4(a)-(d), the impact of the oil column on the flow velocity of the diesel oil is always affected from the contact liquid level to the bottom. But, in the Fig.4(e)-(f), the underlying part of the diesel oil is no longer affected by the oil column, which further confirms the analysis in Fig.3, indicating that as the liquid level rises, the impact of the oil column on the diesel layer began to decrease and the diesel oil within the turbulence area is correspondingly smaller.



(a) H=0.1m (b) H=0.2m (c) H=0.4m

(d) H=0.6m (e) H=0.75m (f) H=0.8m

Fig.3 Three-dimensional contours of oil and gas volume fraction.

In Fig.5, the length of the arrow indicates the fluid velocity, and the direction of the arrow indicates the flow direction of the fluid. Taking the left side diesel oil of the oil column as an example, it can be seen that a clockwise swirling flow is formed in the diesel flow field, and there is also a clockwise rotation in the gas flow field near the oil column and the diesel oil surface. The reasons can be analyzed as follows. In the diesel flow field, under the impact of the oil column, the diesel oil around the oil column moves downward incidentally, then it moves to the left side of oil tank after touching the end plate, thus forming a clockwise swirl. In the gas flow field, as the rapid decline of the oil column, the surrounding air was driven to downward movement. When the air meets the liquid level of the diesel, a part of the air moves to the left to form a clockwise swirl, while another part of the air enters the diesel layer along the gap between the

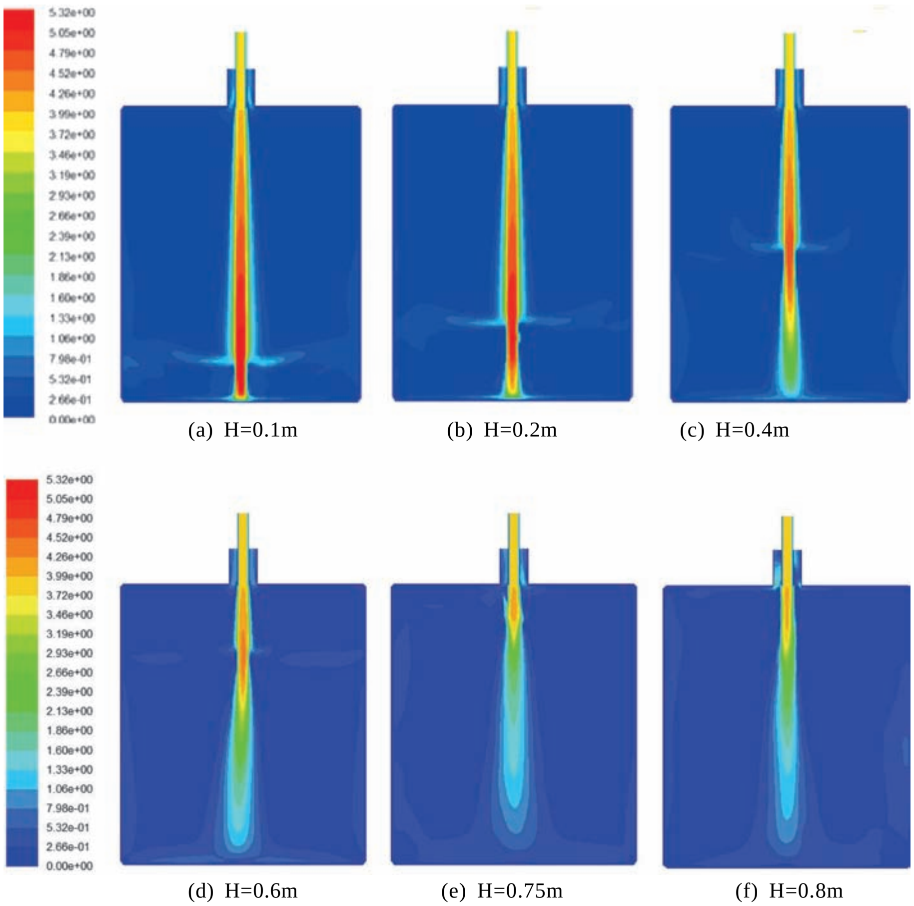


Fig.4 Velocity contours of tangent plane of oil flow

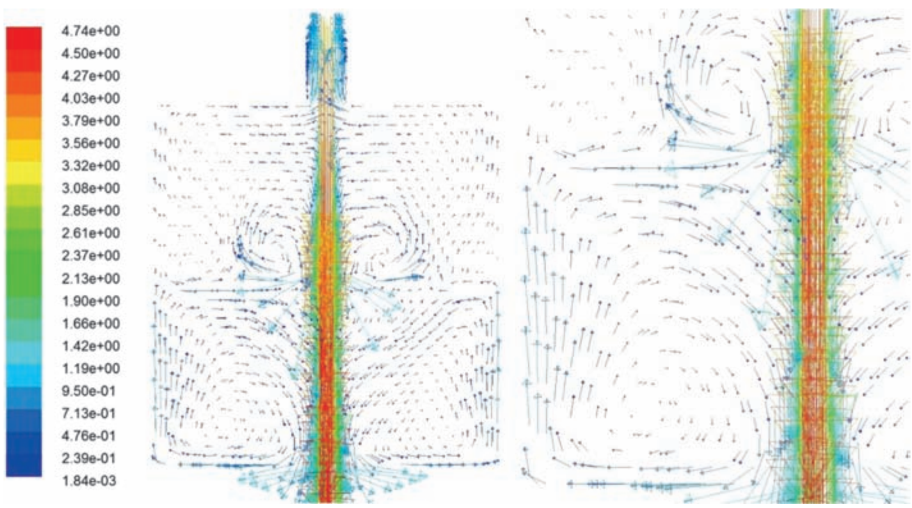


Fig.5 Velocity vectors of oil and gas volume fraction of the tangent plane of oil flow

oil column and the liquid surface, thus forming a bubble as shown in Fig.6.

According to the above analysis, by integrating the three-dimensional contours of oil and gas volume fraction (Fig.3) and velocity contours of tangent plane of oil flow (Fig.4), it can intuitively reflect the generation process of the bubbles in the diesel oil layer and the change of the turbulent area in the process of refuelling. By integrating the velocity contours of tangent plane of oil flow (Fig.4) and velocity vectors of oil and gas volume fraction of the tangent plane of oil flow (Fig.5), the change of the impact speed of the oil column in the flow field as well as the flow of gas-liquid two-phase flow can be found. By integrating velocity vectors of oil and gas volume fraction of the tangent plane of oil flow (Fig.5) and contours of oil and gas volume fraction of the tangent plane of oil flow (Fig.6), it is possible to analyze how the gas is brought into the oil by the oil column to produce bubbles.

4. Numerical simulation of the flow field under the condition of tilting filling during large-flow rapid refuelling process

Compared to the refuelling gun used in vertical direction, it is more common that the refuelling gun are filled with oil under inclined condition. So, it is meaningful to conduct the numerical simulation of the internal flow field of the fuel tank under the condition of using refuelling gun at a certain angle of inclination.

4.1 The establishment of geometric model

The dimensions of model established in this section are the same as those of the above model except that the refuelling gun is tilted at the inlet of the tank (the angle between the barrel axis of the cylinder and the vertical direction is 30°) and the tank port is removed for the sake of ease observation, the other settings and simplification are the same as the previous model. The three-dimensional geometric model is established as in Fig.7.

4.2 Grid partition

According to the principles and methods of the previous section, we use Gambit software to divide the geometric model. The hexahedral elements are adopted as grid units. The volume of the refuelling gun is relatively small compared to whole oil tank, but it is a rather special location, therefore,

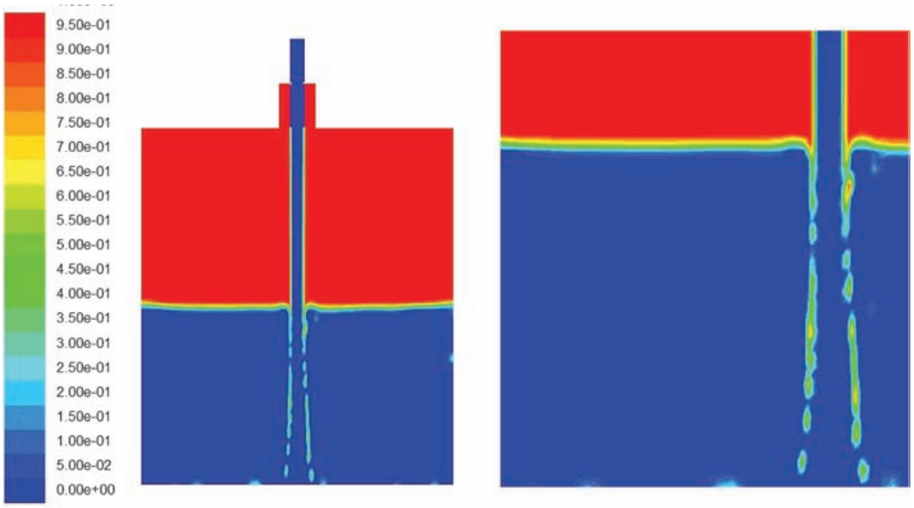


Fig.6 contours of oil and gas volume fraction of the tangent plane of oil flow

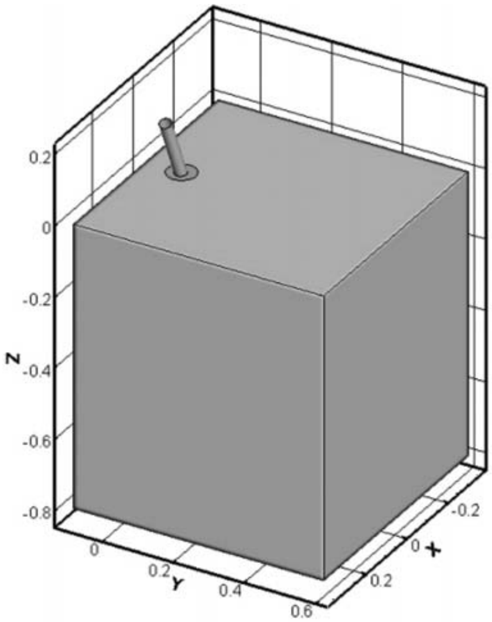


Fig.7 Geometric model

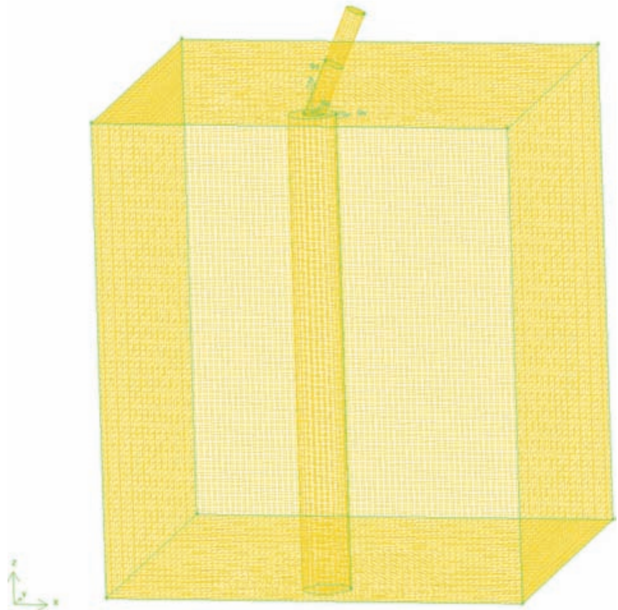


Fig.8 Mesh grids

the mesh between the fuel gun and the fuel tank mouth is encrypted which is shown in Fig.8.

4.3 Boundary conditions and calculation cases

The boundary condition selects the velocity inlet and the pressure outlet. The upper end of the nozzle is set as the velocity inlet, and the annular area on the upper end of the tank mouth is set as the pressure outlet. Except for the entrance and exit, the remaining boundaries are set to fixed wall conditions (wall). The calculated operating conditions for this section are shown in Table 3.

4.4 Results and analysis

When the calculation is complete, the six stages that the level of the liquid reaches the fuel tank 0.1m, 0.2m, 0.4m, 0.6m, 0.75m, 0.8m scale were selected as the study subjects. And the three-dimensional contours of oil and gas volume fraction (Fig.9), velocity contours of tangent plane of oil flow (Fig.10), and velocity vectors and contours of oil and gas volume fraction of the tangent plane of oil flow (Fig.12) were obtained during the refuelling process.

It can be seen from Fig.9 that, at the early stage as shown in Fig.9(a), in the area where the oil column is impacted in the right and bottom of the tank, the bubbles in the liquid phase are larger and densely distributed, and the flow field of the diesel is disordered. As the refuelling process progresses as shown in Fig.9(b)-(d), the bubble volume in the diesel layer is obviously smaller, and is mainly concentrated in the impact zone of the oil column. What is more, there are also some bubbles at the bottom of the diesel layer which shows that the impact of the oil column has always affected the bottom of the diesel layer. To the end of refuelling process as shown in Fig.9(e)-(f), the number of bubbles in the diesel

Table 3. Computational conditions

Refuelling gun caliber (mm)	Filling flow (L/min)	Refuelling gun type	Filling angle	Entrance velocity (m/s)	Outlet pressure (Pa)
32	180	Straight, regular	30 degree angle leaned	3.73	0

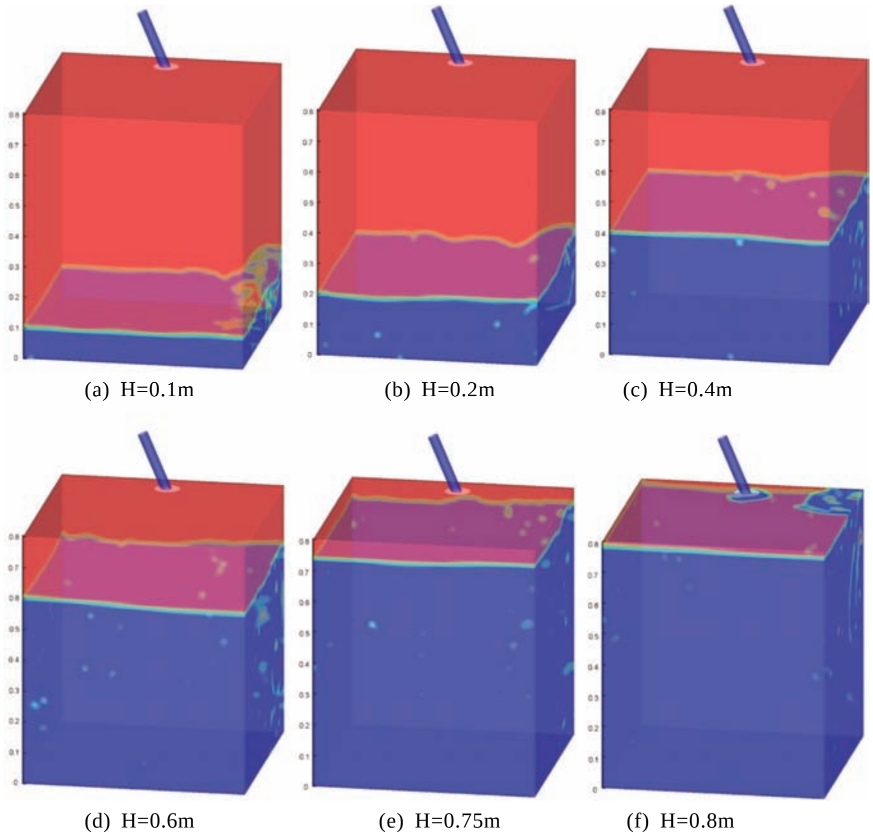


Fig.9 Three-dimensional contours of oil and gas volume fraction

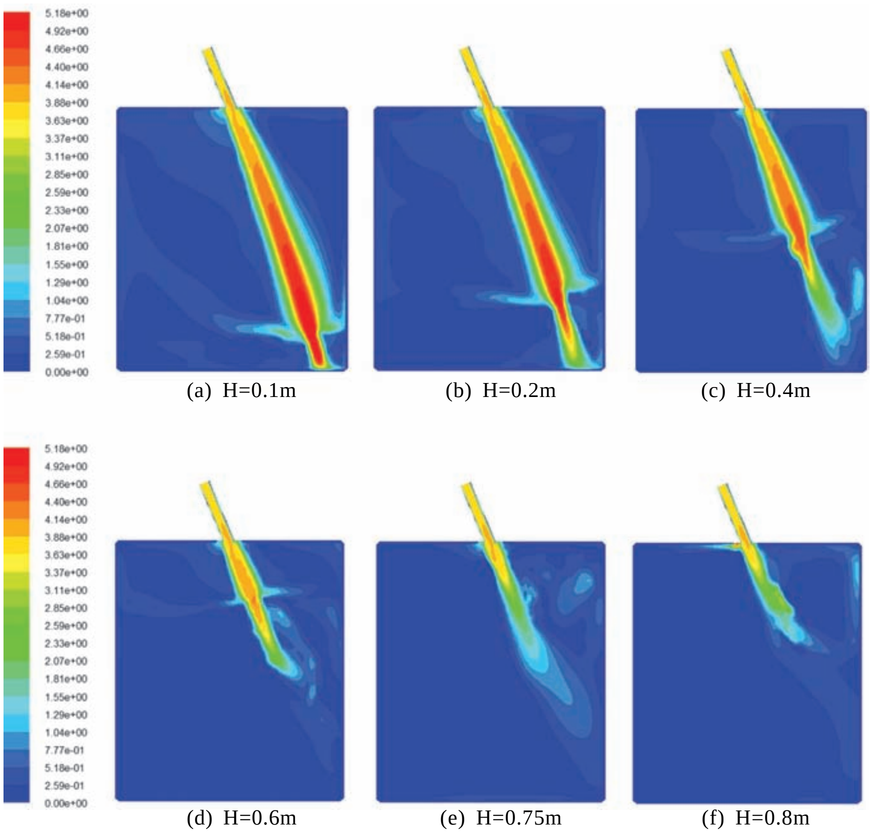


Fig.10 Velocity contours of tangent plane of oil flow

oil layer, especially in bottom position, is gradually reduced, and it can be deduced that the turbulence zone formed by the impact of the oil column has moved up with the liquid level, and a steady flow region appears in the lower part of the diesel layer.

In Fig.10, the maximum velocity in the flow field appears in the position where the oil column is in contact with the diesel oil surface. And as the liquid level increases, the flow rate at the oil column and diesel oil surface contact position is gradually reduced. In Fig.10(a)-(c), the impact of the oil column on the flow velocity of the diesel oil is always affected from the contact liquid level to the bottom. But, in the Fig.10(d)-(f), the underlying part of the diesel oil is no longer affected by the oil column, which further confirms the analysis in Fig.9. Meanwhile, comparing the results of the vertical refuelling pattern of the previous section with the results of the tilted refuelling pattern in this section, it can be found that when the liquid level reaches 0.6 m, the velocity change caused by the vertical oil column still has an effect on the bottom of the diesel oil while the tilted oil column has no effect on it as shown in Fig.11. It is shown that the flow of the diesel flow field is more stable under the condition of tilting refuelling.

In Fig.12, the gas on both sides of the oil column moves downward with the oil column, and when it comes into contact with the gas-liquid interface, a part of the gas diffuses to both sides to form a swirling flow, and the other part of the gas flows along with the oil column and get into the diesel layer to form bubbles. In the liquid phase flow field, due to the small space on the right side of the oil column, a clockwise swirling turbulent zone is formed under the impact of the high velocity oil flow, and the flow field is quite chaotic. The left space of the oil column is larger, and the advection zone is formed ultimately. The swirl only appears near the wall and the oil pillar area. It can be seen that the flow field of diesel is more steady and there is no large turbulence zone during tilting refuelling process.

Through above analysis, on the one

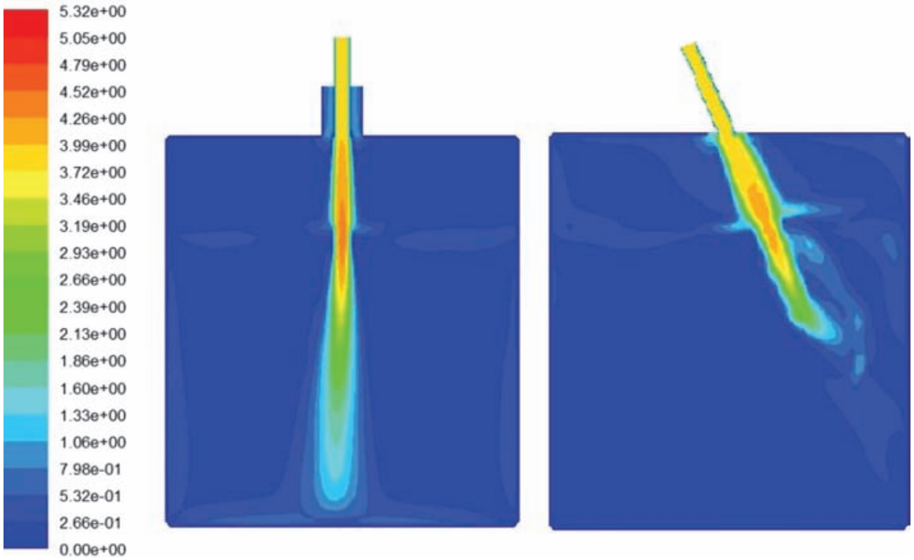


Fig.11 Comparison of velocity contours by two refuelling ways when H=0.6m

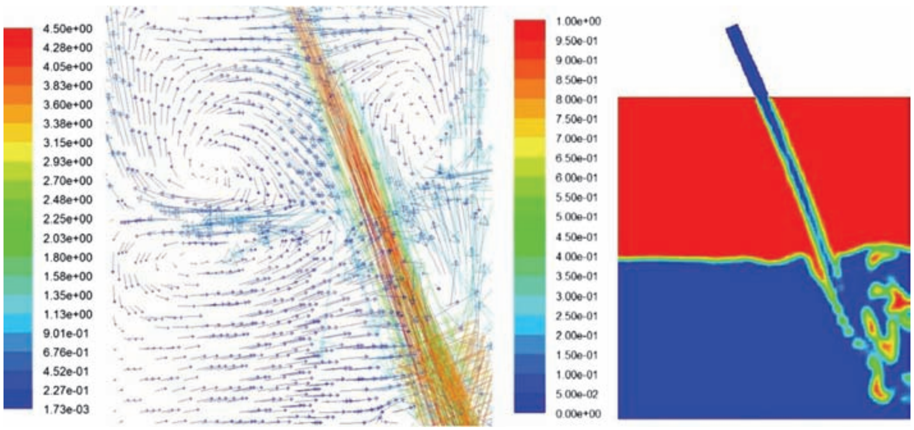


Fig.12 Velocity vectors and contours of oil and gas volume fraction of the tangent plane of oil flow

hand, we understand the specific circumstances of gas-liquid two-phase flow during tilting refuelling process. On the other hand, through the comparison of the vertical refuelling pattern of the previous section with the tilted refuelling pattern in this section, it has been found that the flow of the diesel flow field is more stable under the condition of tilting refuelling.

5. Conclusions

This paper used computational fluid dynamics fluent software and adopt VOF multi-phase flow model for numerical simulation research of the gas-liquid two-phase flow in oil tank under the condition of vertical refuelling and tilting filling during large-flow rapid refuelling process. The characteristics of the flow field inside the fuel tank and the cause of the foam formation under different fuelling conditions are revealed. The results of the two sets of numerical simulation can be summarized as follows:

When the inclined filling method is adopted, the flow field in the tank is relatively stable, and the bubbles are obviously reduced. This is mainly because the impact position of the oil column is at the edge of the liquid phase flow field, and the turbulence zone is limited to the edge of the flow field. At the same time, the oil tank wall limits the impact of the oil column, so the whole liquid phase flow field is relatively stable.

During the refuelling process, the flow field in the tank is divided into two parts: gas field and liquid field. For the gas flow field, the gas on both sides of the oil column moves downward accompanying the oil column. When the air encountered the gas-liquid two-phase interface, a part of the air moves along with the interface to form a clockwise swirl, while another part of the air enters the diesel layer along the gap between the oil column and the liquid surface, thus forming the bubbles in the liquid. In the liquid flow field, the amount of bubbles is determined by the amount of gas entering the liquid phase, which is affected by the velocity of the gas and the size of the gap in the liquid phase of the oil column. These two factors mainly depend on the flow velocity and flow pattern of the oil column. The rule is that: the smaller the oiling speed from the refuelling gun, the more neat will be the flow. Furthermore, the smaller is the velocity of the gas, the less the amount of gas will enter the liquid phase. Consequently, the less bubble would be produced.

Acknowledgments

This paper is supported in part by The National Key Technology R&D Program of China (2017YFC0806306), Education Science fund of the Military Science Institute of Beijing, China (No.2016JY481), and Chongqing Graduate Scientific Research Innovation Project, China (No.CYB17148).

Reference

1. Chen, J. Q., Zhang, N., Wang, J. H., Zhu, L., Shang, C. (2011). CFD numerical simulation onto the gas-liquid two-phase flow behavior during vehicle refuelling process. *Environmental Science*, 32(12): 3710-3716.

Continued on page 153