CHANG CHEN
ZHI HAN
QIXIN ZHANG
SHIFU ZHANG
JIKUN GUO
and
SHIQIANG SONG

Numerical simulation and experimental study on the influence of novel refuelling nozzle with rectifier tube in the process of diesel refuelling

When applying the traditional refuelling gun in large-flow rapid refuelling process, large amount of bubbles are produced under the strong impact between the high speed oil column and the tank wall or internal oil surface, which will reduce the refuelling efficiency seriously. In this paper, a combination of CFD simulation and refuelling experiment has been conducted to analyse the effect of rectifier tube in flow field stability and diesel foam reduction when applying to the process of large-flow rapid refuelling. The simulation result has been further discussed by comparing with the refuelling experiment data. And, it has been proved that the application of rectifier tube in the refuelling nozzle can control the turbulent state of the flow field effectively and reduce foam generation significantly, so as to make contribution to improve the refuelling efficiency and operation security.

Keywords: *Numerical simulation, experimental study, rectifier tube, volume of fluid model, PISO algorithm.*

1. Introdunction

During the process of diesel refuelling, due to the existence of a certain height from the outlet of refuelling gun to the surface of internal diesel oil, coupling with the natural property of the diesel oil, a large amount of diesel bubbles will be produced when the oil column sprayed out of refuelling nozzle impact the oil surface in case of rapid refuelling condition. At the same time, under the action of shock oscillation, the flow field in the tank will be extremely disordered, resulting in reducing refuelling efficiency greatly and increasing risk of refuelling operation. Many scholars have done a lot of researches on the regulation of internal flow and tried to limit the production of bubbles. Lapin, A set up the physical model behind the simulation and typical results obtained in some two-phase gas-liquid flow situations (Lapin and Lübbert, 1994). Lu, Yuanwei built the fairing rings in the inlet and outlet of the vertical parts of inverse U bend respectively in order to restrain the effect of bend on the flow in it (Lu, et al., 2002). Deng, Q studied the phenomenon of liquid drop impact onto the surface of a deep pool of the same liquid by using high-resolution digital photography (Deng, et al., 2007). Chen, J. Q established the reasonable numerical simulation model for the gas-liquid two-phase flow during the refuelling process based on the Gambit software (Chen, et al., 2011). He Zhixia investigated the critical conditions of cavitation inception in diesel injector nozzles and developed a correlation of critical cavitation number with above 6 geometrical and dynamic parameters for nozzle flow through statistical analysis using the Quasi-Newton (BFGS) and Universal Global Optimization (UGO) methods (He, et al., 2015). Gong, Chao implemented CFD (computational fluid dynamics) to study the behaviours of the internal nozzle flow and the corresponding spray characteristics of conical-orifice injector nozzle (Gong and Baar, 2017). Leng Xianyin has been conducted numerical study to examine the effects of the V-type intersecting hole structure on the internal flow of a nozzle and the initial stage of the fuel-air mixing processes (Leng, et al., 2017). JPV Sotillo studies the influence of internal nozzle flow characteristics over a large spectrum of experimental conditions and diagnostics that were carried out for two nozzle geometries and three different fuels (Sotillo, 2017). Wang Xiang presented the modelling of nozzle cavitating flow and detailed coupling of nozzle exit flow and spray. Their final numerical results clarified that the contribution of cavitation phenomena to primary breakup is quite appreciable. What is more, the evolution of the high-pressure and evaporating diesel spray structure greatly changes as cavitation occurs inside fuel injection nozzles (Wang, et al., 2017). Zeidi modeled six-hole diesel injector nozzle with different needle lifts, and comprehensively evaluated the effects of different needle lifts on mass flow rate, discharge coefficient and length of cavitation. Discharge coefficient, mass flow rate and length of cavitation region were compared under different boundary

Messrs. Chang Chen, Qixin Zhang, Shifu Zhang, Jikun Guo, Army Logistics University of PLA, Chong Qing, 401311, Zhi Han, Oil Representative Office of PLA, Beijing 100081 and Shiqiang Song, Logistics Training Center of PLA Rocket Force, Hebei 075000, China. Corresponding author Chan Chen, E-mail: 408430734@qq.com

conditions and the extreme temperature spike at the center of an imploding cavitation bubble was also analyzed as a function of time and initial bubble size (Zeidi, et al., 2017).

It must also be mentioned that much work so far has focused on the internal flow field in fuel tank under regular refuelling circumstances. And we demonstrate through an extensive literature review that the existing researches have not made mention of applying the rectifier tube in the improvement of refuelling nozzle for preventing the bubble generation. What is different from previous studies is that, this paper combined the experimental and numerical methods to investigate the effects of rectifier tube in stabilizing flow field and reducing foam during large-rapid refuelling process.

This paper proceeds as follow. Section 2 presents theoretical models and algorithms for this study. Section 3 carries out numerical simulation of the influence of novel refuelling nozzle with rectifier tube. Section 4 experimentally studies the effects of novel refuelling guns with the application of rectifier tube during large-flow rapid refuelling process. Section 5 concludes this paper.

2. Theoretical analysis

2.1 Volume of fluid model

In the process of diesel refuelling, those two phases including diesel and air are mixed in oil tank. Since two substances are incompatible, there is a contact interface among them. In this paper, we adopted the volume of fluid model in numerical calculation. The volume of fluid model (VOF) is a surface tracking method that different fluid components share a set of momentum equations and are recorded in each calculation unit in the full flow field.

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

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

What is more,

$$\sum_{q=1}^{n} \alpha_q = 1 \quad ... (2)$$

The control equations for the multiphase flow model are as follows:

(1) Momentum conservation equation.

$$\frac{\partial}{\partial t}\left(\rho \vec{v}\right) + \nabla\cdot\left(\rho \vec{v}\vec{v}\right) = -\nabla\rho + \nabla\cdot\left[\mu\left(\nabla\vec{v} + \nabla\vec{v}^{T}\right)\right] + \rho\vec{g} + \vec{F} \quad ... (3)$$

(2) Continuity equation.

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q\right) + \nabla\cdot\left(\alpha_q \rho_q \vec{v}_q\right) = 0 \quad ... (4)$$

In formula (4), ρ_q reprents the physical density of q_{th} phase, $\vec{v}_q$ reprents the velocity of q_{th} phase.

(3) Energy equation.

$$\frac{\partial}{\partial t}\left(\rho E\right) + \nabla\cdot\left[\vec{v}\left(\rho E + p\right)\right] = \nabla\cdot\left(k_{eff}\nabla T\right) + S_n \quad ... (5)$$

$$E = \frac{\sum_{q=1}^{n} \alpha_q \rho_q E_q}{\sum_{q=1}^{n} \alpha_q \rho_q} \quad ... (6)$$

In formula (5) and formula (6), k_{eff} reprents effective heat transfer coefficient, E_q represents energy of each phases.

2.2 Turbulence model

After preliminary calculation, it has been found that the flow parameter RE of diesel oil closing the mouth of refuelling nozzle is above 10000, indicating the diesel is in a state of strong turbulent. Fluent software provides several turbulence calculation models, among them, the RNG *k–ε* model can react better to the effects of transient flow and streamline bending, so this model is suitable for handing the problem in this paper. The RNG *k–ε* model is derived from the transient N-S equation, and the *k* equation and *ε* equation are presented as followed.

$$\frac{\partial}{\partial t}\left(\rho k\right) + \frac{\partial}{\partial xj}\left(\rho k ui\right) = \frac{\partial}{\partial xj}\left[\left(\mu + \frac{\mu t k}{\sigma k}\right)\frac{\partial}{\partial xj}\right] + Gk + Gb - \rho\varepsilon \text{-} YM + Sk \quad (7)$$

$$\frac{\partial}{\partial t}\left(\rho\varepsilon\right) + \frac{\partial}{\partial xi}\left(\rho\varepsilon ui\right) = \frac{\partial}{\partial xj}\left[\left(\mu + \frac{\mu t}{\sigma\varepsilon}\right)\frac{\partial\varepsilon}{\partial xj}\right] + C1\varepsilon\frac{\varepsilon}{k}\left(Gk + G3\varepsilon Gb\right) - C2\varepsilon\rho\frac{\varepsilon^2}{k} + S\varepsilon \quad (8)$$

In formula (7) and formula (8), $C1\varepsilon = 1.44$, $C2\varepsilon = 1.92$, $\sigma k = 1.0$, $\sigma\varepsilon = 1.3$.

2.3 PISO algorithm

The PISO (pressure implicit with splitting of operators) algorithm was first proposed by Issa in 1986. The calculation process includes an estimate step and two calibration steps. In order to better satisfy the requirement of momentum equation and the continuity equation at the same time, the PISO algorithm has been corrected twice to speed up the convergence rate in each iteration.

3. Numerical simulation of influence of novel refuelling nozzle with rectifier tube

3.1 The establishment of geometric model

In the process of modelling the tank and the refuelling gun, because the tank port is located in the center of the vertical position of the tank, thus the research object has the feature of axial symmetry which could be simplified into a two-dimensional model. The difference between the rectifier type refuelling gun and the ordinary refuelling nozzle mainly lies in rectifier tube. Therefore, the main parts of geometric model are the same. As shown in Fig.1, the bottom length of tank equals 0.6 m, height of fuel tank equals 0.5 m, height of oil tank mouth equals 0.15 m, diameter of oil tank mouth equals 0.078 m, length of refuelling nozzle equals 0.15 m, diameter of refuelling nozzle equals 0.025 m.

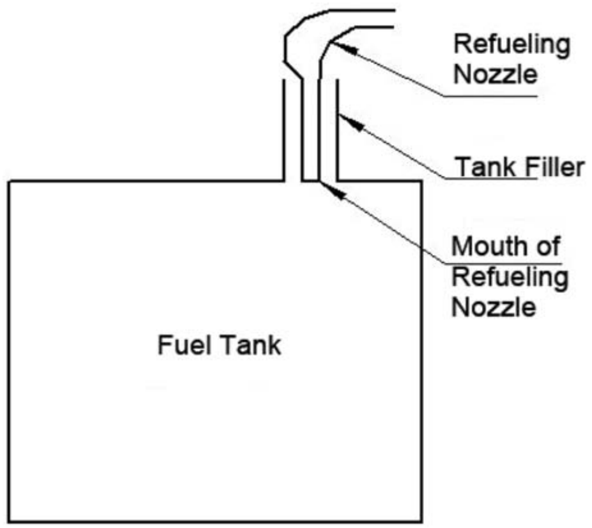


Fig.1. Geometric model

3.2 Grid partition

The grid division for this section mainly focuses on the rectifier part. In order to simplify the structure and reduce the computational complexity, the rectifier is simplified as a structure with nine through holes in radial direction. Rectifier tube length equal 0.03m, diameter of round hole channel equals 0.0015m, spacing distance between holes equals 0.001m. On the basis of the fundamental geometric model, the models for the rectifier type refuelling gun and the ordinary refuelling nozzle are divided by using the triangular and quadrilateral hybrid grid unit form comprehensively, which is shown in Fig.2.

3.3 Boundary conditions

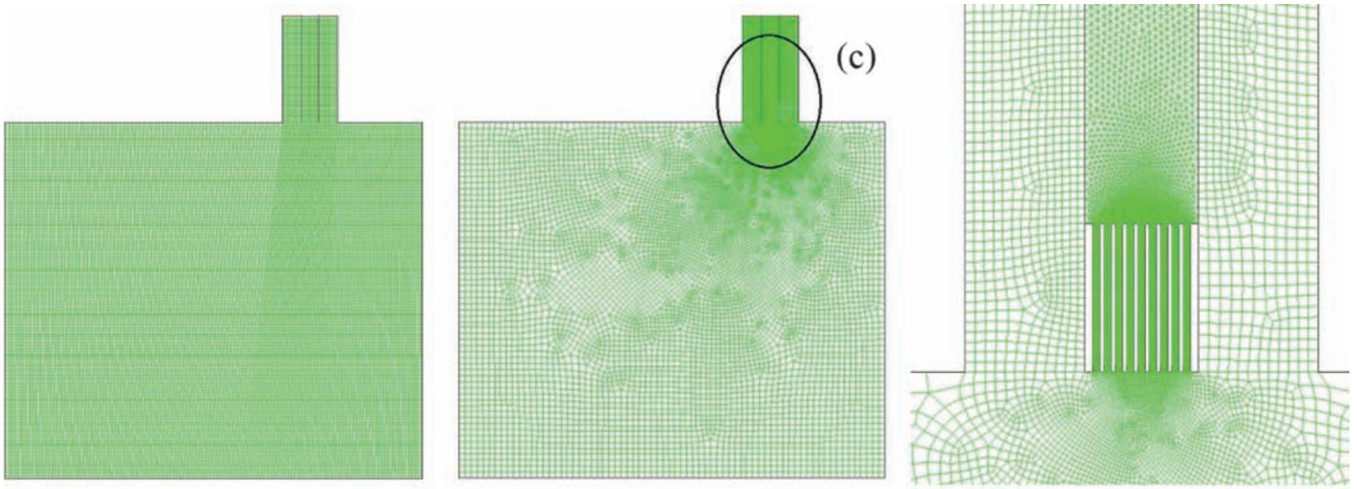
According to the liquid flow inside the tank, the boundary condition selects the velocity inlet and the pressure outlet. The upper end of the fuel gun is regarded as the velocity inlet, and the annular area on the upper end of the tank mouth is regarded as the pressure outlet. Except for the exit and entrance of the tank, the remaining boundaries are set to fixed wall conditions. The calculated operating conditions for this numerical simulations are shown in Table 1.

3.4 Solver and parameter setting

During the solver and parameter settlement, the “standard two-equation” turbulence model is adopted with the single-2d calculation accuracy, and the pressure-based unsteady implicit solver is selected simultaneously. The two-phase fluid materials are set up as air and diesel, and the density of diesel is modified to 830 kg/m^3. In this paper, the gravity effect cannot be ignored, and other operating condition of Fluent solver keeps the default setting. At the entrance of the oil tank, the diesel volume fraction is set as 1, and the diesel speed is set as 2 m/s. At the outlet of the oil tank, the pressure is set as 1 standard atmospheric pressure. What is more, the parameters setting of convergence factor is adjusted as follows: density equals 1, pressure equals 0.3, volume force equals 1, momentum equals 0.2, turbulence kinetic energy equals 0.8, turbulent dissipation rate equals 0.8, and turbulent

Table 1 Computational conditions

Test number	Refuelling gun caliber (mm)	Filling flow (L/min)	Refuelling gun type	Filling angle	Outlet pressure(Pa)
1	25	180	Normal nozzle without rectifier tube	vertical	0
2	25	180	Novel nozzle with rectifier tube	vertical	0



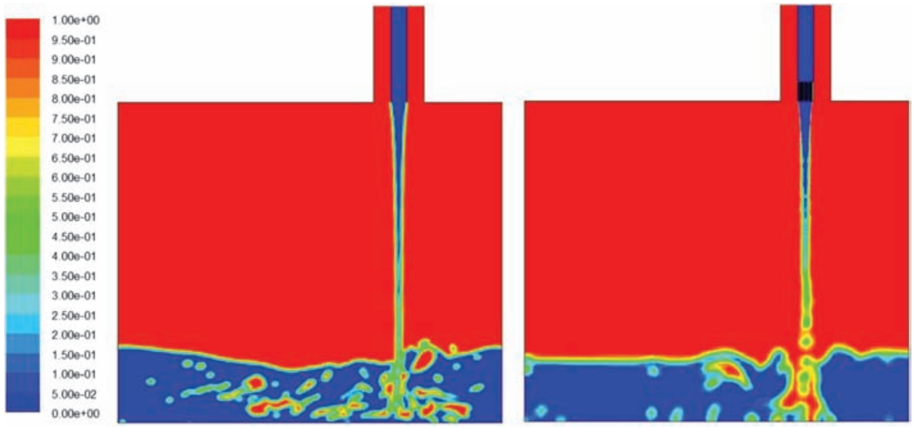
(a) without rectifier tube (b) with rectifier tube (c) local enlarged image of rectifier tube

Fig.2. Mesh grids

viscosity equals 1. “PISO” algorithm is adopted for pressure velocity coupling. In order to accelerate the convergence of numerical calculation, the discretization scheme of momentum equation and turbulent kinetic energy equation is set 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 regarded as the global initial value. The interphase mass and heat transfer are ignored during the calculation. Initialize the iteration calculation sequence and save the data file automatically every 0.1s.

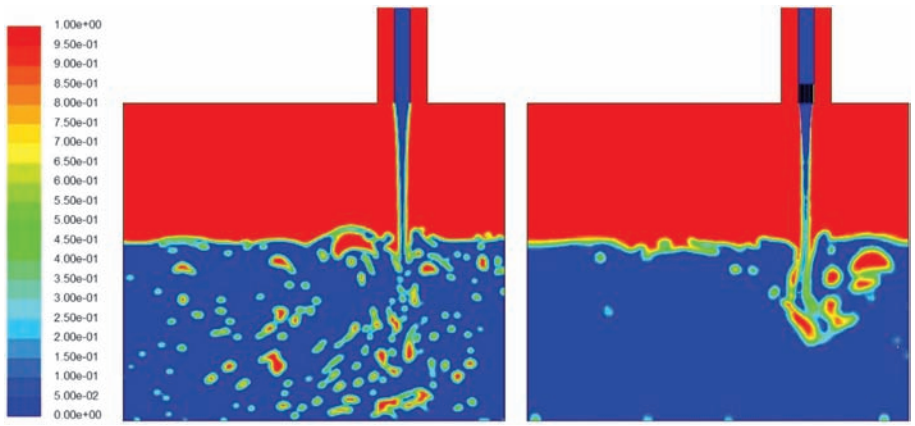
3.5 Results and analysis

In order to analyse the influence of novel refuelling nozzle with rectifier tube in the process of large-flow rapid diesel refuelling, the flow pattern inside the tank under the circumstances of using the rectifier type refuelling gun and the ordinary refuelling nozzles are analyzed at the end of this calculation. And, contours of oil and gas volume fraction when the refuelling time is 3s, 8s and 13.5s which are presented in Figs.3 to 5.



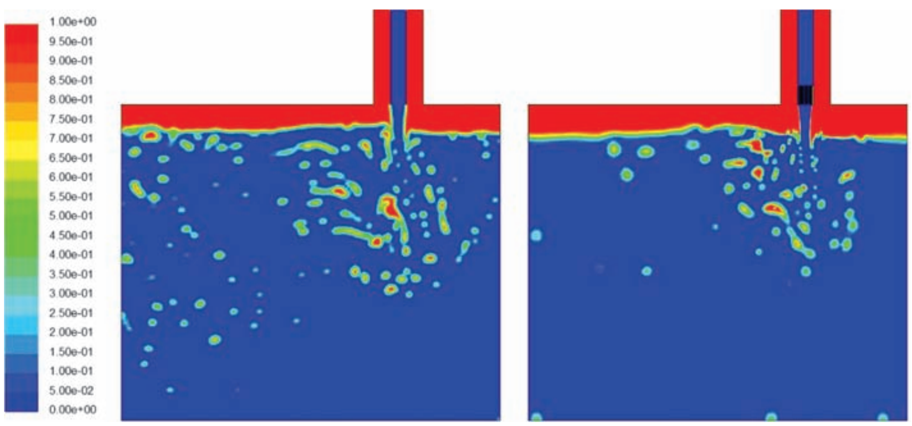
(a) Using regular refuelling nozzle (b) Using novel refuelling nozzle with rectifier tube

Fig.3. Contours of oil and gas volume fraction at t=3s



(a) Using regular refuelling nozzle (b) Using novel refuelling nozzle with rectifier tube

Fig.4. Contours of oil and gas volume fraction at t=8s



(a) Using regular refuelling nozzle (b) Using novel refuelling nozzle with rectifier tube

Fig.5. Contours of oil and gas volume fraction at t=13.5s

In Fig.3, the refuelling time t = 3s is representing the initial stage of refuelling process. It can be seen that, there are more bubbles inside the tank when using the ordinary refuelling nozzle, and the flow field of diesel oil is more in disorder. By contrast, the bubbles inside the tank is relatively few when using the rectifier type refuelling gun, and the internal flow is more stable. In Fig.4, the refuelling time t = 8s is representing the intermediate stage of refuelling process. At this stage, the number of bubbles inside the tank is large and bubbles scattered distribution characteristics are more obviously under the condition of using normal refuelling nozzle. However, the bubbles are much less and concentrated in the intersection of the oil column and the liquid surface when using the novel refuelling gun with rectifier tube. In Fig.5, the refuelling time t = 13.5s representing the closing stage of refuelling process. At this point, the flow field within the tank is similar in both cases. But, the bubbles are much less under the function of rectifier tube. Through the comparative analysis of the flow field during the initial stage, the