A CFD Based Numerical Study of High-speed Refueling

Ye Yang1, Nan Liu2*, Weiguang Shao2, Qiyang He3 and Haojie Yin3

1 North China University of Technology, Beijing, 100144
2 Institute of Systems Engineering, Academy of Military Sciences, PLA, Beijing, 100850
3 Beijing Institute of Technology, Beijing, 100081
*Email: 13426055989@163.com

Abstract. Liquid holdup is an important indicator to evaluate fuel efficiency during high-speed refueling. This paper carried out a numerical study on the fluid flow as refueling a fuel tank based on the volume-of-fluid method in the commercial software FLUENT, and the effects of inlet velocity, injection angle, nozzle diameter, and the shape of the tank on impact phenomena were examined and the corresponding relationships between these factors and the liquid holdup were found by a series of single-variable numerical experiments. The results indicate that: a smaller impact angle (30°-60°), lower inlet velocity (2.5-10 m/s), and larger nozzle diameter (36-40 mm) could result in a higher liquid holdup, and a tank with a square cross-section presents higher and more stable liquid holdup than that of a racetrack-like one. Besides, the effect due to the tank shape is greater than those of the inlet velocity, injection angle, and then the nozzle diameter. This work could be helpful for the design and control of a high-speed refueling equipment.

1. Introduction

High-speed refueling technology has a wide range of needs in military and civilian fields, such as military aircraft, armored vehicles, racing cars, public transport vehicles, and large industrial machinery. The oil is usually involved in a large amount of air during the high-speed refueling process, which reduces the liquid holding rate in the tank and increases the volatilization of the oil, resulting in a reduction in refueling efficiency and an increase in energy loss [1]. Therefore, avoiding the generation of foam and the overflow of oil is of great significance for improving the high-speed refueling efficiency and reducing oil waste.

The issue of gas-liquid two-phase flow is a key scientific problem in the research of high-speed refueling. So far, many researchers and refueling equipment R&D institutions have used computational fluid dynamics to conduct numerical studies on the refueling process. Banerjee et al. [2-3] employed commercial CFD software ANSYS FLUENT to establish a simplified refueling pipe and tank model to study the flow of gasoline and then optimized the refueling pipe structure based on the simulations [4]. This researcher further carried out CFD analysis on the refueling system of a light truck, including the refueling nozzle, refueling pipe, return pipe, and fuel tank, and simulated the refueling process at different flow rates from 4 to 80 L/min, and proposed a liquid surface evaporation model with multiple species, which successfully simulates the vapor emission of the truck and predicts the generation of vapor and air entrainment during the refueling process [5]. Wiesche [6] simulated the evolution of the liquid interface in the pipe during refueling based on the VOF method and the home-made finite difference code and then optimized the geometry of the fuel pipe and the inlet and outlet shapes for an automobile fuel system. Ding et al. [7] used MSC.Dytran software combined with an adaptive multi-Euler domain technology to establish a simulation model for a refueling system including a fuel tank, fuel pipe, and return pipe. Struckmeier et al. [8] combined numerical simulation and experimental
testing and applied convection-diffusion theory to examine the oscillation effect of gas-liquid mass transfer in a tank. Hassanvand et al. [9] simulated the gas-liquid two-phase flow in a refueling system composed of a fuel nozzle, a refueling pipe, a tank, and a return pipe based on FLUENT, and numerically studied the volatilization issue as the gasoline splashing in the tank and then the effect of the factors such as loading speed, temperature and initial oil-air mass fraction on the volatilization, which can be referred to the structure optimization of a refueling system.

In this paper, a gas-liquid two-phase flow simulation model is established for a high-speed refueling system composed of a fuel nozzle and a tank, and the effects of refueling speed, incident angle, and refueling structure on the flow state as well as the liquid holding rate during the refueling process are then investigated. This work can provide a theoretical reference for the design, optimization, and control of a high-speed refueling equipment.

2. Modeling of Gas-liquid Multiphase Flows

2.1. Problem Setup of Refueling

Figure 1 shows the simplified geometry of a fuel tank, where the dimensions of Φ38 mm and Φ58 mm are the inner and outer diameters of the nozzle of the refueling gun, the outlet of the tank is of a diameter of Φ100 mm, and the area closed by ABCDO is the main domain of the fuel tank.

![Figure 1. Simplified geometry of a fuel tank](image)

The estimated Reynolds number \( \text{Re} = \frac{\rho v d}{\mu} = 835 \times 5 \times 0.038 / 0.0042 = 37774 \), indicating a turbulent flow state. The simulations are conducted based on FLUENT, where the \( k-\varepsilon \) Realizable turbulence model is used and the physical parameters, such as density 835 kg/m3, viscosity 0.0042 kg/(m·s), and the surface tension coefficient 0.0265 N/m are applied to the liquid oil. The boundary conditions indicated in Figure 1 are as follows: IJ is taken as the velocity inlet; EF and MN are the pressure outlets; the return phase is set as air; all the walls are set with a no-slip condition.

2.2. Fluid Model Based on VOF Method

The VOF method is applied to capture the two-phase flows of the oil-air system, in which the fluids are regarded as incompressible and immiscible. The transport equation for the volume fraction is as follows:

\[
\frac{\partial f}{\partial t} + \mathbf{u} \cdot \nabla f = 0.
\]

(1)

Where \( f \) is the volume fraction of the oil phase, and then the air phase is \( 1-f \).

Both the oil and air are considered as Newtonian fluids. The \( k-\varepsilon \) Realizable turbulence model in FLUENT is used to calculate the fluid dynamics. The momentum equation and turbulent kinetic ener-
gy equations are given as:

\[
\begin{align*}
\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) &= -\nabla p + \rho \mathbf{g} + \nabla \cdot \mathbf{\tau} + \nabla \cdot \mathbf{R} \\
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho k \mathbf{u}) &= \nabla \cdot \left( \left( \frac{\mu}{\sigma_k} \right) \nabla k \right) + G_k + G_b - \rho \varepsilon - Y_{st} \\
\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \varepsilon \mathbf{u}) &= \nabla \cdot \left( \left( \frac{\mu}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) + \rho C_{\varepsilon} \varepsilon - C_{\varepsilon} \rho \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} + C_{\varepsilon} \frac{\varepsilon}{k} C_{\varepsilon} G_b
\end{align*}
\]

(2)

Where \( \mathbf{\tau} = \mu \left( \nabla \mathbf{u} + \nabla \mathbf{u}^T \right) - \frac{2}{3} (\nabla \cdot \mathbf{u}) \mathbf{I} \) is the average stress tensor; \( \mathbf{R} = \mu_t \left( \nabla \mathbf{u} + \nabla \mathbf{u}^T \right) - \frac{2}{3} (\rho k + \mu_t \nabla \cdot \mathbf{u}) \mathbf{I} \) is the Reynolds stress tensor. In addition, the boundary layers are modeled by the standard boundary layer function in FLUENT.

3. Simulation Results and Discussion

3.1. Evolution of the Liquid Holding Rate

Before we carry out the effect of interested factors, the flow characteristics of a benchmark case are examined, where the diameter of the fuel nozzle of 38mm and the inlet velocity of 10 m/s. A few frames with the volume fraction distribution are shown in Figure 2. The impact of the oil on the free surface causes the surface fluctuation. A big cavity and a series of involved small bubbles are formed at the lower left of the liquid column, and a concave hole is formed at the upper right, washing the tank wall, as shown in Figure 2b. As the refueling proceeds, the oil on the right side of the pit reaches the top of the tank, as shown in Figure 2c.

![Figure 2](image-url)

**Figure 2.** Evolution of the volume fraction distribution at a gun with a diameter of 38mm and a flow rate of 10 m/s

Figure 3 shows the oil holding rate during the refueling process. It can be seen that the oil holding rate drops significantly, and the minimum value can reach about 80%, fluctuating in a range of about 10%.
Figure 3. Evolution of the oil holding rate during the refueling process at a gun with a diameter of 38mm and a flow rate of 10 m/s.

3.2. Effect of Injection Angle
To examine the effect of different injection angles on the impact phenomenon, the angles between the oil and the static liquid surface are taken as 30°, 45°, and 60° for simulations. Their volume fraction distributions are shown in Figure 4. As the injection angle increases, the impact of the refueling oil on the pre-stored oil in the tank becomes stronger, involving more air.

Figure 4. Volume fraction distributions under different injection angles (t=0.09 s)

Figure 5 shows the oil holding rate during the refueling process. It can be seen that the minimum oil holding rate are 91%, 85%, 81% for the injection angles 30°, 45°, 60°, respectively, indicating that reducing the injection angle can be helpful to improve the holding rate of the oil.
3.3. Effect of Inlet Velocity

To examine the effect of different inlet flow velocities on the impact phenomenon, cases with different inlet velocities of 2.5, 5, and 10 m/s and the same injection angle of 45° are simulated. The volume fraction distributions are shown in Figure 6.

Figure 5. Evolutions of the oil holding rate under different injection angles.

Figure 6. Volume fraction distributions under different inlet velocities (t=0.09 s).

Figure 7 shows the evolutions of the oil holding rate during the refueling process with these three inlet-velocity conditions. It can be seen that the oil holding rate under the velocity of 2.5 m/s is much more stable than the others and its minimum rate is about 92%. When the inlet velocities are 5 m/s and 10 m/s, the oil holding rates fluctuate in the range of about 85% to 95% and 80% to 90%, respectively. The results indicate that the averaged oil holding rate is more stable and higher for the case with a lower inlet velocity than that of the case with a higher velocity.
3.4. Effect of Nozzle Diameter

To examine the effect of different nozzle diameters on the impact phenomenon, the cases with a nozzle diameter of 36, 38, and 40 mm and the same injection angle of 45° and inlet velocity of 10 m/s are simulated, respectively. Their volume fraction distributions (not shown) look similar. The evolutions of the oil holding rate during the refueling process are plotted in Figure 8. At the beginning of the refueling process, these plots almost overlap, and then the holding rates fluctuate in a range of smaller than 10% for a while. After that, the fluctuations in these three cases are quite different, where the case with a smaller nozzle diameter shows a larger fluctuation.

![Figure 7](image1.png)

**Figure 7.** Evolutions of the oil holding rate under different inlet velocities.

![Figure 8](image2.png)

**Figure 8.** Evolutions of the oil holding rate under different nozzle diameters.

3.5. Effect of Tank Shape

To examine the effect of different fuel tank shapes on the impact phenomenon, two tanks whose cross-sections are square and racetrack-like, respectively, are simulated. A few frames of volume fraction obtained from these two simulations are shown in Figure 9. For the racetrack-like tank, the provoked oil impacts the top wall first, and then easily involves more air as the provoked oil falls back,
resulting in a reduction of the oil holding rate.

Figure 9. Volume fraction distributions under different tank shapes.

Figure 10 shows the evolutions of the oil holding rate during the refueling process. It can be seen that the fluctuation of the holding rate in the tank of a racetrack-like cross-section is much larger than that in the tank of a square cross-section, and the minimum value is also much lower, 66% and 80% for these two tanks, respectively. The results indicate that the refueling state in the tank of a square cross-section is better than that in the tank of a racetrack-like cross-section.

Figure 10. Evolutions of the oil holding rate in different tanks.

4. Conclusion
This work simulated a high-speed refueling process based on the VOF method and numerically studied the impact phenomenon under different flow rate conditions and structural conditions. The simulation results show that the oil holding rate fluctuates in a range of about 10% and its minimum value is about 80% when the injection angle is 45°, inlet velocity 10 m/s, nozzle diameter 38 mm, and the cross-section of the tank is square. The holding rate is higher as the injection angle is smaller (30°, 45°,
and 60°) and the inlet velocity is lower (2.5, 5, and 10 m/s). The fluctuation of the holding rate is smaller as the nozzle diameter is larger (36, 38, and 40 mm). The holding rate is higher and the refueling is more stable in the tank of a square cross-section than those in the race track-like tank.

5. Acknowledgments
This work was supported by the Scientific Research Foundation of North China University of Technology (NO. 11005136002).

6. References
[1] Yang X, Liu H, Cui H, et al. 2015 Vehicular volatile organic compounds losses due to refueling and diurnal process in China: 2010–2050 Journal of Environmental Sciences 33: 88-96.
[2] Banerjee R, Bai X, Pugh D, et al. 2002 CFD simulations of critical components in fuel filling systems SAE Transactions : 324-340.
[3] Banerjee R, Isaac K M, Oliver L, et al. 2002 Features of automotive gas tank filler pipe two-phase flow: experiments and computational fluid dynamics simulations Journal of Engineering for Gas Turbines & Power 124(2): 412-420.
[4] Banerjee R, Isaac K M 2004 An algorithm to determine the mass transfer rate from a pure liquid surface using the volume of fluid multiphase model International Journal of Engine Research 5(1):23-37.
[5] Banerjee R, Burke C, Gepper D 2007 Experimental and numerical study of gasoline refueling nozzle spray pattern SAE World Congress.
[6] Wiesche S A D. 2004 Simulation of automotive fuel tank filler pipe flows Forschung im Ingenieurwesen 68(3): 139-149.
[7] Ding P, Buijk A J, Van der veen W A 2005 Simulation of fuel tank filling using a multi-material Euler solver with multiple adaptive domains SAE Technical Paper : 2005-01-1915.
[8] Struckmeier D, Tsuru D, Kawauchi S, et al. 2016 Multi-component modeling of evaporation, ignition and combustion processes of heavy residual fuel oil SAE Powertrains Fuels & Lubricants Meeting.
[9] Hassanvand A, Hashemabadi S H, Bayat M 2010 Evaluation of gasoline evaporation during the tank splash loading by CFD techniques International Communications in Heat & Mass Transfer 37(7): 907-913.