A Numerical Study of Oil Cavitation in a Refueling Pipe

Nan Liu¹, Weiguang Shao¹, Ye Yang²*, Haojie Yin³ and Qiyang He³

¹ Institute of Systems Engineering, Academy of Military Sciences, PLA, Beijing, 100850 China
² North China University of Technology, Beijing, 100144 China
³ Beijing Institute of Technology, Beijing, 100081 China
Email: yangye@ncut.edu.cn

Abstract. The issue of cavitation of oil in a pipe is a concerning problem in the design of a high-speed refueling device. This paper established a CFD model for the fluid flow of oil in a refueling pipe based on FLUENT and applied a combined VOF multiphase model and Schnerr-Sauer cavitation model to simulate the cavitation behavior as the oil flows around obstacles in a pipe. The effects of refueling speed, obstacle structure, and pipe diameter on the cavitation were numerically studied using the combined model, and the volume and mass rates of cavitation and their distribution characteristics were obtained. The numerical results show that: the cavitation is strong behind the obstacle, which presents a length of 217 mm, maximum volume rate 53%, and maximum mass rate 0.18% as the refueling speed is 10 m/s; reducing the inlet velocity and increasing the pipe diameter can dramatically weaken the cavitation behavior; adding more obstacles leads the cavitation area reduces first and then keep stable gradually, in which the cavitation phenomenon starts from the first obstacle; placing the obstacles alternatively in the pipe can significantly intensify the cavitation, while the cavitation always starts from the last obstacle.

Keywords: Cavitation, Multiphase flow, Refueling pipe, Numerical simulation

1. Introduction
The phenomenon of cavitation is widespread in the case of high-velocity liquid flowing around solid obstacles. During the refueling process, when the flow rate of the oil is high enough, the pressure will drop sharply after passing a certain solid structure. When the pressure drops below the saturated vapor pressure of the oil at the local temperature, many tiny bubbles will be produced. The bubbles will shrink or even disappear as the pressure increases. The cavitation generated during the refueling process will not only reduce the refueling efficiency but also cause huge waste of resources and environmental pollution. Therefore, studying the cavitation phenomenon in the fueling pipeline is of great significance to the structural design of the fueling pipeline and the control of the fueling process.

In terms of the theoretical model of cavitation phenomenon, Plesset [1] established the Rayleigh-Plesset equation including liquid viscosity and surface tension based on the early free-cavitation Rayleigh equation, aiming at the complete process of spherical bubbles from growth to collapse. Based on the Rayleigh-Plesset equation, Singhal et al. [2] considered the effects of factors, such as incompressible gas, turbulent pulsating pressure, and surface tension, in the liquid on the cavitation phenomenon, and proposed a relatively complex total-cavitation Singhal model. Based on the Kubota [3, 4] and Gerber [5] models, Zwart [6] proposed the Zwart-Gerber-Belamri cavitation model based on the liquid and gas mass fraction transport equations. The cavitation phenomenon in a pipeline will cause the reduction of the fluid holdup rate and serious cavitation corrosion on adjacent
solid walls [7]. Some theoretical and experimental studies on the cavitation phenomena in a pipe and its similar structures have been reported. Andrea Marcon et al. [8] found that as the size of the jet nozzle increases, the cavitation intensity increases significantly. This result was verified by high-speed camera analysis and pitting experiments. Alister Simpson et al. [9] proposed a numerical method of multiphase fluid dynamics model to examine the effects of key parameters, such as orifice plate thickness, orifice sharpness, and wall angle, on cavitation phenomenon. The cavitation phenomenon in pipes is closely related to the fluid flow state and the structure of the pipe, and the degree and distribution of cavitation are very sensitive to the flow and geometric factors. There are few reports on the research on the cavitation formation and the effects of the structural characteristics on cavitation during an oil refueling process.

In this paper, a numerical study of the flow and cavitation of oil in an oil-gun pipe during high-speed refueling is carried out, focusing on the analysis of factors, such as the flow of oil and structural characteristics, on the degree and distribution of cavitation of the oil to find the effects of the relevant factors on the distribution of oil holdup in the refueling pipe. This work will offer a theoretical reference to the design of an oil gun pipe and the control of fueling parameters.

2. Modeling of the Gas-Liquid Flows

2.1. Problem Setup of the Refueling Pipe

According to the local structural characteristics of the fuel gun pipe, the calculation domain of the cavitation model is simplified as shown in figure 1, a two-dimensional axisymmetric geometry for the refueling pipe, where AB is the inlet, BC and FG are the symmetry axes of the pipe, the CDEF area is an obstacle block, GH is the outlet, and AH is the wall. The length of the pipe is 400 mm, the radius is 25 mm, the thickness of the cylindrical obstacle is 10 mm with a radius of 15 mm and a distance to the outlet of 365 mm.

![Figure 1. A simplified geometry of the oil-gun pipe.](image)

The numerical study is carried out by using FLUENT software. The Reynolds number is estimated as $Re = \frac{\rho v d}{\mu} = 835 \times 5 \times 0.05/0.0042 = 49702$, indicating a turbulent flow state, and the $k$-$\varepsilon$ Realizable turbulence model is then used to model the fluid dynamics. The density of the liquid oil is 835 kg/m$^3$, the viscosity is 0.0042 kg/(m·s), the surface tension coefficient is 0.0265 N/m, the saturated vapor pressure is 37100 Pa, and the gaseous density after cavitation is 0.8 kg/m$^3$. The boundary conditions are as follows: AB is the velocity inlet; GH is the pressure outlet ($10^5$ Pa); BC and FG are the symmetry axes; AH, CD, DE, and EF are the non-slip wall surfaces.

2.2. Cavitation Modeling Based on the VOF Method and Schnerr-Sauer Model

To simulate the gas-liquid two-phase flow behavior in the cavitation process, the VOF multiphase flow method is employed. Since a gas-liquid phase transition occurs during the cavitation process, the corresponding mass generation and disappearance must be included in the volume fraction transport equation as follows:
\[
\frac{\partial \alpha_i \rho_i}{\partial t} + \nabla \cdot (\alpha_i \rho_i \mathbf{u}) = R
\]

(1)

where \( \mathbf{u} \) is the velocity field, \( \alpha_i \) and \( \rho_i \) are the volume fraction and density of the \( i \)-th phase, and \( R \) is the source term for the net mass conversion during the cavitation. The Schnerr-Sauer model is applied to calculate the net mass source term of the gas phase as follows:

\[
R = \frac{\rho_v \rho_l}{\rho} \frac{D\alpha_v}{Dt}.
\]

(2)

where \( \rho \) is the combined density, and the subscripts \( v \) and \( l \) indicate the gas phase and the liquid phase, respectively. The vapor volume fraction \( \alpha_v \) has the following relationship with the cavitation bubble number density \( n_b \) and bubble radius \( r_b \):

\[
\alpha_v = \frac{4}{3} \frac{n_b \pi r_b^3}{1 + n_b \frac{4}{3} \pi r_b^3}.
\]

(3)

The gas-phase net mass source term derived from the Schnerr-Sauer model is finally given as follows:

\[
R = \frac{\rho_v \rho_l}{\rho} \alpha_v (1 - \alpha_v) \left( \frac{2}{3} \frac{\sqrt{2 (p_v - p)}}{\rho_l} \right).
\]

(4)

Where \( p \) is the pressure, \( p_v \) is the saturated vapor pressure of the liquid phase.

3. Simulation Results and Discussions

3.1. Cavitation Characteristics in the Refueling Pipe

Taking the inlet flow rate of the refueling pipe as 10 m/s, a quasi-steady-state simulation is computed, obtaining the volume fraction of the oil, static pressure, and velocity distribution as shown in figure 2. When the oil passes the obstacle block, the open area decreases, the flow velocity rises, and the static pressure drops sharply, resulting in cavitation. The length of the cavitation region is about 217 mm. The oil at the right ahead of the obstacle block is strongly cavitated.

![Figure 2](image.png)

Figure 2. The distributions of the volume fraction, static pressure, and velocity of the liquid phase of the oil in the pipe with an inlet flow rate of 10 m/s.

The cavitation volume rate and mass rate in the pipe are shown in figure 3. When the oil flows
around the obstacle block, cavitation occurs rapidly, where the cavitation volume rate reaches about 53% and the mass rate about 0.18% in a short time. The cavitation volume rate slowly drops to 43%, and the mass rate to 0.12%. As the oil pressure rises above the saturated vapor pressure, the cavitation phenomenon no longer continues, and the cavitation rates quickly drop to zero.

![Figure 3](image3.png)

**Figure 3.** The distributions of cavitation volume rate and mass rate along the pipe.

### 3.2. Effect of Flow Rate

To examine the effect of different flow rates on the cavitation phenomenon, the cases with inlet velocities of 9 m/s, 8 m/s, and 7 m/s are respectively simulated and compared with the results obtained at 10 m/s. Their volume fraction distributions are shown in figure 4. When the inlet flow rate is 9 m/s, the length of the cavitation region is about 42 mm, and the oil with a small area on the top of the obstacle block is completely cavitated while the cavitation is very weak and even disappeared elsewhere; when the inlet flow rate is 8 m/s, the cavitation region reduces to with a length of 8 mm and occurs only at the upper left corner of the obstacle. The cavitation disappears as the inlet velocity is 7 m/s. It can be seen from the simulation results that the inlet velocity has a significant impact on the occurrence and severity of cavitation.

![Figure 4](image4.png)

**Figure 4.** The distributions of volume fraction of the liquid oil at different inlet velocities.

Figure 5 shows the plots of the cavitation volume rate and the cavitation mass rate at the flow rates of 8 m/s and 9 m/s. When the flow rate decreases, the cavitation rate decrease significantly. When the inlet flow rate is 9 m/s, the maximum cavitation volume rate is about 30% and the mass rate is about 0.06%. When the flow rate is reduced to 8 m/s, the cavitation volume rate drops below 10% and the mass rate is below 0.01%. When the inlet flow rate is 7m/s, cavitation disappears.
3.3. Effect of Pipe Structure

To examine the effect of different pipe structures on the cavitation phenomenon, the simulations were performed by changing the position and number of obstacles. The obstacle block was placed at the distances of 280 mm and 195 mm to the outlet, respectively. Their volume-fraction distributions are shown in figure 6, where the length of the cavitation regions is 239 mm and 181 mm, respectively. Compared with the benchmark case with a distance of 365 mm to the outlet, it can be seen that as the position of the obstacle block moves from upstream to downstream in the pipe, the length of the cavitation zone first increases and then decreases.

Figure 6. The distributions of volume fraction of the liquid oil under different obstacle positions.

The plots of the volume rate and mass rate of the cavitation are shown in figure 7. Except for the change in the range of the cavitation region, both the volume and mass rates are changing in a similar trend to that with a distance of 365 mm.
Figure 7. The distributions of cavitation volume rate and mass rate in the pipe when the distance between the obstacle and the outlet is (a) 280 mm and (b) 195 mm, respectively.

In addition, the effect of the pipe structure with more obstacles was also examined. The refueling pipe under different number of obstacles is simulated, and the volume fraction distribution obtained is shown in Figure 8. When the number of obstacles is 2, the length of the cavitation region is about 105 mm, and the oil between the two obstacles is partially cavitated; when the number of obstacles is 3, the length of the cavitation region is about 129 mm, the oil between the first and second obstacles is completely cavitated while that between the second and third obstacles is partially cavitated; when the number of obstacles is 4, the length of the cavitation region is about 130 mm, and the oil between the second and third and the third and fourth obstacles is partially cavitated. Compared with the cavitation in the benchmark case with one obstacle where the length of the cavitation region is about 217 mm, adding one obstacle can significantly reduce the length of the cavitation region while the effect of adding more obstacle again is not obvious.

Figure 8. The distributions of volume fraction of the liquid oil under different number of obstacles.

Figure 9 shows the plots of the cavitation volume rate and mass rate under different number of obstacles. When the number of obstacles is 2, the cavitation rate at the first obstacle is lower where the volume rate is about 43% and mass rate about 0.12%; the cavitation rate at the second obstacle increases, and the volume rate is about 50% and mass rate about 0.16%; as the number of obstacle blocks increases to 3, the cavitation rate is the highest at the first obstacle, the volume rate is about 53% and mass rate about 0.18%, while the rates at the second obstacle reduce to about 48% and 0.14%, respectively, and slightly increases at the third obstacle to about 49% and 0.15%; when the number of obstacles is 4, the cavitation rate is the highest at the first obstacle with volume rate 52% and mass rate 0.18%, the cavitation rates at the second and third obstacles gradually reduce and finally increases at the last obstacle.
Figure 9. The distributions of cavitation volume rate and mass rate in the pipe with (a) 2, (b) 3, and (c) 4 obstacles.

The changes in the pipe structure caused by the shape of obstacles were further examined, where the obstacle blocks and obstacle rings are alternately placed in the pipe (shown as rectangular blocks alternately arranged up and down in the axisymmetric computational domain). Figure 10 shows the volume fraction distribution. After adding one obstacle ring downstream of the obstacle block, cavitation disappears after the first obstacle block and appears at the last obstacle ring, forming a jet cavitation area and continuing to the outlet. Adding more obstacle blocks and rings alternately, cavitation always appears at the last obstacle and extends to the outlet of the pipe.

Figure 10. The distributions of volume fraction of the liquid oil under alternately arranged obstacles.
3.4. Effect of Pipe Radius
To examine the effect of the pipe radius on the cavitation phenomenon, the pipe radius was changed and the simulation results are shown in figure 11. When the pipe radius is 30 mm, the length of the cavitation region is only 11 mm; when the radius increases to 35 mm, the cavitation almost disappears. Compared with the results of the benchmark with a radius of 25 mm, it can conclude that increasing the pipe radius will greatly reduce the cavitation.

![Figure 11. The distributions of volume fraction of the liquid oil under different pipe radius.](image)

4. Conclusion
This work simulated and numerically studied the cavitation phenomenon in the refueling pipe by using the Schnerr-Sauer model and a two-phase model based on the VOF method. When the oil flow rate is 10 m/s, severe cavitation occurs at the obstacle, and the length of the cavitation region can be 217 mm. As the flow rate decreases, the cavitation phenomenon gradually weakens, while increasing the pipe radius will greatly weaken the cavitation. When the obstacle moves from upstream to downstream of the pipe, the length of the cavitation region first increases and then decreases. When the number of obstacles is increased, the length of the cavitation area first decreases and then gradually remains stable, and the cavitation phenomenon always appears from the first obstacle. However, when the obstacle blocks and rings are alternately arranged in the pipe, the cavitation phenomenon is intensified and always appears from the last obstacle.

Acknowledgments
This work was supported by the Scientific Research Foundation of North China University of Technology (NO. 110051360002).

References
[1] Pan S S 1990 Leaps and obstacles on cavitation research history Proc. of Cavitation & Multiphase Flow Forum. ASME/CSME, Toronto, Canada.
[2] Singhal A K, Athavale M M, Li H, et al. 2002 Mathematical basis and validation of the full cavitation model Journal of Fluids Engineering 124(3): 617-624.
[3] Kubota A, Kato H, Yamaguchi H, et al. 1989 Unsteady structure measurement of cloud cavitation on a foil section using conditional sampling techniques Journal of Fluids Engineering 111(03): 204-210.
[4] Kubota A, Kato H and Yamaguchi H 1992 A new modeling of cavitating flows: a numerical study of unsteady cavitation on a hydrofoil section Journal of Fluid Mechanics 240: 59-96.
[5] Gerber A G 2002 A CFD model for devices operating under extensive cavitation conditions ASME International Mechanical Engineering Congress and Exposition.
[6] Zwart P J, Gerber A G and Belamri T 2004 A two-phase flow model for predicting cavitation dynamics Proc. of The International Conference on Multiphase Flow, Yokohama, Japan.
[7] Knapp R T, Daily J W and Hammitt F G 1970 Cavitation New York: McGraw-Hill.
[8] Macron A, Melkote S N and Yoda M 2018 Effect of nozzle size scaling in co-flow water cavitation jet peening Journal of manufacturing processes 31: 372-381.
[9] Simpson A and Ranade V V 2018 Modelling of hydrodynamic cavitation with orifice: Influence of different orifice designs Chemical Engineering Research and Design 136: 698-711.