An efficient SPH model for the simulation of water-dropping from airtankers

Xiang-Shan Guan,Peng-Nan Sun*,Xiao-Ting Huang
School of Ocean Engineering and Technology, Sun Yat-sen University
Zhuhai, China
sunpn@mail.sysu.edu.cn

***Abstract*—The simulation of water-dropping of airtankers, which are aircrafts dedicated to firefighting missions, has always been a challenge in Computational Fluid Dynamics (CFD) simulation due to its complex mechanism and vast splashing space. Although the Smoothed Particle Hydrodynamics (SPH) method is well-suited for addressing splashing phenomena, its multiphase flow model often suffers from low computational efficiency in large spatial domains. This paper presents an efficient SPH model that incorporates airflow resistance using a single-phase coupling algorithm between fluid particles and airflows. The information of airflows can be obtained through the interpolation based on background grids. By calculating the airflow resistance of fluid particles based on their windward area and surface normal, the model can accurately simulate the trajectories and splash patterns of SPH particles under the effect of high-speed airflows. Subsequently, the model is applied to simulate the full-scale water-dropping pattern of an airtanker, showcasing its capability to handle large-scale engineering problems in water-dropping simulations.**

I. INTRODUCTION

Airtankers are crucial in fire prevention and are regarded as essential assets for firefighting operations [1], as shown in Figure 1. The distribution and splashing behavior of water droplets under high-speed airflows directly influence firefighting effectiveness. However, the simulation of water-dropping presents significant challenges. The splashing behavior of water in airflows is highly complex, involving various phenomena such as deformation, fragmentation, fusion, and atomization. As a result, the study of water-dropping dynamics has remained a key focus in the field of Computational Fluid Dynamics (CFD) [2].

Fig. 1. The water-dropping operation of airtankers [1].

The Smoothed Particle Hydrodynamics (SPH) method, a well-established meshless Lagrangian method, has gained increasing recognition as an effective numerical tool for CFD simulations, offering a promising approach for modelling splashing phenomena [3]. Currently, multiphase flow SPH models are commonly employed to simulate splashing scenarios, primarily focusing on water jets. However, in large-scale splashing situations, such as water splashing from an aircraft, using multiphase flow particles to fill the extensive space requires significant computational resources. In contrast, adopting a single-phase flow model for liquids and accurately calculating their airflow resistance can significantly reduce the computational load.

In large-scale splashing problems, the primary focus is on the pattern and distribution of the splashing water. In open spaces, the impact of splashing water on airflow is minimal, and small airflow changes do not significantly affect the splashing pattern. As a result, researchers have increasingly opted to use a single-phase flow model to simulate the splashing patterns while considering the influence of airflows [4]. An efficient SPH model that incorporates high-speed airflows based on a single-phase coupling algorithm between fluid particles and airflows is proposed in this paper. The model uses SPH particles to simulate the evolution of the fluid volume, accounting for airflow effects on the fluid surface throughout the splashing process. Combined with free surface recognition and surface normal calculation techniques, the model can accurately compute air resistance based on the shape and surface normal of the splashing water, which can simulate phenomena such as fragmentation, fusion, and atomization of the splashing water under the effects of airflows and surface tension. Additionally, compared to multiphase flow models, the scheme significantly reduces the required number of particles.

II. SPH MODEL

A. *The governing equation of δ-SPH model*

Based on the weakly compressible SPH (WCSPH) model, Antuono et al [5] introduced a density dissipation term and proposed the δ-SPH model, which can improve the computational accuracy of WCSPH models. The δ-SPH model has become a popular and successful SPH algorithm, whose governing equations are detailed below:

$$\begin{cases} \dfrac{D\rho_i}{Dt} = -\rho_i \sum_j \left(\boldsymbol{u}_j - \boldsymbol{u}_i\right) \cdot \nabla_i W_{ij} V_j + \delta h c_0 D_i \,, \\ \dfrac{D\boldsymbol{u}_i}{Dt} = -\dfrac{1}{\rho_i} \sum_j \left(p_i + p_j\right) \nabla_i W_{ij} V_j + \boldsymbol{g} + \dfrac{1}{\rho_i} \boldsymbol{F}_i^{\sigma} + \dfrac{1}{\rho_i} \boldsymbol{F}_i^{V} \,, \\ \dfrac{D\boldsymbol{r}_i}{Dt} = \boldsymbol{u}_i \,, p = c_0^2 \left(\rho - \rho_0\right) \,, \\ D_i = 2 \sum_j \psi_{ji} \left(\boldsymbol{r}_j - \boldsymbol{r}_i\right) \cdot \nabla_i W_{ij} V_j / \left|\boldsymbol{r}_j - \boldsymbol{r}_i\right|^2 \,, \\ \pi_{ij} = \left(\boldsymbol{u}_j - \boldsymbol{u}_i\right) \cdot \left(\boldsymbol{r}_j - \boldsymbol{r}_i\right) / \left|\boldsymbol{r}_j - \boldsymbol{r}_i\right|^2 \,, \\ \psi_{ji} = \left(\rho_j - \rho_i\right) - 0.5 \left[\langle \nabla \rho \rangle_i^L + \langle \nabla \rho \rangle_j^L\right] \cdot \left(\boldsymbol{r}_j - \boldsymbol{r}_i\right) \,. \end{cases} \tag{1}$$

where ρ and ρ_0 represent the density and reference density of particles, respectively. The terms p and V correspond to the pressure and volume of fluid particles. $\boldsymbol{r}$ and $\boldsymbol{u}$ denote the position and velocity of particles, with subscripts i and j indicating the central particle and its neighboring particles. The operator D/Dt refers to the material (or Lagrangian) derivative with respect to time t. The kernel function is denoted by W, and the smoothing length h is defined as 1.25 times the spacing between fluid particles in this study. The vector $\boldsymbol{g}$ represents the gravitational acceleration, while c_0 stands for the speed of sound. The term $\delta h c_0 D_i$ accounts for density diffusion, where δ is typically fixed at 0.1 across various scenarios.

The viscosity term $\boldsymbol{F}_i^V$ and surface tension term $\boldsymbol{F}_i^{\sigma}$ are explained in the [6] and [7].

B. Calculation method for airflow force on fluid particles.

The information of airflows can be imported into the SPH computational domain through background grids, and the velocity of airflows $\boldsymbol{v}_{air}$ where SPH particles are located can be obtained through the interpolation of background nodes. The calculation formula for airflow velocity is $\boldsymbol{v}_{air} = \frac{\sum_j \boldsymbol{v}_j W_{ij}}{\sum_j W_{ij}}$.

The windward area S_r of fluid particles in airflows can be expressed as the projected area in the direction of airflows, which is shown in Figure 2 and can be calculated by the following formula.

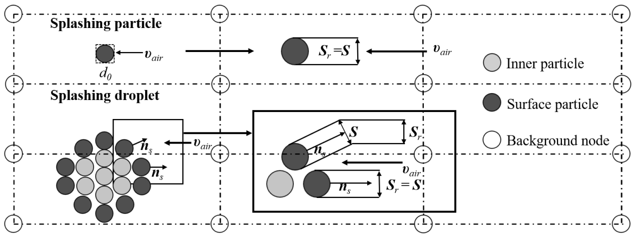

Fig. 2. Schematic diagram for calculating the airflow force on particles.

$$S_r = \begin{cases} S & n < n_\lambda \\ \left|\boldsymbol{n}_g \cdot \boldsymbol{n}_s\right| S & n \geq n_\lambda \,\text{and}\, \boldsymbol{n}_g \cdot \boldsymbol{n}_s < 0 \\ 0 & \text{otherwise} \end{cases} \tag{2}$$

where S represents the surface area, and $\boldsymbol{n}_s$ is the unit normal vector of particle surfaces. $\boldsymbol{n}_g$ refers to the unit vector of relative velocity. The term n denotes the number density, while n_0 is the particle number density at the initial state, when the particle support domain is fully defined. n_λ is the threshold number density, typically set at $0.3n_0$. When the particle number density falls below n_λ, the particle is considered to be exposed to airflows, regardless of its normal vector. The surface area of fluid particles is calculated as follows [7]:

$$S = \begin{cases} \pi r^2 (n_{flat} - n)/n_{flat} + d_0^2 n/n_{flat} & n < n_{flat} \\ d_0^2 (n_0 - n)/(n_0 - n_{flat}) & n_{flat} \leq n < \beta n_0 \\ 0 & n > \beta n_0 \end{cases} \tag{3}$$

where d_0 denotes the distance between fluid particles in the initial state, r denotes the radius of discrete particles, and their transformation relationship is $r = \sqrt[3]{\frac{3}{4\pi}} d_0$. n_{flat} is the particle number density on a flat surface in the initial state, and its value is generally $0.6n_0$.

Droplets that require correction are constructed by discrete particles whose particle number density is less than n_λ. If the spacing between SPH particles is less than d_{smd}, the surface area of droplets constructed by discrete particles can be corrected by $S = k\pi r^2$. Among them, $k = \frac{2r}{d_{smd}}$, and $d_{smd} = \frac{\sum N_i d_i^3}{\sum N_i d_i^2}$. N_i is the number of droplets with a diameter of d_i.

The resistance of fluid particles in the airflow is calculated as follows:

$$D_f = 0.5 C_d \rho_{air} S_r \left|\boldsymbol{u} - \boldsymbol{v}_{air}\right| \left(\boldsymbol{u} - \boldsymbol{v}_{air}\right) \tag{4}$$

where ρ_{air} denotes the air density, which is $1.204\,\mathrm{kg/m^3}$. C_d denotes the drag coefficient of fluid particles in the airflow, which can be calculated as the following formula [4].

$$C_d = \begin{cases} \frac{24}{Re}\left(1 + \frac{Re^{2/3}}{6}\right) & Re \leq 1000 \\ 0.424 & Re > 1000 \end{cases} \tag{5}$$

where Re is the Reynolds number, which can be calculated according to $Re = \frac{2r|\boldsymbol{u} - \boldsymbol{v}_{air}|}{\nu_{air}}$. ν_{air} denotes the kinematic viscosity of air, which is $1.5 \times 10^{-5}\,\mathrm{m^2/s}$.

III. NUMERICAL RESULTS

The airtanker used in this study can carry 12 tons of water for fire-fighting operations. The water collection part of the airtanker consists of four tanks, each of which can hold 3 tons of water. The airtanker remains stationary in the simulation, the water in tanks is released freely, and the airflow moves relative to the airtanker at different velocities. When splashing water falls to the ground, it will be collected by the collector, forming a spreading pattern. The detailed water-dropping numerical model is shown in Figure 3.

d_{smd} is the droplet size that converges and conforms to the actual results. If the spacing between SPH particles is smaller than d_{smd}, the spreading pattern will remain convergent and agree well with the experimental results, as discussed in [7]. To determine the appropriate particle spacing, the operating conditions with particle spacing of 25 mm, 22.5 mm, and 20 mm are selected for convergence analysis. The spreading pattern is adopted as the criterion for convergence, and the spreading patterns at different particle spacings are shown in Figure 3. When the particle spacing is less than 22.5 mm, the spreading pattern remains consistent, and the convergence of calculation results is guaranteed. Due to the powerful computing power of GPUs, the computation time for particle spacing of 20 mm is not much, and the computation time based on Geforce RTX3090 is only 20 hours. Therefore, a particle spacing of 20 mm is selected in the subsequent calculation.

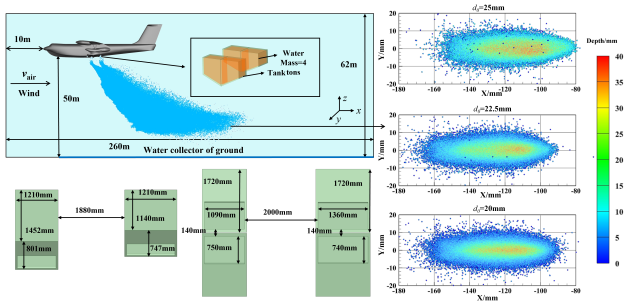

Fig. 3. The water-dropping numerical model of an airtanker and comparison of spreading patterns between different particle spacings.

The velocities of airflows are set to 40 m/s, 55 m/s, 65 m/s, and 75 m/s, respectively. After that, the water-dropping processes of the airtanker at different speeds are simulated, and the simulation results are shown in Figure 4. It can be seen that as the cruising speed of the airtanker increases, the splashing pattern is increasingly influenced by the airflow, and the splashing water gradually disperses, lagging behind the position relative to the airtanker.

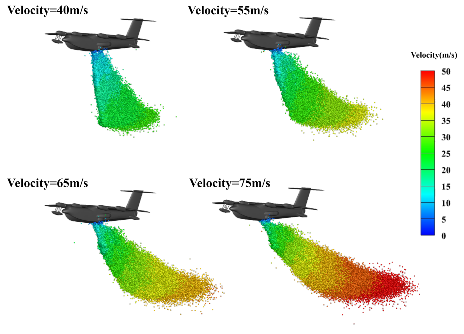

Fig. 4. Comparison of splashing patterns of the airtanker between different velocities (40 m/s, 55 m/s, 65 m/s, and 75 m/s) at Time = 2 s.

Due to the fact that the test flight results of this amphibious aircraft have not yet been made public, accurate test data cannot be obtained. Therefore, only the water-dropping pattern during the test flight can be selected for comparison. The splashing pattern of the airtanker at a speed of 55 m/s is obtained and compared with the splashing morphology during the test flight, as shown in Figure 5. It can be seen that the splashing pattern simulated by SPH models at 2 seconds is consistent with the experimental image, demonstrating the effectiveness of the SPH model considering airflows proposed in this paper in simulating large-scale splashing problems.

IV. CONCLUSIONS

An efficient SPH model considering airflows based on the single-phase coupling algorithm between fluid particles and airflows is proposed in this paper, which can calculate the airflow resistance of fluid particles based on their windward surface and surface normal. The SPH model does not require air particles to construct airflows, which can improve computational efficiency. The entire water-dropping pattern of an airtanker is simulated and consistent with experimental results, proving the feasibility of the algorithm for simulating large-scale water-dropping engineering problems.

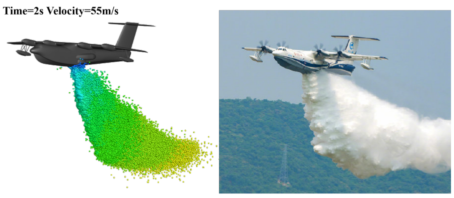

Fig. 5. Comparison of the water-dropping patterns between SPH results and experimental results from a airtanker (provided by AVIC General Huanan Aircraft Industry Co.).

V. ACKNOWLEDGMENT

This research is supported by: the Guangdong Basic and Applied Basic Research Foundation (Grant No. 2024B1515020107); the National Natural Science Foundation of China (Grant No. 52171329); the Fundamental Research Funds for the Central Universities, Sun Yat-sen University (Grant No. 231gbi023); the State Key Laboratory of Disaster Prevention & Mitigation of Explosion & Impact (Grant No. NOLGD-SKL-202201).

REFERENCES

[1] Legendre, Dominique. “Fluid Dynamics of Airtanker Firefighting.” *Annual Review of Fluid Mechanics* 56.1 (2024): 577-603.
[2] Calbrix, Corentin, et al. “Numerical simulation of aerial liquid drops of Canadair CL-415 and Dash-8 airtankers.” *International journal of wildland fire* 32.11 (2023): 1515-1528.
[3] Liu, M. B., and Liu, G. R. “Smoothed particle hydrodynamics (SPH): an overview and recent developments.” *Archives of computational methods in engineering* 17 (2010): 25-76.
[4] Prometech Software, Inc. “Particleworks Theory Manual, Version 7.1.0” *Prometech Software, Inc* (2021): 20-22.
[5] Antuono, Matteo, et al. “Free-surface flows solved by means of SPH schemes with numerical diffusive terms.” *Computer Physics Communications* 181.3 (2010): 532-549.
[6] Vergnaud, A., et al. “C-CSF: Accurate, robust and efficient surface tension and contact angle models for single-phase flows using SPH.” *Computer Methods in Applied Mechanics and Engineering* 389 (2022): 114292.
[7] Guan, Xiang-Shan, et al. “Efficient smoothed particle hydrodynamics modeling of airtanker water dropping: From basic validations to practical applications.” *Physics of Fluids* 36.11 (2024).