33RD CONGRESS
OF THE INTERNATIONAL COUNCIL
OF THE AERONAUTICAL SCIENCES
STOCKHOLM, SWEDEN, 4-9 SEPTEMBER, 2022

A FAST PREDICTION METHOD FOR DROPPING WATER OF FIRE FIGHTING AIRCRAFT BASED ON EXTERNAL BALLISTICS OF PARTICLES

Hongyan Li1[1], Xu Zhao2[1], Zhaoxin Xia3[1]

[1] School of Aeronautics, Northwestern Polytechnical University, Xi'an, Shaanxi, 710072, China1

Abstract

For the process of fire fighting aircraft water-dropping, this paper refers to the cylindrical jet atomization theory under transverse inflow, uses the KH-RT model of secondary jet atomization, the droplet breakdown theory, and the particle exterior ballistics to establish the related theory model, and develops a rapid prediction program for fire fighting aircraft water-dropping. The concentration distribution of the water body on the ground obtained from the fast prediction is in good agreement with the test results, and the shape and size of the water body are in good agreement with the experimental results. This verifies the feasibility of the fast prediction method of the theoretical model developed in this paper, and gives the defects of the model and the correctable methods. In addition, this paper uses CFD numerical simulation to simulate the launching process of L-188 fire fighting aircraft, verifies the accuracy and reliability of the two-phase flow model developed in this paper, and lays a solid foundation for further research

Keywords: Fire Fighting Aircraft, water-dropping, KH-RT Model, Particle External Ballistics, CFD

1. Introduction

Aircraft firefighting is recognized as an advanced firefighting method in the world. Large and medium fixed-wing aircraft have been widely used in the world's major forestry powers to extinguish forest fires. Water-dropping in a suitable position above the fire site to ensure the precise landing of water body to a predetermined position is a key problem to be solved for aircraft dropping water. Due to the complex spatial breakup and movement of water body, the regular rigid body kinematics law cannot be followed. The key engineering problem to be solved in this paper is to develop the trajectory law of water body suitable for firefighting aircraft and to complete the precise launching task with the existing fire control system. The fast forecasting method for the dropping law of water body of fire fighting aircraft can replace the experience of aircraft pilots and play an assistant decision support role for the fire fighting task of fixed-wing fire fighting aircraft.

Wate-dropping schemes play an important role in aircraft fire fighting tasks. When a firefighting aircraft performs firefighting tasks, the aircraft starts dropping water bodies at suitable locations. Water bodies gradually breakup from a large mass of water body under the action of molecular cohesion, surface tension, airflow impact (surface pressure), inertia force and gravity, and the relative speed with airflow decreases gradually, and then drops gradually. As shown in Fig. 1, the breakup and dispersion of water body is a complex and unsteady two-phase flow problem in large space. The breakup and dispersion of water body can be divided into three stages [1]:

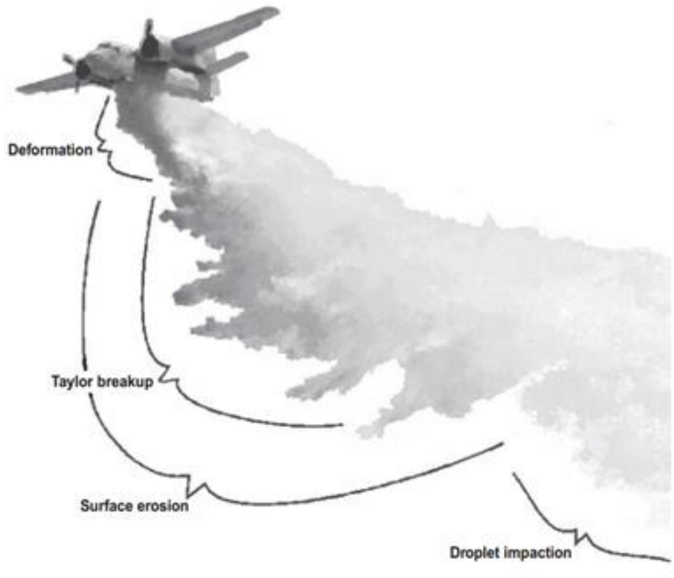

Figure 1 – Breakup and dispersion of water body during aircraft water-dropping.

The simulation method has become the main method to predict the fire fighting effect. The numerical simulation method becomes very important in optimizing the fire fighting operation and detecting the fire fighting effect of new chemical products. There are currently five rapid water assessment programs, PATSIM, FireDrop, FARSITE, RAM, and Aerial Drop Model [2,3]. PATSIM is a model developed by Honey Well, USA, based on experimental data to predict the ground distribution of flame retardants in aircraft, including breakup module. FireDrop is a model developed in the United States to predict surface sediment distribution for helicopter firefighting. The flame growth model FARSITE can provide a planned firefighting scheme based on the actual efficiency given by users. RAM is a EU-funded flame retardant application model. Aerial Drop Model uses the theory of RT and KH instability, and the water drop trajectory is obtained by Lagrange method. The purpose of these programs is to quickly evaluate, optimize the effect of water-dropping, and the simulation of water-dropping process is not detailed enough.

Computational Fluid Dynamics (CFD) simulation software can more accurately simulate the physical process of water- dropping than the rapid assessment of water dispensing program, and has developed rapidly recently. In 2014, the Russian Federal University of Southern Russia and Moscow Aeronautical Research Institute [4] used FlowVision software to simulate Ka-32 helicopter water-dropping. The k-ε model was used for the turbulence model, and liquid gasification was considered. Satoh[5] simulates the water-dropping for PS-1 seaplane, simplifies liquid water into heavy gas with the same density using Nagare Code, and simulates the air-air mixing flow. The results show that the water diffusion morphology is in good agreement. Liang Tianshui [6] of Zhengzhou University also uses FLUENT software heavy gas model and DPM model to simulate the firefighting process of ultra-fine water mist in enclosed space. Takeshi [7] in Japan uses VOF and DPM models based on an understanding of flow phenomena. The data are in good agreement with the wind tunnel experiments. Fig. 2 shows the comparison of water velocity and distribution obtained from wind tunnel experiments with CFD results.

In addition, the research team carried out numerical simulation on nine fire fighting conditions of Jiaolong Aircraft in 2017 using FLUENT software, and obtained water body space variation laws under different flight speeds, heights, temperatures, sidewinds and water feeding modes. The accuracy of water density and water level position was ensured by verification with wind tunnel model water feeding experiments in literature.

Generally speaking, there is little research on the water body trajectory of fire fighting aircraft. Most of the research mainly focuses on simulation test and actual test, and no theoretically complete and applicable water-dropping algorithm has been formed.

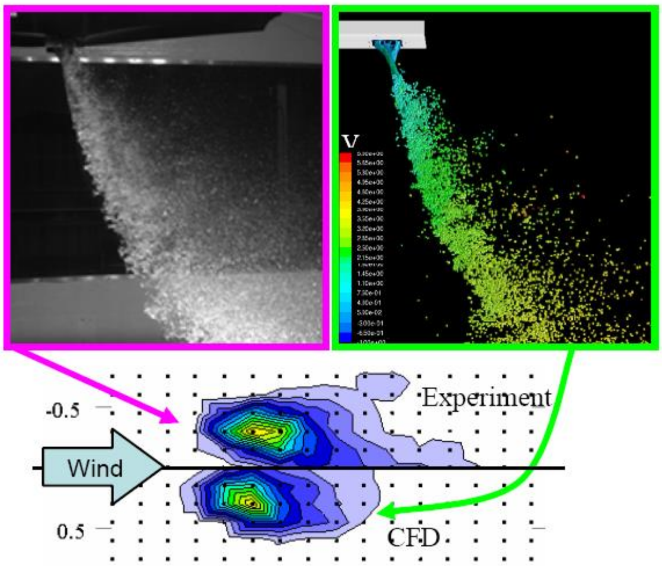

Figure 2 – Comparison of water velocity and falling area obtained from wind tunnel test with CFD results.

For this reason, a fast forecasting program has been developed in this paper, which uses the primary breakup model and the secondary atomization model to simulate the breakup and diffusion process of water body, the particle external ballistics theory to complete the droplet settling process and the empirical formula to deal with the distribution of water body on the ground. In this paper, the feasibility and accuracy of the fast forecasting procedure are verified by the water-dropping test of L-188 fire fighting aircraft. In addition, this paper also uses CFD numerical calculation method to simulate the water-dropping process of L-188 fire fighting aircraft, verify the accuracy of CFD numerical simulation, and lay a solid foundation for further research.

2. Theoretical model

The breakup and diffusion process of the large amount of water released by the aircraft in the air is a multi-stage process. When the water body is thrown out of the aircraft compartment door and subjected to the action of transverse air flow, the liquid jet breaks once, resulting in the formation of ligaments and large droplets, which then form the spray area [8] due to the secondary breakup of the droplets, as shown in Fig.3. Therefore, this paper converts the primary breakup of jet column into the atomization of cylindrical jet under transverse inflow. The KH-RT model of secondary atomization of jet is mainly used to solve this part of the problem. The secondary breakup is simulated by droplet breakup model.

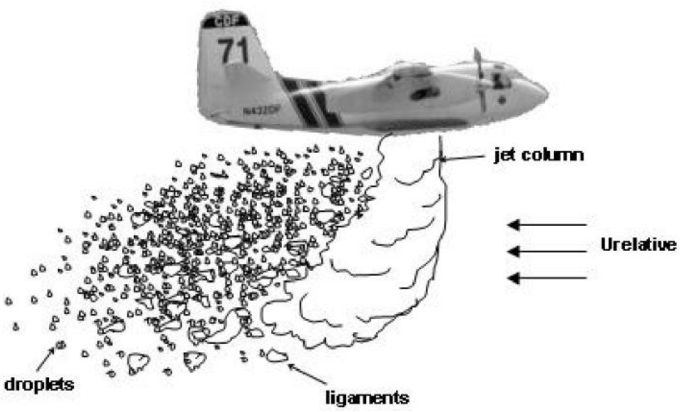

Figure 3 – Diagram of water structure for aircraft dropping

After the water body breakup and diffuses, the droplets make free settling under the action of air drag and gravity. In this part, the motion track of small droplets is calculated by referring to the numerical solution of external ballistics of particles. In this process, only two directions (horizontal and vertical) are considered in this paper. In this paper, the lateral distribution of droplets on the ground is calculated according to empirical formula after droplets fall to the ground.

2.1 Primary breakup

As discussed earlier, researchers typically compare the atomization of a jet column in a transverse flow to the secondary breakup of a droplet. Jet particles are affected by Kelvin-Helmhotz and Rayleigh-Taylor instabilities simultaneously in the airflow. Patterson and Reitz [9] (1998) consider that the breakup of droplets is the result of competition between KH instability and RT instability. Like the KH model, the RT model determines the breakup time and breakup mode of droplets by predicting the wavelength of the fastest growing wave, but the RT model considers that the disturbance is caused by the accelerated instability of the droplet surface rather than the aerodynamic instability. The mixed model in this paper firstly determines whether RT breakup occurs or not, the breakup characteristic time of droplets is longer than RT time scale. When RT breakup does not occur, then judge whether KH breakup occurs. Mixed model KH-RT is shown in Fig. 4 [10]. In KH model, the condition for droplet breakup is that the Weber number of droplets is larger than the critical Weber number and the critical Weber number is set at 12.

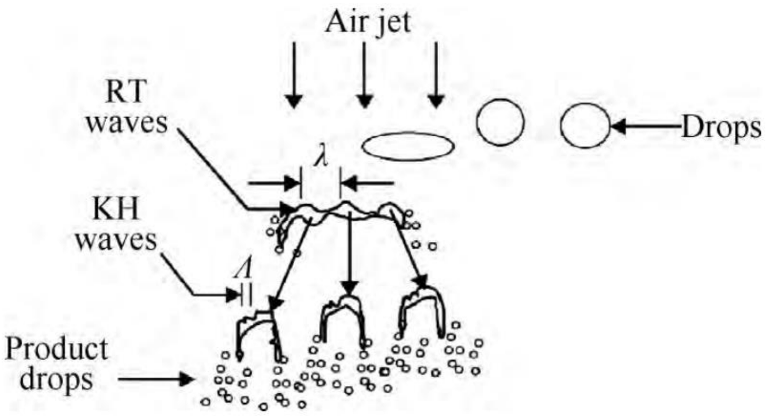

Figure 4 – Schematic diagram of breakup mechanism involving both Rayleigh-Taylor（RT）and Kelvin-Helmholtz（KH） waves

Reitz [11] fits the maximum growth rate_corresponding to the solution of the dispersion equation and the corresponding wavelength into the following expressions(1) and (2).

$$\Omega_{KH} = \frac{0.34 + 0.38 \cdot We_G^{1.5}}{(1 + Oh_L) \cdot (1 + 0.4 \cdot Ta^{0.6})} \sqrt{\frac{\sigma}{\rho_L \cdot R_p^3}} \tag{1}$$

$$\Lambda_{KH} = 9.02 \cdot R_p \frac{\left(1 + 0.45 Oh_L^{0.5}\right) \cdot \left(1 + 0.4 Ta^{0.7}\right)}{\left(1 + 0.87 We_G^{1.67}\right)^{0.6}} \tag{2}$$

The Ω_{KH} is the maximum growth rate of surface wave, Λ_{KH} is the corresponding wavelength, R_p indicates the droplet radius, σ is the surface tension coefficient, and WeG represents the gas Weber number, which represents the ratio of inertia force to surface tension. The physical meaning of the Oh number is the ratio of viscous force to surface tension, $Ta = OhL \times We_G^{0.5}$. Based on the stability theory of liquid jet, small "child droplets" will "peel off" the surface of the "droplet" (also known as the "parent droplet"). Assuming that the size of these small "child droplets" is proportional to the wavelength of the fastest growing surface wave, the radius of the broken "child droplets" can be expressed as follows (3).

$$R_{KH} = B_0 \cdot \Lambda_{KH} \tag{3}$$

B_0 is constant and 0.61 is used. Because the KH unstable wave strips the liquid, the radius of the jet column will be reduced continuously, and its radius change rate and breaking time are as follows (4), (5).

$$\frac{dR_p}{\mathrm{dt}} = -\frac{R_p - R_{\mathrm{KH}}}{\tau_{\mathrm{KH}}} \tag{4}$$

$$\tau_{KH} = \frac{3.726 * B_1 * R_p}{\Lambda_{KH} * \Omega_{KH}} \tag{5}$$

B_1 is usually taken from 1.73 to 40. The B_1 value in the model is defined as 10. At the same time as the KH unstable wave is generated, the RT wave is also generated on the surface of the liquid mass. This unstable wave occurs at the interface between two fluids of different densities in the gravitational field, the interface between liquid and air. The radius of the droplets separated by the RT unstable wave is shown in Formula 6.

$$R_{RT} = \frac{\pi * C_{RT}}{K_{RT}} \tag{6}$$

In this expression, $C_{RT}$is a parameter which can be adjusted by measuring result and is related to the condition of aircraft door,0.3 is taken. Wave number K_{RT} of RT wave as in Formula 7.

$$K_{RT} = [-\frac{(g+a) * (\rho_L - \rho_G)}{3\sigma})\ ]^{0.5} \tag{7}$$

Where g and a are the accelerations of gravity and drag in the direction of droplet flight, respectively. Footers, except for the first page, contain the page number, centered.

2.2 Secondary breakup

The droplets on the surface of the liquid jet column after primary breakup will undergo secondary breakup. The characteristic time scale (t*) of secondary breakup is given by formula 8 [12].

$$t^* = \frac{D_{C0}}{U_R}\sqrt{\frac{\rho_L}{\rho_G}} \tag{8}$$

Where U_R is the relative velocity between the droplet and the gas, D_{C0} is the initial diameter of the droplet after stripping from the surface of the liquid . During secondary breakup, the droplets will deform into discs, increasing the front diameter and drag coefficient. At the end of the deformation period, the shape of the droplets remains unchanged and the droplets continue to move until the breakup time is reached. Secondary breakup occurs through one of two mechanisms: bag breakup with Weber number less than 100 and shear breakup with Weber number greater than 100. During model building, it is found that the droplets are mainly shear breakup with Weber number greater than 100. Therefore, shear breakup are detailed in this paper.

When the Weber number is greater than 100, the following methods are used to determine the size and velocity of the droplets generated by shear breakup. At each time step, the droplet mass resulting from shear breakup is calculated as follows:

$$M_{strip} = 0.42 * \Delta t * \frac{M_c}{t^*} * exp[\,0.8 * (\frac{t_b}{t^*} - 3.5)^2] \tag{9}$$

Where $M_c$is the droplet mass, t_b is the total breakup time.

$$M_c = \frac{\pi}{6} * \rho_L * D_c \tag{10}$$

The droplet size distribution agrees with the Rosin-Rammler expression most widely used in spray studies, which is defined as [13]:

$$F(D_{cc}) = 1 - exp[-0.693 * (\sqrt{\frac{D_c}{MMD}} - 1)^q] \tag{11}$$

Typical values for the dispersion parameter q range from 1.5 to 4 and are defined here as droplet sizes of 3.5. MMD is given as a function of time in shear breakup, detailed reference [14].

2.3 Falling model

In the whole process after the water body is launched by the aircraft, the water body and droplets move under the action of air drag and their own gravity. The trajectory of the water body and droplets is solved in combination with the particle exterior ballistic model. In this part, we make the following assumptions:

a) The water body moves as a whole in the aircraft compartment, ignoring the effect of the opening of the compartment door on the water movement; Ignoring the influence of aircraft shape structure, the jet near the aircraft has no whirlpools and uniform speed.

b) Assuming that the speed at which the water body is dropped out of the hatch is constant, the flight speed of the aircraft remains constant during the release process; After the water body leaves the compartment door, the jet cross-section keeps its shape unchanged during its movement.

c) The velocity of each jet and droplet is the same after the water body is broken.

d) The air density is constant, and the air density is taken as 1.29 kg /m3.

e) The droplet motion after breaking does not interfere with each other and the average distance between adjacent droplets is sufficient to make the drag coefficient close to that of a single droplet. The deformation of broken droplets is not discussed in this paper.

After the water body leaves the aircraft hatch, it continues to move under the action of air drag and its own gravity. The established motion model is shown in formula (12):

$$\begin{cases} m\dfrac{\mathrm{d}v_x}{\mathrm{d}t} = -F\cos\ \beta \\ m\dfrac{\mathrm{d}v_y}{\mathrm{d}t} = -F\sin\ \beta - mg \\ v_x = \dfrac{\mathrm{d}x}{\mathrm{d}t} \\ v_y = \dfrac{\mathrm{d}y}{\mathrm{d}t} \\ \cos\ \beta = \dfrac{v_x}{v} \\ \sin\ \beta = \dfrac{v_y}{v} \end{cases} \tag{12}$$

Where: v_x and v_y are the horizontal and vertical velocity components of the water body respectively; T is the movement time; x and y is the horizontal and vertical displacement of the water body respectively; F is the air drag received by the water body when it moves in space, which is opposite to its moving speed v; β Is the angle between the velocity direction and the horizontal direction when the water body moves.

According to the external ballistics theory [15], the air drag calculation formula is as follows in formula (13):

$$F = \frac{1}{2}\rho_k v^2 A(x) C_d \tag{13}$$

Where: ρ_k is the density of air; V is the velocity of the water body; $A(x)$ is the cross-sectional area change function, $A(x)$ will change with the breakup and diffusion, A_0 is the cross-sectional area of the water body when it leaves the hatch; CD is the air drag coefficient. In order to solve the motion of water body the initial conditions and boundary conditions of its differential equations must be determined. Initial conditions of differential equations: when t = 0, x = 0, y is the height of the aircraft from the ground, and the horizontal initial speed v_x and vertical initial speed v_y are the flight speed of the aircraft and the downward water-dropping speed of the aircraft respectively.

By using the above calculation model, the water body movement of firefighting aircraft during water-dropping test is analyzed and calculated, and the change of speed and displacement of water body with time is obtained. The range and density of the scattered distribution of the water body in the horizontal direction are thus determined.

2.4 Water distribution on the ground

The distribution of water body landing on the ground after the breakup and diffusion has certain laws with the speed V of the aircraft, the time t of water-dropping the size of the water-dropping hatch and the speed of water bodies leaving the hatch [16]. The meaning of the length and width of the water body landing distribution is shown in Fig. 5. Equation (14) (15) can calculate the length and width of the landing distribution pattern.

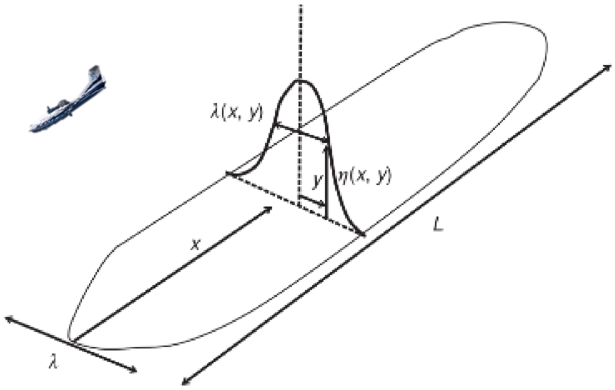

Figure 5 – Schematic diagram of water body landing distribution

Some formulas are as follows：

$$L = V \cdot t \ + 2\lambda \tag{14}$$

$$\lambda \approx d_S f_2 q^{1/5} \tag{15}$$

Where L is the length of ground distribution of water body, λ is the width, d_s=$S^{1/2}$, s is the hatch area, f_2 and q are the correlation functions.

As shown in Figure 6, the average liquid distribution follows a Gaussian distribution in the form of: η_{max} is the maximum coverage level obtained along the centerline, and the expression (16) is as follows:, Φ It is defined as the volume proportion of liquid falling on the ground, usually between 0.55 and 1, λ_0 is defined as standard deviation, standard deviation λ_0 and width of ground water body λ The relevant formula is as follows: formula 17, ground coverage level of water body $\bar{\eta}(y)$ Distribution along the transverse direction is shown in formula (18)

$$\eta_{max} \approx 4\sqrt{\frac{ln(2)}{\pi}}\frac{\Phi V}{(L-2\lambda)\lambda} \tag{16}$$

$$\lambda \approx 2(2ln\,(2))^{1/2}\lambda_0 \tag{17}$$

$$\bar{\eta}(y) = \eta_{max} exp\,[-y^2/2\lambda_0^2] \tag{18}$$

By considering the test results obtained under different working conditions, the accuracy of the water cover level distribution formula has been confirmed.

3. Theoretical model verification

The wildland fire chemical systems (WFCS) project tested the water-dropping experiments of various fixed wing aircraft and rotary wing aircraft to determine the parameters of the best coverage under various fire fighting agents and fire conditions. In 2000, the WFCS project team adopted Electra L-

188 firefighting aircraft to conduct aircraft water dropping experiment [17].

Electra L-188 (similar to Orion p-3a) is a type I aerial tanker for firefighting. Electra L-188 is equipped with a rads II constant current tank that can hold 3000 gallons of fighting agent. The flow rate of releasing fighting agent is controlled by changing the angle of hatch opening. The discharge of fighting agent depends on the opening time of the hatch. There are two different materials for fire fighting agents: water and gum thickening flame retardant. See Fig. 6 for the water body breakup and diffusion form of Electra L-188 fire fighting aircraft. The operating conditions of water injection test of L-188 firefighting aircraft are shown in Table 1 below.

Figure 5 – Water body breakup and diffusion morphology of Lectra L-188 firefighting aircraft.

Table 1 – Lectra L-188 fire fighting aircraft water dropping test conditions.

Volume	Door area	Water dropping velocity	Time	Flight velocity	Height
$3.7m^3$	$2.5m^2$	0.92m/s	1.6s	234km/h	50m

Based on the second part of this paper, this paper develops the calculation program of the relevant theoretical model, which can quickly obtain the distribution of the landing water body of the firefighting aircraft. Input the working condition parameters of the above tests into the calculation program, quickly calculate the ground concentration distribution of the water body, and compare the test results, as shown in Fig. 7 and Fig. 8.

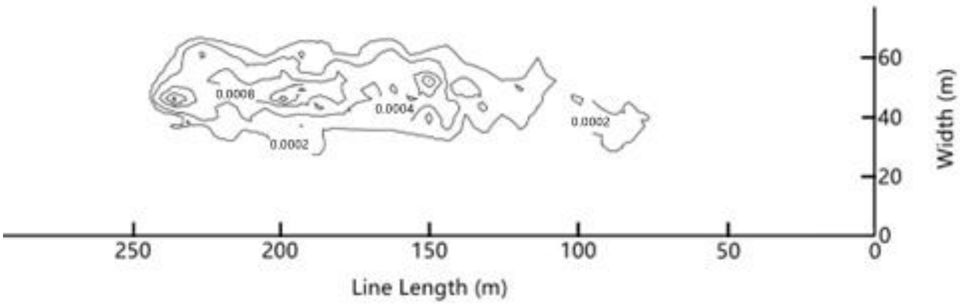

Figure 7 –Ground distribution of water body from L-188 aircraft water-dropping test

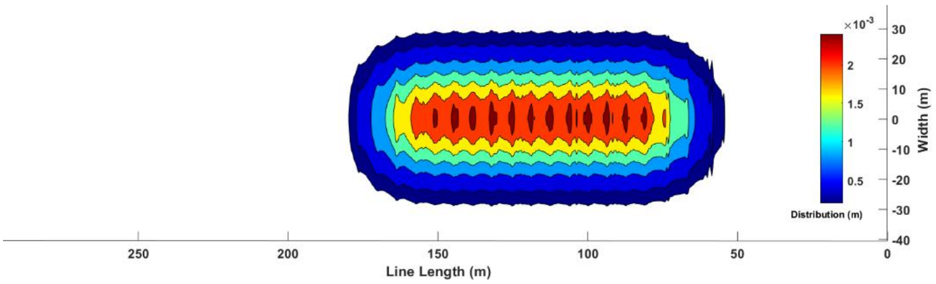

Figure 8 – Prediction of ground distribution of water body from L-188 aircraft water dropping theory

Fig. 7 and Fig. 8 show the ground distribution of the water body landing from the experimental and theoretical predictions. Compared with the test results, the theoretical predicted ground distribution length of water body is 125 m, which is 167 m shorter than the experimental value, with an error of 25%. The theoretical predicted ground distribution width of water body is 56m, which is 38m wider than the experimental value. The initial and final water level distributions of the water body are closer to the aircraft, 55m and 182m respectively, and the comparison test values are 77m and 245m, with errors of 28% and 26% respectively. The concentration distribution of water body on the ground is in good agreement with the test results, and the shape and value are in good agreement with the test results.

The reason why the length of water body distribution is shorter is that in the construction of settlement model, the collision, fusion and deformation of droplet movement are neglected in this paper. Therefore, the range of droplet diameter in the model is smaller than that in actual test, resulting in a shorter length of water droplet distribution. The theoretical predicted ground distribution width of water body is calculated based on length. In addition, the evaporation effect of droplets in the movement process is neglected in this paper, so the width also has a large error. The reason why the water body falling at the beginning and the end is closer to the aircraft is that the water body will undergo three processes: deformation, breakup and diffusion after being launched. In the theoretical model, the deformation process is neglected in this paper, which leads to the water body breakup too fast, leading to the early conversion of large droplets into small droplets, which will decelerate sharply in the air, resulting in a short horizontal distance of droplet movement.

Although there are large errors in the results predicted by the theoretical model, the errors of the results of the theoretical model are within an acceptable range. In addition, the results predicted by the theoretical model are similar to the test results, similar in shape and close to the concentration range of ground distribution. This verifies the feasibility of the theoretical model fast forecasting method developed in this paper, and gives the defects and correction methods of the model, which provides basis and reference value for the further research and improvement of the follow-up theoretical model.

4. Numerical simulation model

In addition to the program of fast forecasting of theoretical model developed above, this paper also builds a numerical calculation model with L188 fire fighting aircraft as the main research object in combination with the existing research results of this subject team. In this paper, unsteady simulation, Euler-Euler model, sliding grid and UDF compiling technology are used to simulate the water-dropping process of fire fighting aircraft. The comparison between the calculated and experimental landing distributions is shown in Fig. 9 and Fig. 10.

Compared with the test results, the ground distribution length of water body obtained by numerical simulation is 176m, which is 167m longer than the experimental value, with an error of 5%. The ground distribution width of the water body in the numerical simulation is 29m, which is narrower than the experimental value of 38m.The initial and end positions of water body ground distribution are similar to the test results, 82 m and 258 m respectively, and the comparison test values are 77 m and 245 m, with errors of 6% and 5% respectively.

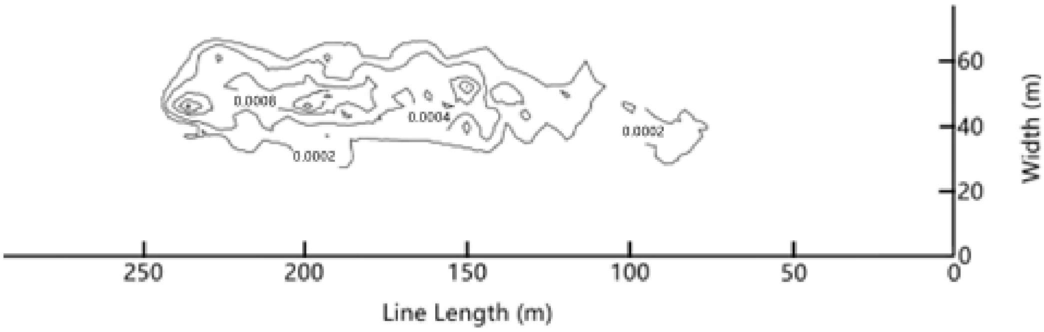

Figure 9 – Ground distribution of water body from L-188 aircraft water-dropping test

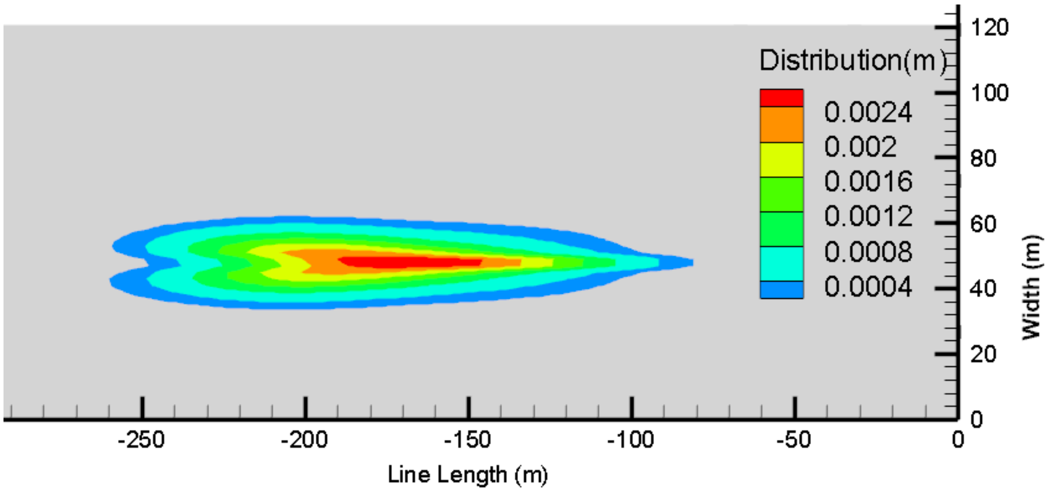

Figure 10 – Ground distribution of water body from CFD simulation of L-188 aircraft

The concentration distribution of water body on the ground is basically consistent with the test results, which verifies the accuracy and reliability of the two-phase flow model developed in this paper and lays a solid foundation for further research. However, the numerical calculation of fire fighting aircraft launching requires a lot of time, manpower and material resources, compared with the fast prediction program of the theoretical model developed in this paper, the calculation can be completed in a few minutes. In general, the fast prediction efficiency of the theoretical model is higher, the accuracy of the theoretical model needs to be improved, and the CFD numerical calculation has higher reliability.

5. Conclusions

The process of breakup and diffusion of large amounts of water released by aircraft in the air is a multi-stage and complex process. The theoretical model developed in this paper refers to the theory of cylindrical jet atomization under transverse inflow. The KH-RT model of secondary atomization of jet is mainly used to solve the problem of primary breakup of jet column and the droplet breakup model is used to simulate and calculate the secondary breakup of water body. The trajectory of water body and small droplets can be calculated by referring to the numerical solution of external ballistics of particles. After droplets fall to the ground, the distribution of droplets on the ground can be calculated by referring to the empirical formula.

The accuracy of the theoretical model developed in this paper is verified by L-188 firefighting aircraft water-dropping test. It is calculated that the concentration distribution of water body on the ground is in good agreement with the test results, and the shape and value are in good agreement with the test results. Although there are large errors in the results predicted by the theoretical model, the errors of the results of the theoretical model are within an acceptable range. This verifies the feasibility of the theoretical model fast forecasting method developed in this paper, and gives the defects of the model and the correction method.

In this paper, CFD numerical simulation is also used to simulate the water-dropping process of L-188 firefighting aircraft. The calculated concentration distribution of water body on the ground is basically consistent with the test results, which verifies the accuracy and reliability of the two-phase flow model developed in this paper and lays a solid foundation for further research.

6. Contact Author Email Address

Xu Zhao, Associate Professor, School of Aeronautics: xuzhao@nwpu.edu.cn

7. Copyright Statement

References

[1] Ann Suter. Drop Testing Airtankers—A Discussion of the Cup-and-Grid Method. USDA Forest Service，Technology & Development Program, 2000.

[2] J. H. Amorim. Numerical Modelling of the Aerial Drop of Fire Fighting Agents by Fixed-wing Aircraft. Part I: Model Development. International Journal of Wildland Fire. 20: 384–393, 2011.

[3] J. H. Amorim. Numerical Modelling of the Aerial Drop of Fire Fighting Agents by Fixed-wing Aircraft. Part II: Model Validation, International Journal of Wildland Fire, 20: 394–406, 2011.

[4] I.V. Borisov, A.V, Tsipenko. Computing Simulation for Fire-fighting Helicopter Operation Analysis, 29th ICAS，2014.

[5] Kohyu Satoh, Kunio Kuwahara，K.T. Yang. A Numerical Study of Water Dump in Aerial Fire Fighting, Proceedings of IMECE04, ASME International Mechanical Engineering Congress and Exposition, November 13-20, Anaheim, California, USA, 2004.

[6] Liang Tian-shui et al. Experimental and Numerical Study of Fire Suppression Performance of Ultral-Fine Water Mist in a Confined Space, Proceeding Engineering 52: 208 – 213, 2013.

[7] Takeshi Ito, et.al. Evaluation of Mission Performance and Safety for Fire-fighting Amphibian. 28th ICAS, 2012.

[8] Wu P.K., Kirkendall K.A., Fuller R.P. and Nejad A.S., 1997. Breakup processes of liquid jets in subsonic crossflows. Journal of Propulsion and Power 13, 64-73.

[9] Patterson M A，Reitz R D. Modeling the Effects of Fuel Spray .Characteristics on Diesel Engine Combustion and Emission［R］. Detroit: SAE Technical Paper 980131，1998.

[10] Kenneth K Kuo. Fundamentals of Turbulent and Multi‐ phase Combustion［M］. New Jersey: Published by John Wiley & Sons，Inc.，Hoboken，Published Simultaneously in Canada，2012

[11] Kuo KK. Recent Advances in Spray Combustion: Spray Atomization and Drop Burning Phenomena［C］. Wash‐ington: Progress in Astronautics and Aeronautics，1996.

[12] Ranger A.A. and Nicholls J.A., 1969. The aerodynamic shattering of liquid drops. AIAA Journal 7(2), 285-290.

[13] 2000 Liu H., 2000. Science and Engineering of Droplets: Fundamentals and Applications. William Andrew Inc. 225 pp.

[14] Chou W.H., Hsiang L-P. and Faeth G.M., 1997. Temporal properties of secondary drop breakup in the shear breakup regime. International Journal of Multiphase Flow 23, 651-669.

[15] HAN Z.P., Rocket exterior ballistics ［M］． Beijing: Beijing Institute of Technology Press，2014: 9—11．(in Chinese).

[16] Legendre. Air tanker drop patterns，International Journal of Wild land Fire 2014 Vol.23 No.2 P272-280 1049-8001.

[17] Paul S, Cammie J. Ground Pattern Performance of the Airspray Electra L-188 with Aero Union Constant Flow Tank. 0057-2851-MTDC，2000，5100/5700.