NUMERICAL SIMULATION OF THE AERIAL DROP OF WATER FOR FIXED-WING AIRTANKERS

X. Zhao*, P. Zhou*, X. Yan, Y. Weng ***, X-L. Yang ******
*** Northwestern Polytechnical University, Xi'an, Shaanxi, 710072, China**
**** No. 38 Research Institute of CETC, Hefei, Anhui, 230088, China**
*****Xi'an Jiaotong University, Xi'an, Shaanxi, 710048, China**
******Chinese Flight Test Establishment, Xi'an, Shaanxi, 710089, China**

Keywords: *unsteady flow, N-S equations, water dump, multiphase flow, airtankers*

Abstract

- *To evaluate the effectiveness of water dropping by fixed-wing airtanker, CFD simulation based on Eulerian multiphase flow model is employed to simulate the dumping process. First, the numerical method is validated by wind tunnel test of airtanker. Results show a satisfied agreement in water precipitation and transient local velocity distribution with test data. Secondly, the full scale water dump model of AG-600 amphibian is developed using sliding mesh technique. Working conditions such as drop height, flight speed, cross wind velocity, atmosphere temperature, released mode and volume are tested. The precipitation area, shape, location and density distribution are compared and analyzed. Compared to the real-scale drop tests of comparable aircraft, the coverage area of water is reasonable, satisfying the designed requirement. Finally, the layout of the water cups in the measurement are inferred from the CFD results. The research of this paper helps to assess the water dumping performance of airtankers and to optimize working conditions.*

1 Introduction

An average of 15.8 million gallons of fire retardant has been used in firefighting each year. Most of this retardant is released from the air [1]. Nowadays, fixed-wing airtankers become a popular firefighting type in world. The characteristics of the water spread pattern (length, width, and coverage level) are influenced by the height and speed of the aircraft, the flow rate and volume of the dumped fluid, the rheological properties of the fluid, and the meteorological conditions [2].

Compared to the laborious dropping water over a tested open cups in arranged grid, numerical method is more economic and efficient to predict water dump effect. To accomplish the simulation, the complicated dump progress needs to be fully understood. It involves aerodynamics, two-phase flow, water deformation and breakup, thermodynamics, phase exchange and even chemical reaction. So far, CFD technique is regarded as the most accurate method to represent the strong interaction between water and air. The review of numerical research [3, 4] shows that software FlowVision, Nagare Code and FLUENT have been employed to discretize the N-S equation so far.

In order to simulate water dump, multiphase flow submodel such as VOF (volume of fluid), Lagrangian method, DPM (discrete phase model) [5] have been applied. The selection of submodel is cautious and realistic. According to water drop flight test, water goes through three stages: deformation of jet flow, surface erosion of the jet flow and droplet impaction [1]. Correspondingly, the space occupancy of the water starts from dense regime, via intermediate regime and finally to dilute regime. Each submodel is based on assumptions, which is only suitable to a certain range of space occupancy problem and no one can cover the whole range. The nominal size of the water can be supposed to start from 1m near the released door and finally decrease until 5mm due to the atomization. To simulate the transient scale-span multiphase problem in large space, a robust multiphase submodel with satisfied accuracy is very important.

This paper employs the Euler multiphase model combined with sliding mesh technique to simulate dumping progress, leaving thermodynamics, phase exchange and chemical reaction to be ignored. CFD results are validated via wind tunnel test of an airtanker, then the full scale AG-600 fire-fighting amphibian (Fig. 1) model is developed to evaluate the dumping effect at various working conditions, to assist the preliminary design of the dropping test.

Fig. 1. The AG-600 amphibian

2 Water Dropping Validation of a Wind Tunnel Model

Ito et al. [6] carried out a test of 1/8 and 1/16 scale CL-415 amphibian in 6.5m×5.5m wind tunnel. The ground is fixed to measure water spread patterns. Working conditions are shown in table 1.

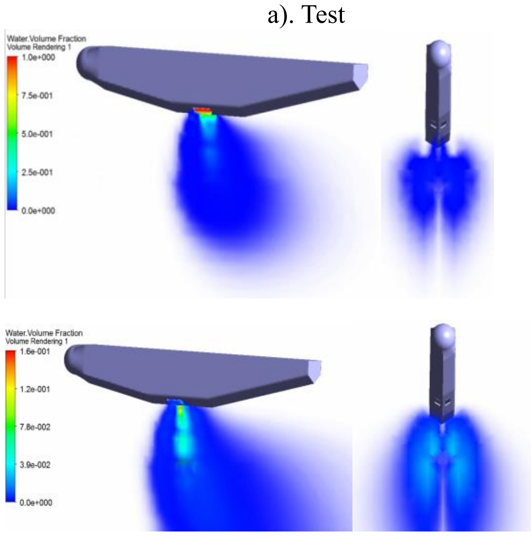
Table 1. Working conditions of the wind tunnel models

Scale	1/8	1/16
Speed(m/s)	15.9	11.3
Height(m)	3.75	1.88
Tank Vol. (liter)	34	4.2
Door (mm)	240×80	120×40
Release duration (s)	0.7	0.5

To build the CFD model, software FLUENT was employed. The Eulerian adiabatic two-phase flow model was developed considering surface tension, gravity, drag and lift between two phases. In the Eulerian multi-phase model, Eulerian parameters was set as multi-fluid VOF model, formulation of volume fraction parameters set as explicit, interface modeling type set as dispersed. Realizable k-ε turbulent model was employed with standard wall function. The diameter of the second phase was set 0.01m. Surface tension of the water is 0.0704N/m. The drag coefficient modification is 0.3 via abundant model tests. Since the airtanker is fixed, the structured mesh around the airtanker is fixed. The number of grid is about 5.23×10^5, and the time step is 0.005s.

Fig. 2 shows the dropped water behavior of the 1/8 model varying with time. Clearly, the water shape of the simulation agree well with the experiment result.

a). Test



b). CFD

Fig. 2. The dropped water behavior of the 1/8 model varying with time

The local velocity distribution of the droplets at the rear of the airtanker is compared with PIV data, see Fig. 3. Both vertical and horizontal velocity are compared. It shows that the CFD result is approaching to the measurement data when time develops. The difference is related to the rear shape of the fuselage and the door opening type employed in experiment and CFD model. At 1.45s, the errors for vertical and horizontal velocity are less 20% and 15% respectively.

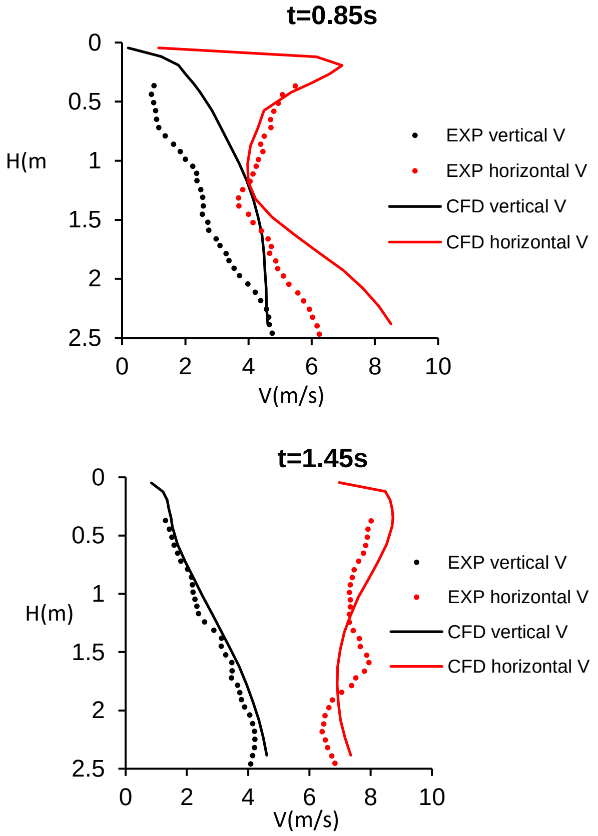


Fig.3. Droplet velocity distribution at different height for 1/8 model

Fig. 4 shows the water spread pattern on the ground for the 1/16 scale model. The location of the maximum precipitation for the CFD model is a little bit closer to the water door than the test. And CFD underestimates the maximum precipitation (3.5mm) than the experimental data (5mm), leaving the inaccuracy less than 30%. Actually, this accuracy is reasonable because it is quite difficult to predict both the location and precipitation at the same time in high accuracy. Other simulations results, such as Takeshi et al [6] used VOF coupled with DPM also underestimate the maximum precipitation.

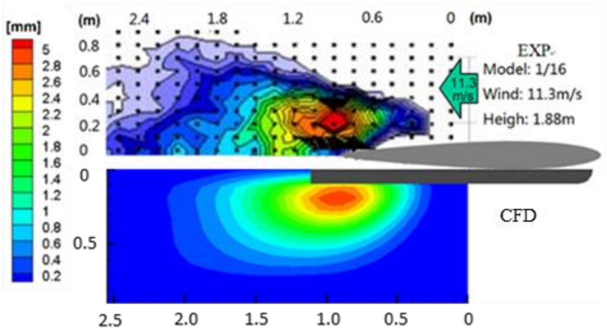


Fig. 4. Water spread pattern on the ground for 1/16 model

The validation of wind tunnel model shows that the CFD model is reliable to predict water spread pattern of airtanker.

3 Simulation of Dropping Water from the Full Scale AG-600 Amphibian

3.1 Model Description

AG-600 is the largest fire-fighting amphibian in China. The first fly is on Dec 24 in 2017. It is 36.9m in length, 38.8m in width and 12.1m in height. It can carry 12ton water in four cabins, see Fig. 5. Since the geometry of the cabins are quite close to each other, approximately, each cabin contains 3ton water. Considering the water flow in vertical direction, cabin door can be opened to maximum 85°. Therefore, the horizontal projection of the cabins was calculated, shown in Fig. 6. The projected area of cabin on x-y plane can be obtained, i.e. A=2(1.09×0.75+1.36×0.74) =3.65(m).

Fig. 5. Water cabins of the AG-600

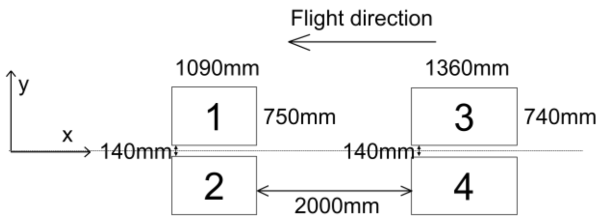


Fig. 6. Cabins projection from the top on the x-y plane

To evaluate the release effect and prepare for the drop test, sliding mesh techniques was employed to implement the movement of the airtanker. Simulation of the airtanker on various working conditions were carried out, see table 2. There are two types of release mode: (1)In the salvo release, water was dropped from cabins at the same time, after 0.7s, the cabins were empty; (2) In the successive mode, two diagonal cabins released 6 ton water in 0.7s, then after 0.8s interval, the other diagonal cabins released their 6 ton water in 0.7s. For the different volume of water released, cabin 2 and 3 were selected for 6 ton, cabin 1 to 3 for the 9 ton, cabin 2 and 3 then followed by 1 and 4 for the 12 ton successive release mode, cabin 1 to 4 for the 12 ton salvo release.

Table 2. Working conditions of full airtanker models

Speed (km/h) - pitch angle	210-8.5°, 230-3.9°, 250-1.1°
Release mode-volumes(ton)	Salvo-6, Salvo-9, Salvo-12, Successive-12
Height (m)	50,100,150
Cross wind velocity (m/s)	0,5,10
Temperature (°C)	4, 35
Release duration (s)	0.7

Altogether 9 CFD models were developed to simulate the 17 cases of working conditions. To save the calculation time, the semi model of the airtanker was employed to deal with the symmetric flow phenomenon, such as 12 ton water at salvo release without cross wind. Fig. 7 shows the geometry of the half airtanker.

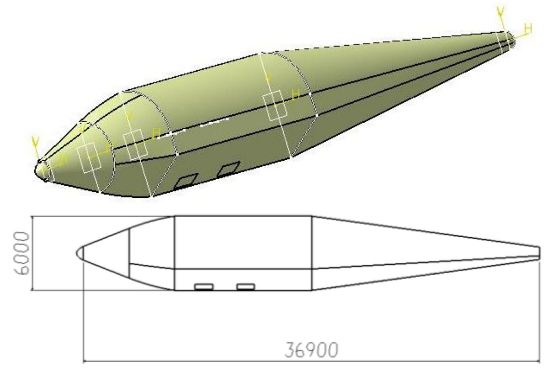


Fig. 7. The geometry and projection on vertical plane of the half airtanker

Fig. 8 shows the CFD model of the half airtanker. The airtanker is located in the moving zone on the top, the middle point of airtanker's longitudinal axis is set as z=0. The sliding zone is 406m ×36.3m ×72.6m in length, width and height. The stationary zone is 553.5m×72.6m×118.7m. Structured mesh was creased, based on different mesh size, 0.1m for the cabin surface, 0.2~1m for airtanker surface, 0.04m for the first boundary layer near the airtanker surface, 2m for the global size. The moving zone was divided into 54,000 grids and the stationary zone in 66,000 grids approximately.

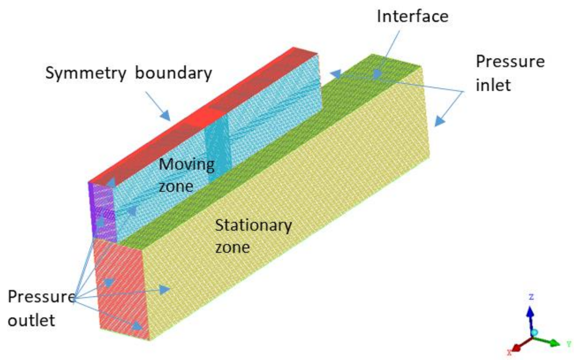


Fig. 8. The CFD model of half airtanker

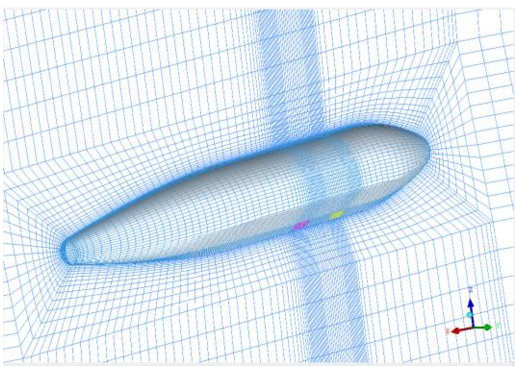


Fig. 9. Surface and adjacent mesh of the airtanker

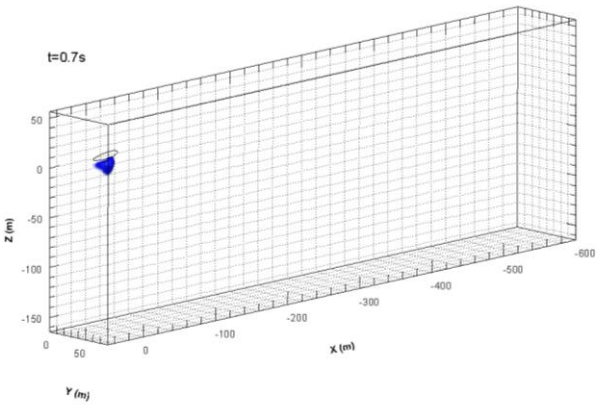
The boundary condition for the half model is plotted in Fig. 8. The sliding zone moves to the anti x direction in flight velocity. Symmetric boundary for the side wall located at the minimum y coordinate for the sliding and stationary zones. Interface between the two zones. Pressure inlet for the side wall located at minimum x coordinate for both zones. The surface of the airtanker is set as wall. The water cabin is set as velocity inlet, its z velocity can be calculated according to Eq. 1, where α represents the pitch angle in table 2. The velocity is referred to adjacent cell zone type.

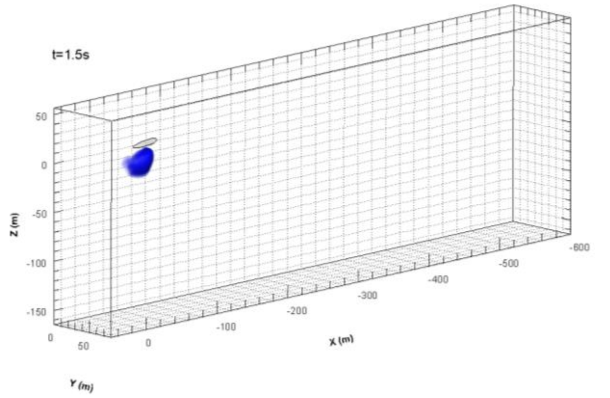
$$V_z = \frac{-Volume}{\mathrm{A} * \mathrm{Duration} * \mathrm{cos}\alpha} \qquad (1)$$

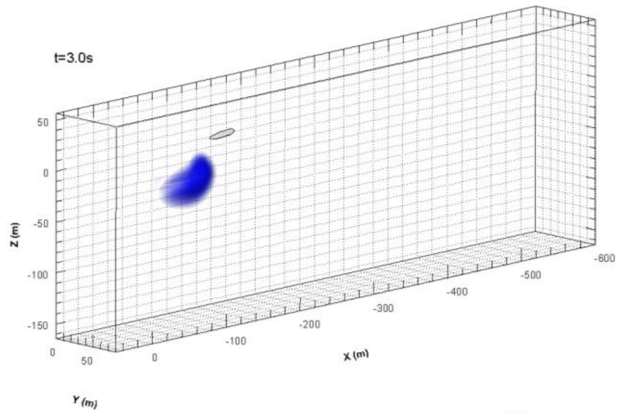
After the duration, the cabin outlet is set as wall. Other remained surfaces are set as pressure outlet. To collect water precipitation on different height, user defined function was developed to obtain the surface below the airtanker. Since the bottom surface of the model is 165m away from the airtanker, water pattern on the 150 height surface can be collected. The time step is set as 0.01s, after 10s, all the water drops on the ground and the calculation is over.

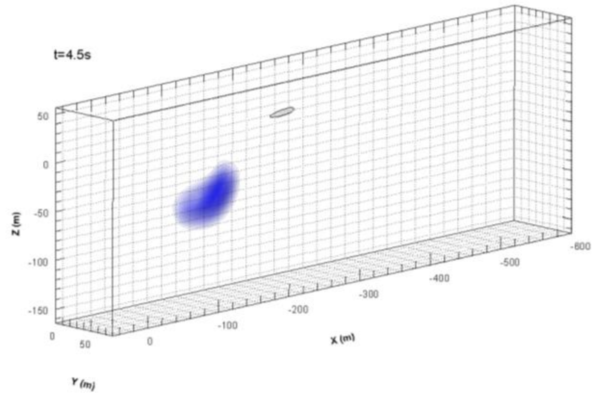
3.2 Simulated Patterns of Dropping Water from AG-600

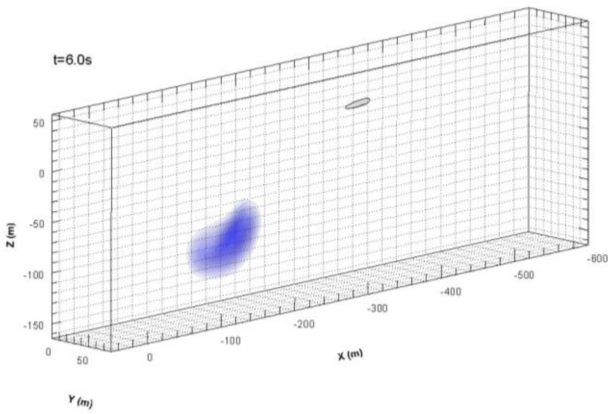
Fig. 10 shows the water spread pattern of the airplane at 230km/h, 35°C salvo release for 12ton without cross wind. It is a half model. The final figure shows the overlapped pattern in side view.











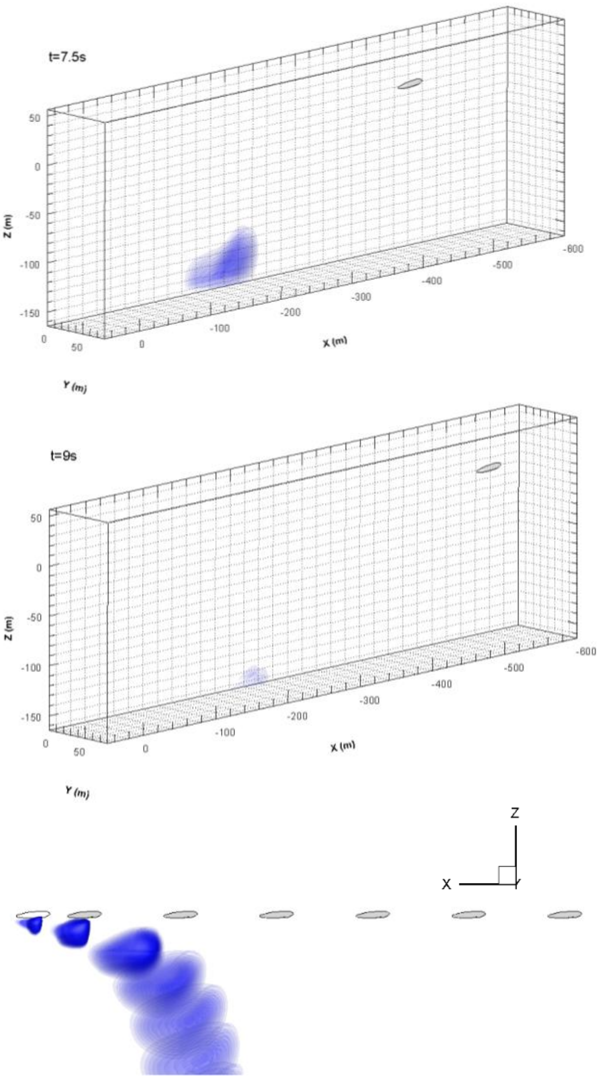


Fig. 10. Water spray pattern of the airplane at 230km/h, 35°C salvo release for 12ton water

It shows that after launch, the water moves forward and spreads, the shape deforms and velocity decelerates, about 2.5~4.5s all the water reaches to 50m height, after 5~7s reaches to 100m height, and after 7.5~9s reaches to 150m height plane.

3.3 Water Precipitation of AG-600

In order to assess the water spread pattern on the specified height, water precipitation distribution is plotted. It shows the maximum precipitation value and location. Meanwhile, the shape of effective precipitation is predicted.

3.3.1 Salvo Release without Cross Wind

Take the previous model as an example, the precipitation on various height is plotted in Fig. 11. As can be seen, when the height increases, dropped area moves to the flight direction, expands in width, and distributes evenly.

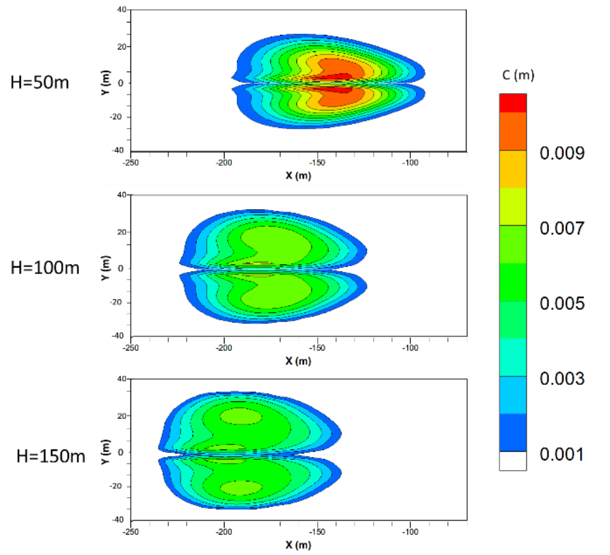


Fig. 11. Water precipitation on three heights for the salvo release 12ton, speed 230km/h at 35°C without cross wind

According to the reference [7], the coverage water area not less than the density of 2L/m^2 is effective for fire suppression. Therefore, the area is calculated for all working conditions and plotted on Fig. 12. In Fig. 12, the real segments are added to represent the distribution of the discrete points. Actually, the unit of L/m^2 in cup-and-grid method equals to mm water precipitation height in CFD model.

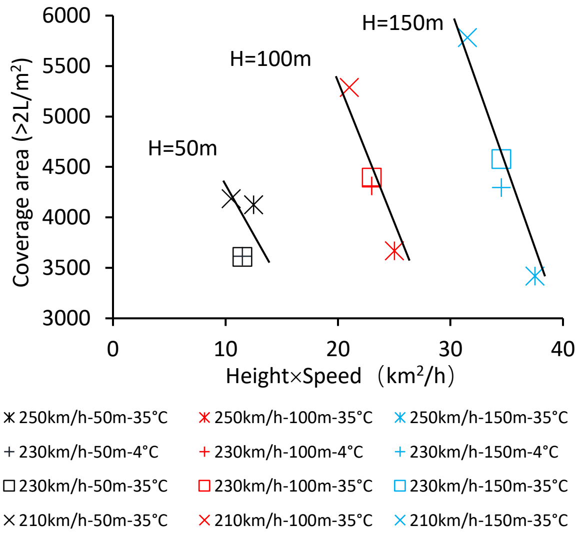


Fig. 12. Comparison of water covered area greater than $2L/m^2$ for salvo release 12 ton without cross wind

From Fig. 12, it is obviously that the covered area is between 3418~5782m^2. Considering the designed area requirement 4000m^2, most of the simulated working conditions meet the requirement. It shows that except 250km/h speed, the area increases with height under same flight speed. The differences between 4°C and 35°C is quite small. Under same height except 50m, the area decreases with flight speed.

The flight test of other airtankers are shown in Fig. 13.

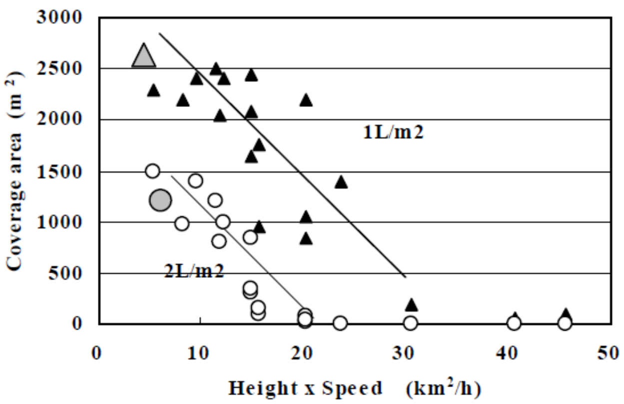


Fig. 13. Water covered area with the density 1 and $2L/m^2$ for real test, where the solid and blank signs are the data by PS-1(8.5ton) and the gray symbols are for US SP-2H (7.6ton) [7]

Compared to the test data in Fig. 13, because the tested water volumes are smaller, if we consider the $1L/m^2$ value, it can be found out that coverage area of the AG-600 is nearly double of the value in flight test.

(1) Flight velocity and height influence

To evaluate the location of the covered area, the coordinates in x direction are compared in Fig. 14, where the maximum, minimum and average value of the area border are plotted. It can be seen that the area location decreases with velocity and height.

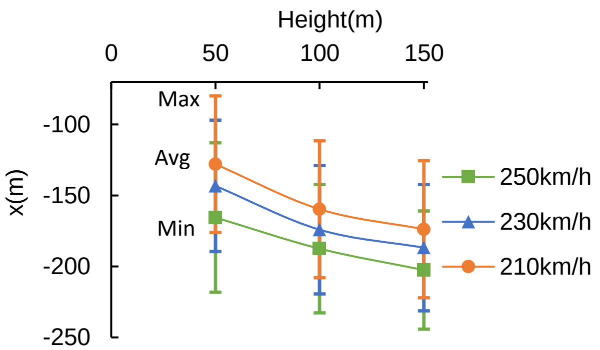


Fig. 14. X coordinate comparison of covered area ($>2L/m^2$) under various flight velocity in salvo release 12ton at 35°C without cross wind

The covered area y coordinate for various velocity is plotted in Fig. 15. Since the half model is used for the three case. Only the maximum y coordinate is used which represents half width of the area.

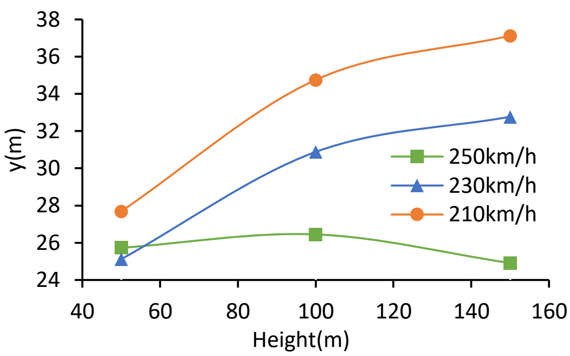


Fig. 15. Y coordinate comparison of covered area ($>2L/m^2$) under various flight velocity in salvo release 12ton at 35°C without cross wind

It shows that except 250km/h case, the width of the area increases with height. At the same height, the width decreases with the velocity, the difference of width becomes bigger when height increases.

To sum up the influence of velocity, when velocity increases, the precipitation location moves to the flight direction and the covered area decreases under same height, except 50m. The average precipitation value decreases, the length and width of the covered area change indistinctively.

The influence of the height is obtained. When the height increases, the precipitation location moves to the flight direction under the same flight velocity, except 250km/h. The average precipitation decreases, the length and width of the covered area change indistinctively.

(2) Atmosphere Temperature Influence

To evaluate the temperature influence on water drop, simulation were carried out for 4°C and 35°C. Parameters including air and water density, viscosity and water surface tension were changed to the relative temperature. Results shows that the precipitation distribution and shape are quite similar.

3.3.2 Successive Release without Cross Wind

As for the full airtanker model, the half model can be reflected, thus the domains are doubled. The symmetric boundary is deleted and new created side walls of the domain are assigned as pressure outlet.

The water precipitation of the successive release is given in Fig. 16. Compared to Fig. 11, the distribution is non symmetrical and stretched in flight direction. This is due to the successive opening of the cabins. Referred to Fig. 6, cabin 2 and 3 were opened first, followed by 1 and 4 after intervals. The water flow interaction between the front and rear cabins and the transient opening consequence of the cabin results in a condensed precipitation at the later opened cabins location, therefore, a non-symmetrical butterfly shape is formed.

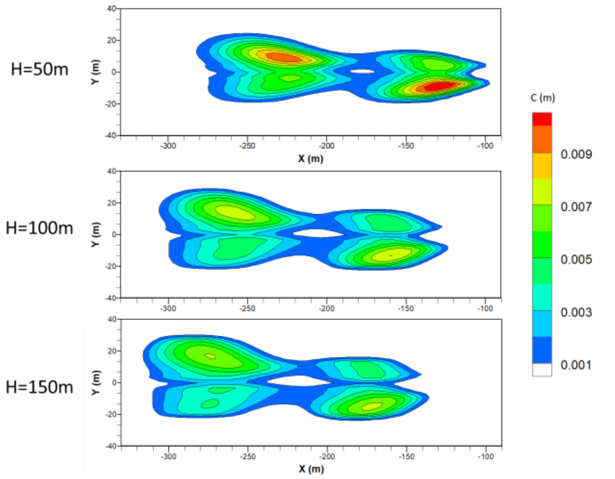


Fig. 16. Water precipitation on three heights for the successive release 12ton, speed 230km/h at 35°C without cross wind

(3) Release mode and volume influence

The covered area varying with release mode and volume are compared in Fig. 17, where successive 12ton represents the 6tons+6tons release. Others are salvo release.

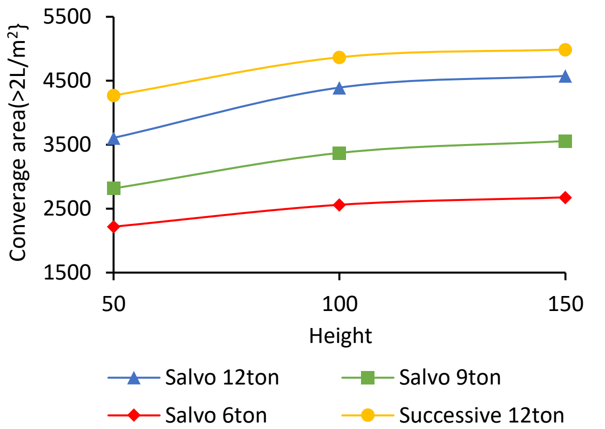


Fig. 17. Covered area comparison for release mode and volume at 230km/h, 35°C without cross wind

It shows that the successive release is better than salvo release for more covered area. As height increases, covered area is increased. The increase of release volume results in area increase. And the area for the successive 12 ton is less than the double of 6 ton in salvo mode.

3.3.3 Salvo Release with Cross Wind

The cross wind model was developed on the full airtanker model. Cross wind is applied on y direction. Thus the stationary zone of the full airtanker model is translated in anti y direction, to align with the sliding zone at the same minimum y coordinate. Therefore, the surfaces at minimum y are applied as velocity inlet, which are the windward surfaces of both zones. The leeward surfaces of both zones remain pressure outlet. The launching point of airplane is fixed at origin of the coordinate in all models.

Fig. 18-19 show the precipitation at cross wind 5m/s and 10m/s for the 230km/h salvo release mode. Compared to Fig. 11, it is obviously that precipitation is non-symmetrical in cross wind condition. The precipitation is concentrated in the windward direction with shape compressed. As height increase, the covered area moves to the wind direction with shape expanding, a diffusion on the leeward resulting in an elongated tail.

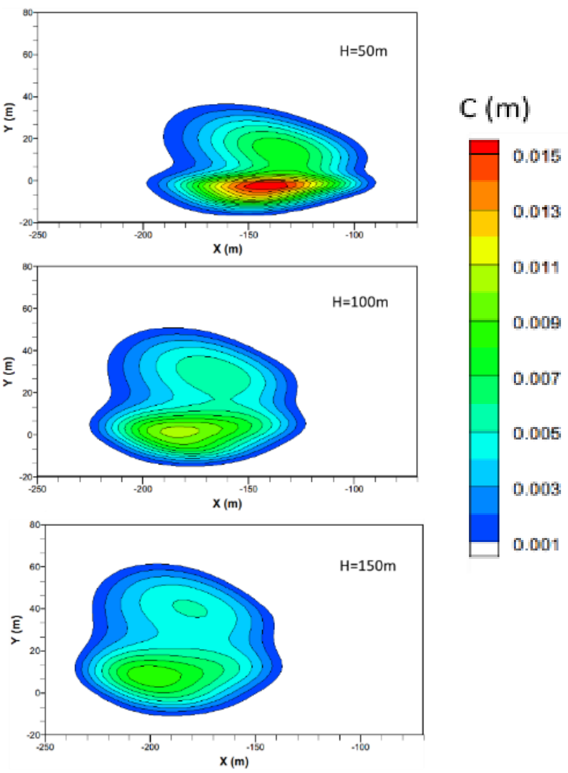


Fig. 18. Water precipitation on three heights for the successive release 12ton, speed 230km/h at 35°C with cross wind 5m/s

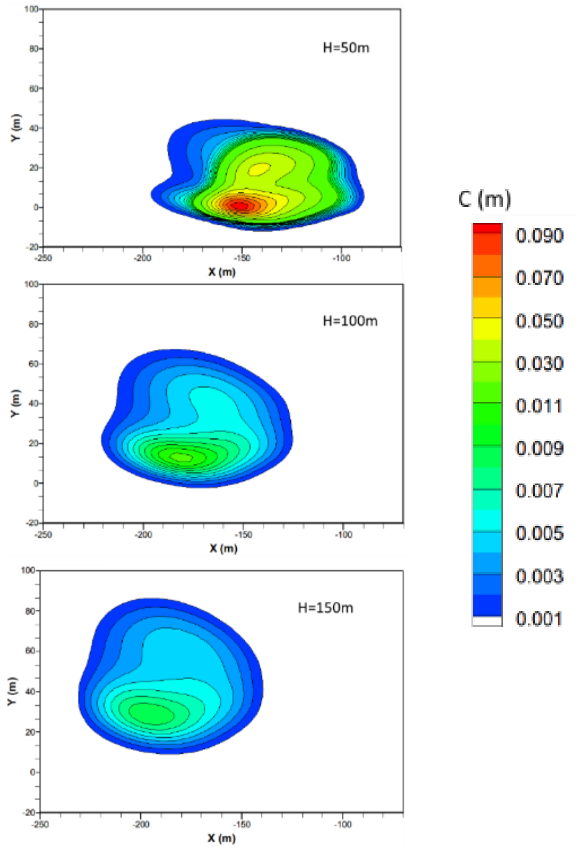


Fig. 19. Water precipitation on three heights for the successive release 12ton, speed 230km/h at 35°C with cross wind 10m/s

The covered area are compared in Fig. 20. It shows that the area varies with height in a complex way. At 50m height, 10m/s cross wind results in the maximum area but at 150m height, results in the minimum area.

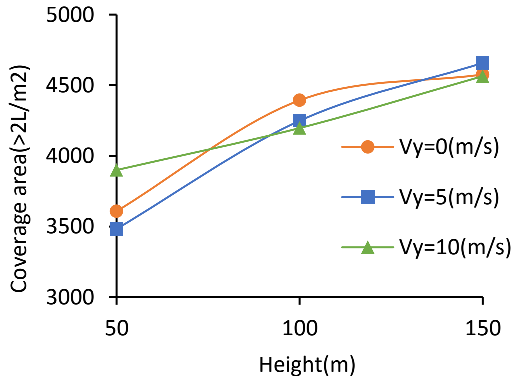


Fig. 20. Covered area comparison for cross wind at 12 ton salvo release at 230km/h and 35°C

The location of the coverage area in y coordinates is compared in Fig. 21. The minimum value, maximum value and average value are present. It shows that as wind velocity increases, the location of covered area moves to the wind direction and increase in height.

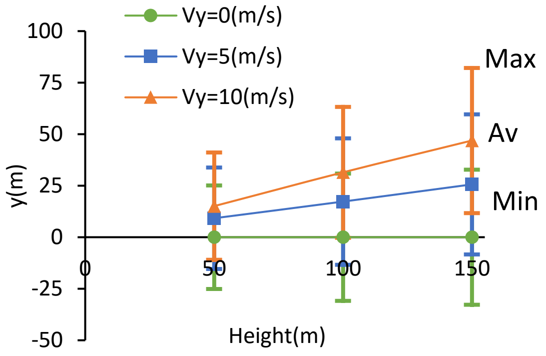


Fig. 21. Covered area y coordinate comparison for cross wind at 12 ton salvo release at 230km/h and 35°C

(4) Cross wind influence

It can be concluded from the previous analysis that the cross wind brings in the precipitation location to move in cross wind direction. But the covered area has a small change, a condensed precipitation is located in the windward direction.

4 Conclusions

The CFD simulations of water dropped by airtankers were carried out in this paper. Numerical results were validated with wind tunnel test. Both the water pattern and local velocity agree with the experimental data with satisfied accuracy.

The full scale model of AG-600 amphibian was developed based on sliding mesh and Euler multiphase flow model. Working conditions including flight velocity, height, volume of water, release mode, atmosphere temperature, cross wind were tested. The influences of each factors on the water precipitation distribution were discussed in details.

The water precipitation are compared with other aircraft flight test results.

The layout of the water cups in the measurement can be inferred from the CFD results. The research of this paper helps to assess the water dumping performance of airtankers and to optimize working conditions.

Acknowledgments

The authors are grateful to Chinese Flight Test Establishment for the funding and cooperation in the course of the work.

References

[1] Suter A, Drop testing airtankers: a discussion of the cup-and-grid method. *USDA Forest Service, Technology & Development Program*, 2000.

[2] Swanson D H, Luedecke A D and Helvig T N Experimental tank and gating system (ETAGS). *Honeywell Defense Systems Division*, 1978.

[3] Amorim J H, Numerical modelling of the aerial drop of firefighting agents by fixed-wing aircraft. Part I: model development. *International Journal of Wildland Fire*, Vol.20, pp384-393, 2011 a.

[4] Amorim J H, Numerical modelling of the aerial Drop of firefighting agents by fixed-wing aircraft. Part II: model validation, *International Journal of Wildland Fire*, Vol.20, pp 394–406, 2011b.

[5] Ito T et al. Evaluation of mission performance and safety for fire-fighting amphibian. *28th ICAS*, Australia, Brisbane, 2012.

[6] Ito T et al. Water-dropping aerodynamics for fire-fighting amphibian. *27th ICAS*, Nice, France, 2010.

[7] Satoh K et.al. A numerical study of water dump in aerial firefighting. *Fire Safety Science–Proceedings of the Eighth International Symposium*, *International Association for Fire Safety Science*, pp 777-787, 2005.

Contact Author Email Address

xuzhao@nwpu.edu.cn

Copyright Statement