ADVANCES IN FOREST FIRE RESEARCH

2022

Edited by

DOMINGOS XAVIER VIEGAS
LUÍS MÁRIO RIBEIRO

Numerical simulation of the aerial drops of the Canadair CL-415 and the Dash-8 airtankers

Corentin Calbrix[1,2]; Alexei Stoukov[1]; Axelle Cadière[2]; Benoit Roig[2]; Dominique Legendre[1,*]

[1]Institut de Mécanique des Fluides de Toulouse (IMFT) - Université de Toulouse, CNRS-INPT-UPS, 31400 Toulouse, France, {corentin.calbrix@toulouse-inp.fr}, {stoukov, Legendre}@imft.fr
[2]Université de Nîmes ; Laboratoire CHROME. 7 Place Gabriel Péri, France, {axelle.cadiere, benoit.roig}@unimes.fr

** Corresponding author*

Keywords

Airtanker, Aerial drop, Numerical simulation, Fluid mechanic, Liquid atomization

Abstract

Fighting wildland fires is a major issue for the protection of populations and environment with a need of more efficient means to fight fires. Airtankers are able to drop volumes of liquid (water or fire retardant) varying from less than 1m3 to several tens of m3, directly on fire or with the objective to form barriers of retardant to stop or reduce the fire propagation. However, the dynamics of a liquid dropped from an airplane has received few attentions in the scientific community, and related studies have mostly focused on Newtonian liquid jets injection in cross flow from millimetric injectors for combustion applications. For firefighting purpose, the liquid can be a retardant (a non Newtonian fluid) and the characteristic size of the delivery systems is of the order of a meter. The objective of this work is to demonstrate that Computational Fluid Dynamics (CFD) can be used to provide a deep understanding of the liquid fragmentation and dispersion when dropped from an aircraft. A numerical investigation is proposed for the analysis of airtanker performance and applied here to the biggest airtankers used in Europe: the Canadair CL-415 and the Dash-8. A numerical approach based on the Volume of Fluid method (VoF) is used to provide an accurate description of the tank discharge as well as to study the liquid ejection, fragmentation and atomization in air. The 3D unsteady resolution of the Navier-Stokes equations for both the liquid and the air allows us to provide a description of the main characteristics of the resulting liquid cloud, characterized by the vertical penetration of the liquid, its lateral expansion and the process of atomization.

1. . Introduction

Airtanker performances are usually tested with the cup & grid method (Suter 2000) developed by the USDA Forest service in USA and by the CEREN in France. Depending on drop conditions and systems, the resulting liquid deposit can present some irregularities resulting in zones of large concentration of product separated by zones of lower concentration offering a lower protection against fire (Plucinski *et al.* 2007). However, the dynamics of a fluid dropped from an airplane has received few attentions in the scientific literature with the development of simplified modelling (Amorim 2011; Legendre *et al*. 2014; Qureshi and Altman 2018), experiments in wind tunnel (Ito *et al*. 2010;) or numerical simulation (Rimbert 2003; Zhao *et al*. 2018). When considering airtanker the size of the delivery system doors is of the order of a meter, thus 3 orders of magnitude larger than millimetric injectors in combustion applications extensively investigated (Broumand and Birouk 2016), resulting in fragmentation and atomisation over a larger range of spatial scales making the experimental investigation more challenging. For that purpose, the use of Computational Fluid Dynamics (CFD) that allows to solve the liquid evolution after its exit from the tank can offer a promising tool. Indeed, all the information of the fluid evolution both in time and in space are available for a deep investigation. The objective of this work is to introduce the numerical approach proposed for the analysis of airtanker performance.

This study considers the aerial fleet of the French Civil Protection, more specifically the Canadair CL-415 and the Dash-8. These two airtankers are using different delivery systems. The Canadair CL-415 is made of two tanks incorporate in its fuselage and can drop a maximum of 6000 L of liquid. It has scoops on its belly, and can slide along a water surface to fill its tanks. The Canadair CL-415 is mostly used for a direct attack of the fire with water. The Dash8 is a fret plane reconverted with a tank attached under its belly and can drop a maximum of 10,000 L of liquid. Its tank is filled on the tarmac and this airtanker is used to provide line of

https://doi.org/10.14195/978-989-26-2298-9_265

retardant in coordination with ground operations. A numerical approach based on CFD is used here to describe the liquid drop and atomization in air.

The paper is organized as follows. In section 2, the numerical method used for this study is presented. The liquid atomization and dispersion in air is then described in section 3.

2. Numerical methods

In this first approach we consider a Newtonian fluid (water) of density ρ_L and viscosity μ_L released at velocity U_L in air of density ρ_G, viscosity μ_G and relative velocity U_G. For that purpose, we solve the 3D unsteady Navier-Stokes equations for two incompressible and immiscible Newtonian fluids. Considering the Volume of Fluid (VoF) approach, the interface between the two fluids is obtained by solving the transport equation of the VoF function (or liquid volume fraction) α ($0 \leq \alpha \leq 1$):

$$\frac{\partial \alpha}{\partial t} + \nabla . \alpha v = 0 \tag{1}$$

The evolution of the fluids velocity v and pressure P are given by the conservation of mass and momentum

$$\nabla . v = 0 \tag{1}$$

$$\frac{\partial \rho v}{\partial t} + \nabla . (\rho v v) = -gradP + \nabla . \bar{\bar{T}} + \rho g \tag{2}$$

where t is the time, $\rho = \alpha\ \rho_L + (1-\alpha)\ \rho_G$ is the one-fluid density, $\bar{\bar{T}}$ is the viscous stress tensor expressed using the one-fluid viscosity $\mu = \alpha\ \mu_L + (1-\alpha)\ \mu_G$ and g is the gravity. The numerical solver of Star CCM++ has been used and turbulence has been modeled with a standard k-ε model. The meshes built for the simulations are described in the following sections. For each case, at least two meshes of different resolution have been considered in order to test the effect of the grid resolution on the results. The simulations have been performed on the regional Super computer CALMIP based in Toulouse.

The proposed numerical approach has two steps. First, CFD is used to describe the respective tank discharge in order to provide an accurate description of the discharge flow rate. For that purpose, the tank geometries of the CL-415 and Dash-8 have been designed and meshed. For the geometry of the Canadair's tank, a trimmed cell mesher has been used. The number of cells is around 810 000 and their size ranges from 1.10^{-2} m at the center of the tank to 5.10^{-6} m at the walls. For the geometry of the Dash tank, a polyhedral mesher has been used. The number of cells is around 1.9 millions, and their size ranges from 2.10^{-2} m at the center of the tank to 5.10^{-6} m at the walls. The resulting time evolution of the liquid velocity U_L(t) at the exit of the two tanks has recorder. The maximum velocity observed after 0.5s for the two tanks is around 6 m/s for the Canadair and 4.8m/s for the Dash8. The time to discharge the Canadair tank is around 1s while it is around 4s for the Dash8. This information is then considered for the second step to study the liquid drop and atomization in air. U_L(t) is imposed as inlet condition as described in the next section.

3. Liquid drop from Canadair CL-415 and Dash8

3.1. Domains and meshes

The second objective of the numerical approach is to simulate the dynamics of the liquid ejection, fragmentation and dispersion in air. The simulations are conducted in the frame of reference moving with the airplane. For each case, the air flow velocity is imposed at U_G=50 m/s at the domain inlet, which corresponds approximately to an operational speed for these aircrafts. Each belly of the studied airplanes has been meshed in an adapted domain. The domain is selected in order to observe the liquid fragmentation during its evolution in air while minimizing the size of the computational domain to reduce the number of cells. Several domain shapes have been tested and the selected ones are shown in Fig. 1 for the Canadair and the Dash-8.

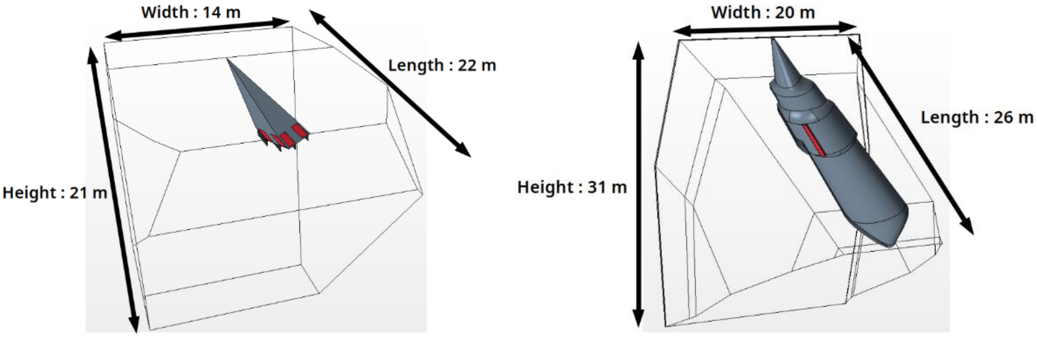

Figure 1 - Dimensions of the numerical domain. (left) under a Canadair belly with the four exits in red. (right) under a Dash8 belly with the exit in red.

Due to the complex geometry of the belly and the sharp angles on the corner of the domains, we have chosen in both cases a polyhedral mesher. The grid size ranges from 8.10^{-6}m at the walls to 8.10^{-2}m in the center of the domain. The total number of cells are 6.9 millions and 10.9 millions for the Canadair and Dash8, respectively. Parallel simulations made use of 180 and 720 processors, with computation CPU time of 18h and 40h, respectively. The liquid evolution in air is described in the following by considering its vertical penetration, lateral expansion and atomization. The liquid interface being tracked with the VoF model, we are able to select different values for the liquid volume fraction α for the liquid cloud inspection. "The liquid core" corresponds to $0.9 \leq \alpha \leq 1$, while the envelop of the liquid cloud can be described by considering $0.001 \leq \alpha \leq 1$.

3.2. Liquid vertical penetration

The liquid vertical penetration is defined as the evolution of the vertical distance *z* of the front of the liquid surface as a function of the streamwise distance *y*, red and black arrows in Fig 2, respectively. The shape of the corresponding curve indicates how the liquid front is deformed by the impact of the relative air flow. The liquid penetration *z* is compared for the Canadair and Dash8 in Fig. 2 at *t*=0.5s. As shown the penetration of the Dash8 and the Canadair are very close because the liquid velocity is of the same order of magnitude at *t*=0.5s. This confirm results observed for millimetric liquid jets where the penetration is controlled by the momentum ratio $q = \rho_L U_L^2 / \rho_G U_G^2$. Then the liquid penetration for the Canadair rapidly decreases and becomes less important than observed for the Dash-8.

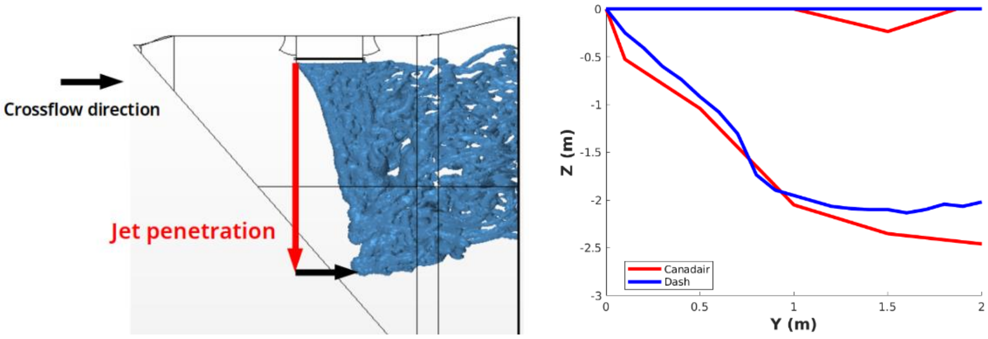

Figure 2 – Liquid penetration. (left) Visualization of the iso-surface of a=0.001 for the Dash8 at t=1s. (right) Comparison at t=0.5s of the liquid vertical penetration z as a function of the streamwise distance y for a Canadair (red) and a Dash8 (blue).

3.3. Liquid lateral expansion

The lateral (or transverse) expansion *x* is defined as the maximum width of the dispersed liquid as shown in figure 3. This parameter is important because its evolution controls the final width of the liquid deposit on ground. As shown in the figure, the lateral expansion is reduced for the Dash-8 compared to the Canadair, resulting in a more narrow liquid deposit on ground, and thus a higher liquid concentration.

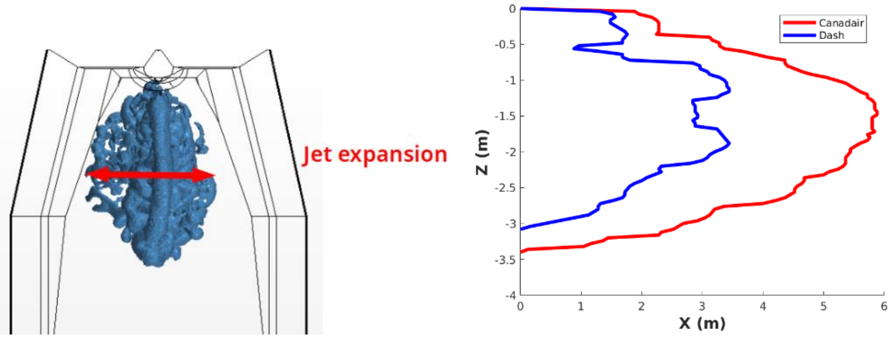

Figure 3 – Liquid lateral expansion. (left) Visualization of the iso-surface of a=0.001 for the Dash-8 at t=0.5s. (right) Comparison of the liquid lateral expansion x as a function of the streamwise distance y for a Canadair (red) and a Dash8 (blue).

3.4. Liquid atomization in air

One important aspect of the liquid evolution in air is related to the liquid fragmentation and dispersion. The objective is to finally obtain on ground a uniform liquid deposit. We consider here the evolution of the number and the size of produced droplets as illustrated by figure 4 with the observation of the formation of ligaments and droplets of large volume (several tens of cm). The shear stress from the air at the liquid surface tear off liquid ligaments, which are then fragmented into liquid volumes of different size. Their following dynamics may differ and result in different contributions to the ground pattern.

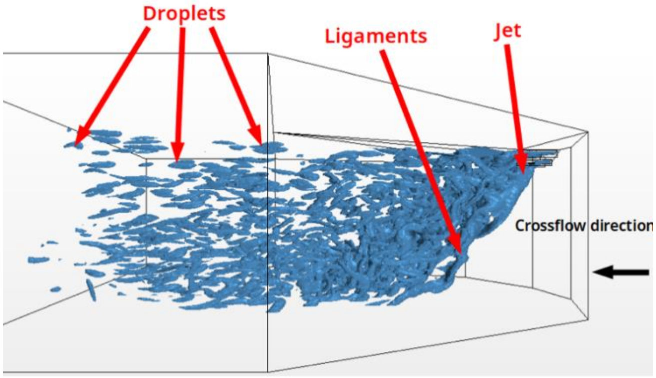

Figure 4 – Liquid atomization process for a Canadair CL-415 drop at t=1s. The iso-surface of the volume fraction is a =0.001.

A Matlab code has been developed to reconstruct the 3D liquid structures at each instant, to characterize their respective velocities and sizes. Figure 5 (left) reports the time evolution of the number of the identified liquid structures for the Canadair CL-415 for two thresholds for the detection: $\alpha \geq 0.9$ and $\alpha \geq 0.1$. The first one identifies droplets while the second one identifies volume of fragmented liquid in a more important number. Figure 5 (right) shows the distribution of droplets with their respective velocities for three different classes of size. The velocity components v_Z, v_Y and v_X, for the vertical, streamwise and transverse components, respectively, are reported. As shown the streamwise velocity is much larger because imposed by the relative airflow (50m/s), and the vertical velocity of lower magnitude is controlled by the velocity at ejection. The order of magnitude of the transverse velocity is around 1m/s, showing a significant liquid dispersion in the transverse direction.

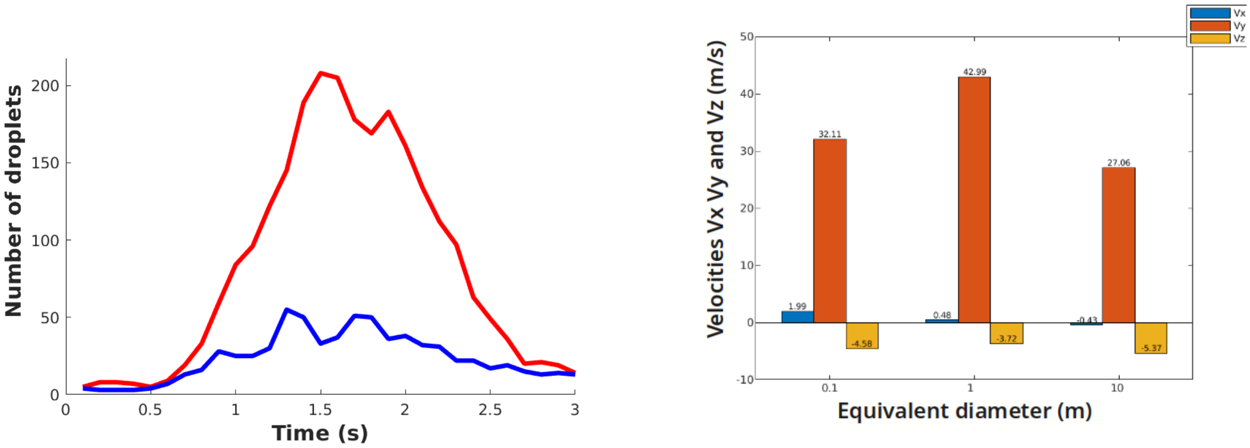

Figure 5 – Liquid fragmentation for the Canadair CL-415. (left) Time evolution of the number of identified fluid structures. Comparison for a detection using a liquid volume fraction a between 0.1 and 1 (red), and 0.9 and 1 (blue). (right) Repartition of the droplets with their respective velocities and size at t=1s for a detection with a volume fraction a between 0.1 and 1.

4. Conclusion

In this work, a numerical strategy has been proposed to deeply investigate the drop of a liquid from an airtanker. For this study, we have considered water drop from the Canadair CL-415 and the Dash-8. The 3D unsteady resolution of the liquid motion for both the liquid and the air allows us to provide a description of the main characteristics of the resulting liquid cloud. These liquid drops have been characterized through three main parameters: the vertical penetration, the lateral expansion and the structure of the liquid cloud during the atomization process. The next step will consist in extrapolating from these simulations the resulting liquid deposit on ground in order to make possible a direct comparison with the real tests performed using the cup & grid method.

5. Acknowledgments

This project is funded by Nîmes Métropole. It also benefited computation resources on the super computer CALMIP under the allowance 2020 – P20015.

6. References

Amorim J. H. (2011) Numerical modelling of the aerial drop of firefighting agents by fixed-wing aircraft. Part I: model development, *International Journal of Wildland Fire*, **20**, 384–393.

Broumand M, Birouk M (2016) Liquid jet in a subsonic gaseous crossflow: Recent progress and remaining challenges Progress, *Energy and Combustion Science*, **57**, 1–29.

Ito T, Kato H, Goda Y, Tagawa S, Negishi E (2010) Water-dropping aerodynamics for fire-fighting amphibian, 27th International congress of the aeronautical science (ICAS2010).

Legendre D, Becker R, Alméras E, Chassagne A (2014) Air tanker drop patterns, *International Journal of Wildland Fire*, **23**(2), 272-280.

Plucinski M, Gould J, Mccarthy G, Hollis J (2007) The effectiveness and efficiency of aerial firefighting in Australia, Part 1, Bushfire CRC Technical Report n°A0701, Australia. Www.bushfirecrc.com/sites/default/files/managed/resource/aerial_suppression_report_final_web_0.pdf

Qureshi S, Altman A, (2018) Studying fluid breakup and dispersion to predict aerial firefighting ground drop patterns, AIAA Aerospace Sciences Meeting.

Rimbert N (2003) Contribution à l'étude de la pulvérisation et de la dispersion dans l'air de fluides newtoniens et non-newtoniens. Application au largage aérien d'eau et de mélanges retardants, PhD thesis, Institut National Polytechnique de Lorraine, Nancy, France.

Suter A (2000) Drop testing airtankers: a discussion of the cup-and-grid method, USDA Forest Service, Missoula Technology and Development Center, Technical Report 0057-2868-MTDC.

Zhao X, Zhou P, Yan X, Weng Y, Yang X-L (2018) Numerical simulation of the aerial drop of water for fixed-wing airtankers, 31st International congress of the aeronautical science (ICAS2018).