Applied Computer Science, vol. 19, no. 3, pp. 47–63
doi: 10.35784/acs-2023-24

Submitted: 2023-08-23 | Revised: 2023-09-08 | Accepted: 2023-09-14

Keywords: aerial firefighting, fixed-wing aircraft, CFD analysis, waterbombing

Zbigniew CZYŻ [0000-0003-2281-1149]*, *Paweł KARPIŃSKI* [0000-0001-5786-1248]**,
Krzysztof SKIBA [0000-0001-5504-3946]*** , *Szymon BARTKOWSKI* *

NUMERICAL CALCULATIONS OF WATER DROP USING A FIREFIGHTING AIRCRAFT

Abstract

The study involved a numerical analysis of the water dropping process by fixed-wing aircraft. This method, also known as air attack, is used for aerial firefighting, primarily in green areas such as forests and meadows. The conducted calculations allowed for the analysis of the process over time. The calculations were performed based on a SolidWorks model of the M18B Dromader aircraft. After defining the computational domain and setting the boundary conditions, the simulations were carried out using the ANSYS Fluent software. The resulting water dropping area was used to analyze the intensity of water distribution. The volumetric distribution and airflow velocity distribution were analyzed for specified time steps. The boundary layer where air no longer mixes with water during the final phase of water dropping was also determined. The obtained results provide an important contribution to further analyses aimed at optimizing the water dropping process by fixed-wing aircraft.

1. INTRODUCTION

Aircraft are used worldwide to combat fires in large green areas, technical infrastructure, and residential buildings. In recent years, the climate has significantly changed and more frequent heatwaves and droughts occur resulting in a higher number of large-scale fires. Forest fires and urban fires differ in nature and require different firefighting solutions. Urban fires have a concentrated scope and are isolated, necessitating a denser water delivery for firefighting (Satoh et al., 2000).

In some countries, special airborne units are established for firefighting in green areas. In Poland, the PZL M18 Dromader aircraft are used for extinguishing fires. This study addresses the development of a numerical model for water dropping performed by the mentioned aircraft. Such a topic has been taken up due to the continuous use of this type of aircraft for firefighting purposes and the lack of scientific research dedicated to this aspect

---

[*] Polish Air Force University, Aeronautics Faculty, Poland, z.czyz@law.mil.pl
[**] University of Life Sciences in Lublin, Faculty of Production Engineering, Department of Machine Operation and Production Process Management, pawel.karpinski@up.lublin.pl
[***] Lublin University of Technology, Faculty of Mechanical Engineering, Department of Thermodynamics, Fluid Mechanics and Aviation Propulsion Systems, Poland, k.skiba@pollub.pl

for this particular type of aircraft. The conducted analysis aims to gain a better understanding of the firefighting process using the PZL M18 Dromader and lay the groundwork for future optimization efforts in this area.

The aircraft under consideration can carry on board approximately 2 200 dm^3 of water. The largest fixed-wing aircraft can carry from several thousand (McDonnell Douglas MD-87, 15 000 dm^3) (Walton, 2018) to tens of thousands of dm^3 of retardant (Boeing 747 Supertanker, 72 000 dm^3) (Ahlgren, 2020). In addition to fixed-wing aircraft, helicopters equipped with suspended tanks and buckets are also used for firefighting. A helicopter can be equipped with a foam cannon and its tanks are filled on the ground using high-pressure equipment, whereas suspended buckets can be filled in natural water bodies such as rivers, lakes, or ponds. The efficiency of the aerial firefighting process using helicopters equipped with the Bambi Bucket were experimentally analyzed. To ensure safe and satisfactory helicopter operations, it is necessary to adjust the flight direction, altitude, and speed to terrain characteristics, weather conditions as well as horizontal and vertical smoke spread from the fire site (Kal'avsky et al., 2019). For firefighting missions performed by aircraft, in addition to aerodynamics, the specific aircraft load during maneuvers must also be considered (Kliment et al., 2015).

In the analysis of aerial retardant drop efficiency, the ground coverage level, which represents the volume of fire retardant per area, is essential. Predicting terrain patterns and coverage levels is a complex issue. General factors influencing this process include speed and volumetric flow rate, while aircraft-specific factors involve the geometry and distribution of the tank outlet (Qureshi & Altman, 2018).

Due to differences in water storage and the water dropping process between fixed-wing aircraft and helicopters, separate analyses are conducted on these issues. A commonly used method for conducting flow analyses, including multiphase flows, is Computational Fluid Dynamics (CFD). This method can be applied to analyze the aerodynamics of aircraft (Czyż et al., 2020) and study water dropping patterns by aircraft (Satoh et al., 2004, Satoh et al., 2005). Additionally, a three-dimensional, computational panel method can be used to plot on-body and off-body streamlines in various firefighting aircraft configurations (Varner et al., 2019). The developed numerical models can later be validated in experimental studies in a wind tunnel (Czyż et al., 2022). For modeling multiphase flows using CFD techniques, the Eulerian multi-phase model is employed (Zhao et al., 2018). Aerodynamic research was conducted on water drops released from a firefighting amphibious model in a wind tunnel using Particle Image Velocimetry (PIV), considering the Weber number and Froude number (Ito et al., 2010). This method was adopted to numerically model vertical water drops released by a helicopter in hover using the Volume of Fluent Model and mesh adaptation. Such research allows for determining water distribution on the ground (Zhou et al., 2022). The grid-particle coupling method serves as an alternative to the Volume of Fluid method and the Discrete Droplet Model. In this method, gas and liquid phases are respectively calculated using grids and particles and are coupled using the aerodynamic force model developed by the authors (Tsujimura et al., 2021). The effectiveness of building fire suppression using helicopters can also be evaluated based on scale model experiments. The most efficient fire suppression method is to spray a large amount of water at once during the fire expansion stage (Konishi et al., 2008).

The process of modeling water drops in aerial firefighting is based on a series of mathematical techniques. The gas phase can be modeled by a vegetative canopy model coupled with a modified surface-layer model under diabatic conditions. In the case of the solid particle phase, the primary breakup of the retardant stream is achieved by continuously removing droplets from exposed surfaces using the Rayleigh-Taylor and Kelvin-Helmholtz instabilities (Amorim 2008, Amorim et al., 2008). The Aerial Drop Model can simulate the continuous removal of droplets from the liquid stream using these instabilities with the application of linear stability theory (Amorim 2011). Developed mathematical models often require validation. The aerial drop model can be validated by comparing ground-calculated formulas with experimental data from full-scale aerial drop tests using water and a wide range of firefighting agents' viscosities (Amorim 2011). Additionally, a general dynamic model of an aerial firefighting system was considered. Such a system comprises a tethered hose, a high-pressure water pump, and multi-rotor aircraft (Chaikalis et al., 2022).

Mathematical techniques such as Machine Learning Algorithm (MLA) can be used to quickly determine the optimal dynamics of an aircraft (unmanned or manned) to maximize the efficiency of releasing fire retardant (Zohdi 2021). The water dropping scheme for fixed-wing firefighting aircraft can be optimized using a neural network combined with a genetic algorithm to search for the optimal water dropping scheme (Wang et al., 2021).

The water dropping process is influenced by the design and location of the water tank on the aircraft. A biplane configuration is considered a good solution for long-distance firefighting flights with a high-capacity tank due to its compact design and short turn-around time with a small radius (Goraj et al., 2001). Even with an optimized firefighting aircraft design, pilot training is essential, and Virtual Reality (VR) technology can be used for this purpose (Clifford, Khan et al., 2018, Clifford, Hoermann et al., 2018). Creating a physics-based simulation model of water dropping from a helicopter is crucial for achieving realistic water visualization and increasing the fidelity and immersion of the virtual environment (Han et al., 2018).

In addition to analyzing water dropping by a specific aircraft, selecting an appropriate aircraft for a particular fire is also crucial. For example, the fuzzy proximity measure method can be useful to select the most suitable aerial firefighting aircraft based on decision criteria using a multiple attribute decision making analysis (Ardil, 2023).

In the analysis conducted in this study, water was used as the extinguishing agent. It should be emphasized that other agents such as foam, gel or special chemical agents to reduce flammability are also used in aerial firefighting.

The analysis conducted in this work is related to the field of industrial engineering due to the necessity of the proper design and manufacturing of aircraft adapted for aerial firefighting by specialized aviation production facilities. The chosen topic in this study is particularly significant from a safety perspective. Analyzing the distribution of the liquid stream allows for evaluating the efficiency of the firefighting process while maintaining a safe distance for the aircraft and assessing the impact of water dropping on aircraft's aerodynamics. The obtained results have practical value and can be successfully applied in firefighting operations as well as in the optimization of aircraft intended for this purpose.

The ongoing climate change and the resulting heat waves and droughts in certain regions of the world such as areas of Southern Europe, Central Asia and North America are resulting in extensive forest and grassland fires that are difficult or impossible to extinguish by land units. This is due to the difficulty in accessing terrains (lack of roads, mountainous areas) and the size of the fire area, reaching up to thousands of hectares. The remedy for this problem is to extinguish fires by air using aircraft. This justifies the need for research and optimization of the aerial firefighting process. Due to the high cost of experimental research using specialized aircraft, a very good alternative is simulation research, which successfully can answer questions about the process of extinguishing fires in a quick and low-cost way and allows evaluation of its efficiency.

This work presents numerical calculations of water drops using a firefighting aircraft. For this purpose, a geometric model of the research object was developed, as described in Section 2. Subsequently, the details of the computational model were defined, a computational mesh was created, and boundary conditions were defined (Section 3). The created model was applied to calculate and analyze the obtained results for the water dropping process simulation at the assumed flight height above the ground surface (Section 4). Finally, the performed calculations and analyses were summarized in Section 5.

2. RESEARCH OBJECT

The PZL M18B Dromader, presented in Fig. 1, is a single-engine aircraft with a low-wing configuration and metal construction, equipped with a fixed main landing gear and a tail wheel. Its fuselage and wings are covered with duralumin sheets. Each of the wings contains fuel tanks with a capacity of 200 liters per wing. The fuselage is constructed with a truss made of chrome-molybdenum steel tubes. The power unit consists of an air-cooled radial engine with turbocharging, providing a takeoff power of 1 000 HP. The cockpit accommodates one pilot only. The water and chemical tanks are made of laminate and mounted in front of the pilot's cabin and behind the power unit, which enhances the structure's crush resistance and protects the pilot in case of an aircraft accident. The Dromader aircraft is capable of taking off and landing on grass and unpaved runways with a strength rating of 3.5 kg/cm^2 (Oleksiak et al., 1975).

Fig. 1. M18B Dromader aircraft

3. GEOMETRICAL MODEL OF THE TEST OBJECT

The model for the analysis, shown in Fig. 2, was designed using SolidWorks. Technical drawings of the aircraft were used to develop the CAD model. The geometry of the aircraft was simplified to correctly generate the computational mesh necessary for the numerical calculations.

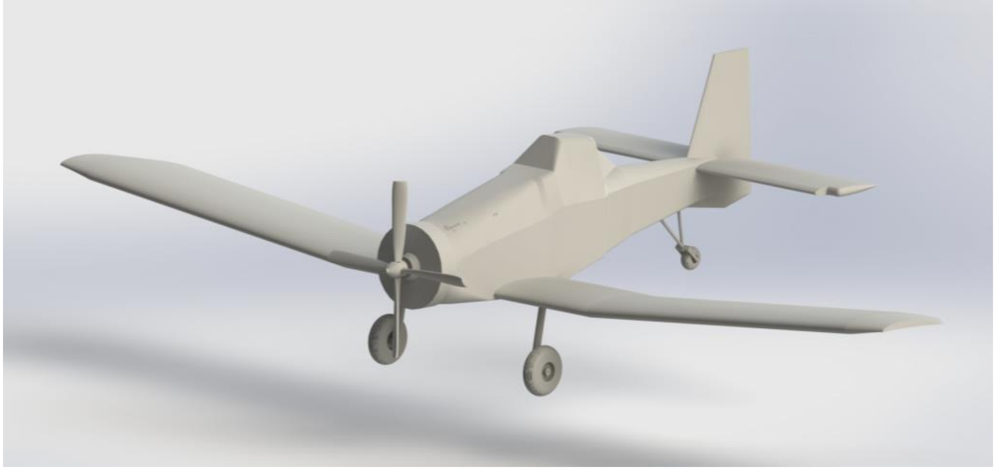


Fig. 2. Model of the PZL M18 Dromader developed in the SolidWorks software

Due to the symmetrical shape of the study object, the computational domain (Fig. 3) was set as a half model with dimensions of 40 × 25 × 40 m (L × H × W) with the drop height set at 15 m.

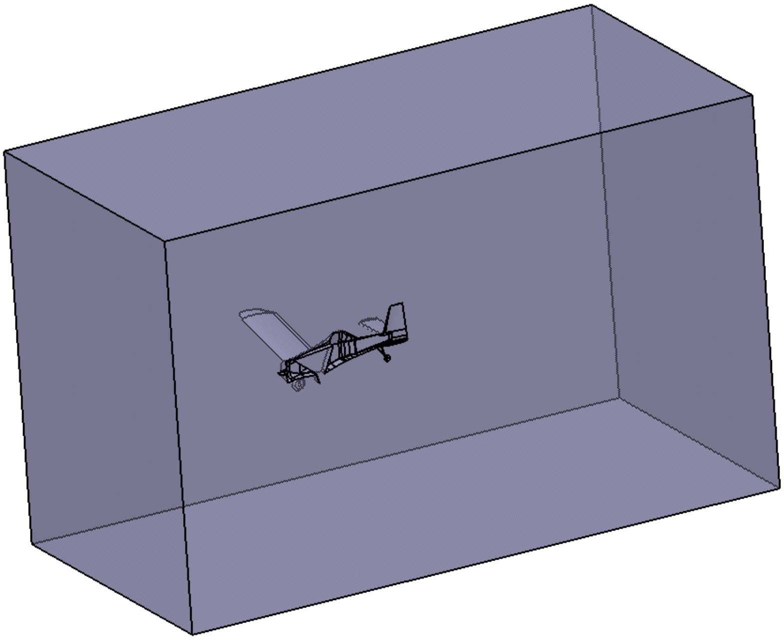


Fig. 3. Computational domain with the research object

4. CFD WATER DROP MODEL

The next preparatory step was mesh generation. For the mesh generation process, the minimum and maximum element sizes were set to 10 mm and 2 000 mm, respectively, with a growth rate of 1.2. The mesh consisted of 3.6 million elements and 912 000 nodes (Fig. 4 and Fig. 5). Tetrahedral elements with a patch conforming method were used to generate the mesh.

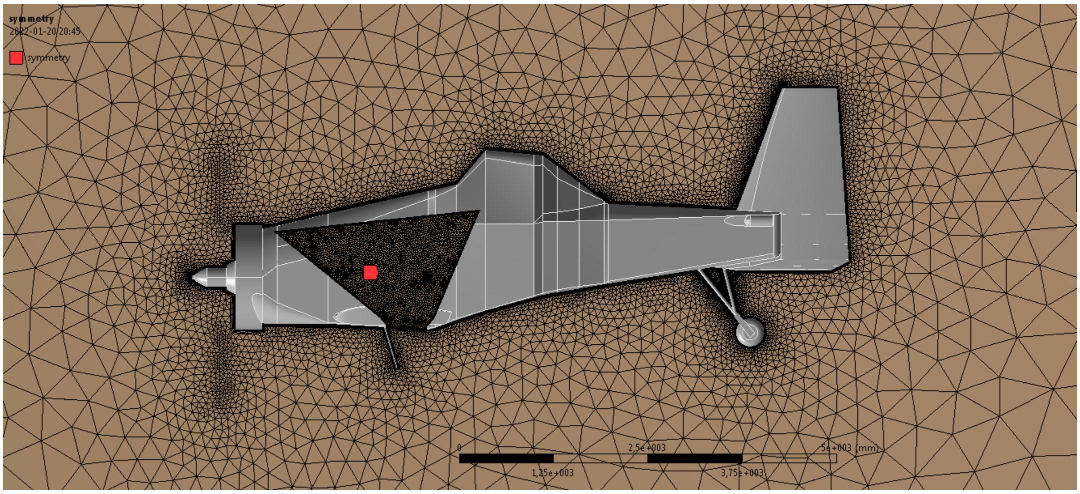


Fig. 4. Computational mesh in the symmetry plane

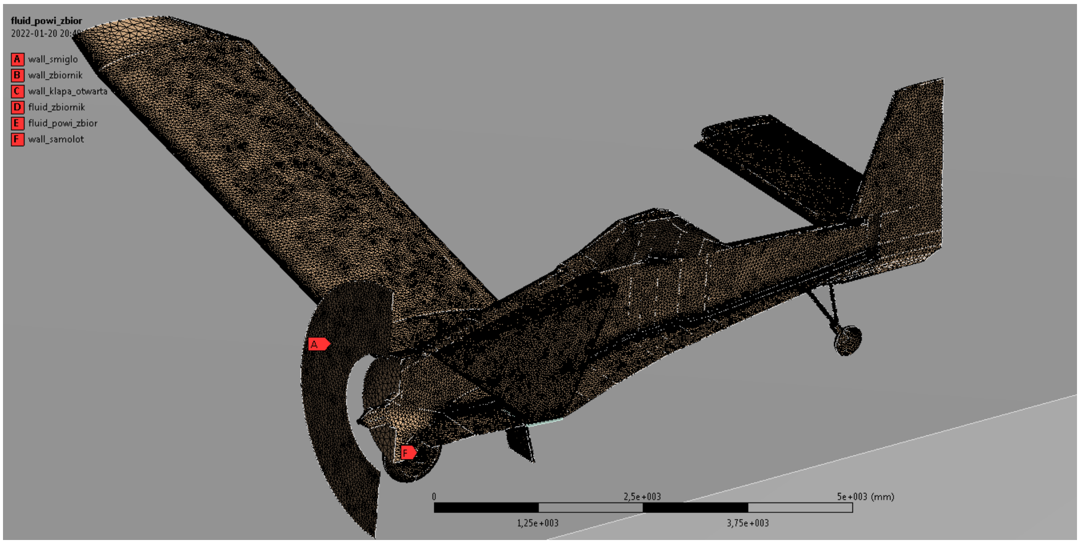


Fig. 5. Computational mesh on the surface of the examined object

The inlet to the computational domain was set as a velocity-inlet, with the airspeed defined at 30.3 m/s, corresponding to the minimum aircraft speed. The turbulent intensity was specified at 1%, and the turbulent viscosity ratio was set at 1. The outlet from the computational domain was defined as a pressure-outlet, with the same turbulent intensity and turbulent viscosity ratio parameters as the inlet. The symmetry plane of the computational domain was established as the symmetry, allowing results to be referenced to the entire computational domain using mirror reflection. Additionally, the model included a surface where a pressure jump boundary condition was applied with a pressure jump value of 752 Pa.

The calculations were performed using an ANSYS Fluent computational solver. The Authors decided to use this software for several reasons. It is one of the world's most popular software for conducting research using computational fluid dynamics (CFD) in various scientific and industrial sectors (aerospace, mechanical or environmental engineering). Its code uses advanced algorithms and mathematical flow models that provides highly accurate results for various boundary conditions (including turbulence models). The software allows calculations for multiphase models. In addition, the Ansys environment includes a CFD-Post module that enables a comprehensive engineering analysis of results. This tool was available to the Authors under a research license at an academic institution. Compared to experimental studies, Ansys Fluent allows a series of tests for different geometries and boundary conditions in a much shorter time and at a much lower cost (no need to build a complex test bench). The analysis type was set as transient. Due to simulating the water flow from the tank into the surrounding air, which represents the air, it was necessary to include the volume of fluid model in the simulation. The k-ε (2 eqn) turbulence model was adopted for the calculations. The k-ε model is one of the most popular turbulence models used in computational fluid dynamics. It is based on a two-variable mathematical algorithm that characterizes turbulence kinetic energy k and turbulence energy dissipation ε. The k-ε turbulence model was adopted for the calculations because it is well suited for general analysis of aerodynamic interactions in turbulent flows with high velocities and large velocity gradients. In particular, the model is used for the analysis of airfoil drag and other aircraft structures. Another frequently used turbulence model in scientific research is the k-ω model. However, this model is applicable at lower flow velocities and at high temperature gradients. In addition, it is preferred over the k-ε model for analyzing the boundary layer. The flow velocities in the conducted study reached 30 m/s. The wall layer around the aircraft fuselage and the tank flap were not analyzed. Therefore, the appropriate model for the calculations was the k-ε model. The first phase was defined as air, while the second one as water. In the next step, the density of water was set to 998.2 kg/m^3, and the density of air was set to 1.225 kg/m^3. The time step for the transient analysis was set to 0.001 s. The entire analysis took 2.684 seconds, resulting in a total of 2 684 computed steps. The results were recorded every 0.02 seconds, equivalent to 20 steps. This corresponds to 134 separate cases. In the calculations, it was assumed that the water drop valve would open at time t_1 = 0.5 seconds. Figure 6 shows the contour of the second phase, i.e. the volume fraction of water in the computational domain at time t = 0.48 seconds, just before the water drop valve opens. Water is marked in red, while air in blue. The changes of each color from red to blue represent a different volume fraction of water in the air, expressed as a percentage.

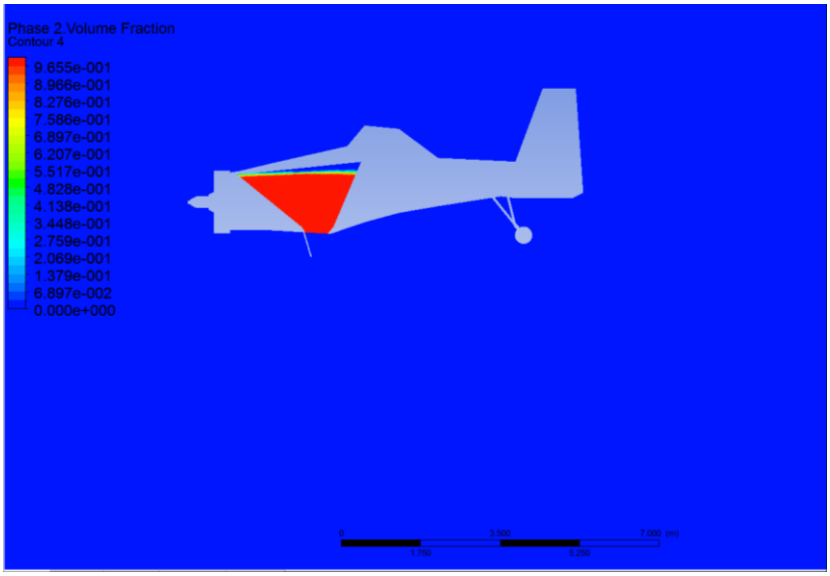


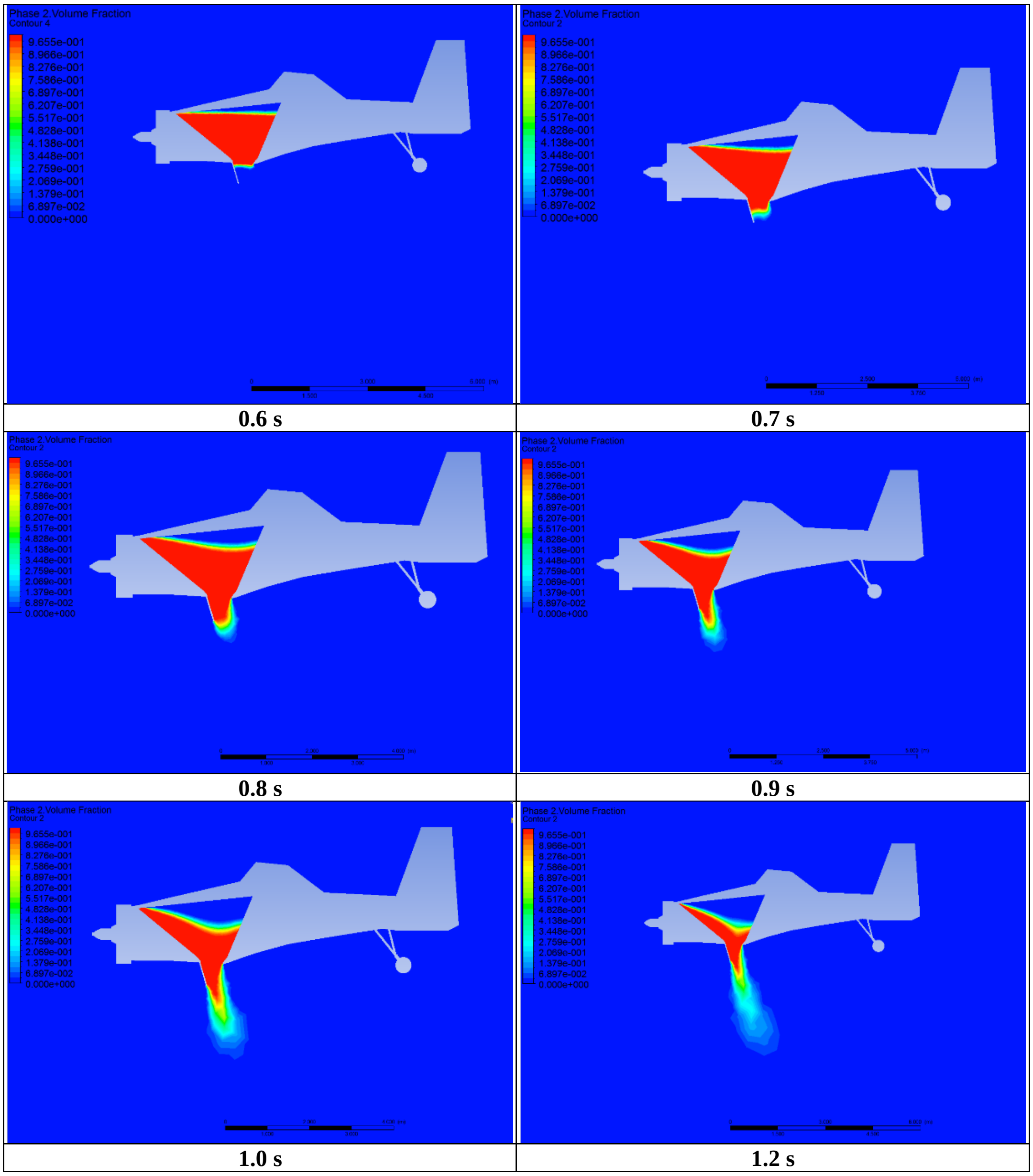
Fig. 6. The distribution of the volume fraction of water in the computational domain at *t* = 0.48 s

5. CALCULATION RESULTS AND ANALYSIS

As a result of the conducted calculations, the distribution of the volume fraction of water in the computational domain was obtained for the specified time points. The results are presented in Tab. 1.

A few tenths of a second after opening the water drop valve (from 0.6 to 1.0 seconds), the water begins to flow as an almost vertical dense jet. A slight deviation from the vertical direction is due to the horizontal forward motion of the aircraft and the interaction of the water jet with the air. After 1 second, the jet starts to disperse into droplets which then mix with the air, forming an aerosol with varied volume fraction values. At a distance of about 2 meters from the water drop valve, the water dispersion results in a volume fraction of approximately 40 percent. In the subsequent time steps, the length of the jet increases, and the degree of water dispersion in the air rises, leading to a decrease in the volume fraction values. After 2.4 seconds, the jet takes the shape of a long plume with a relatively low volume fraction value of a few percent. The end of the jet is approximately 3 meters above the ground.

Tab. 1. Variation of the volume fraction distribution of water in the computational domain over time



0.6 s

0.7 s

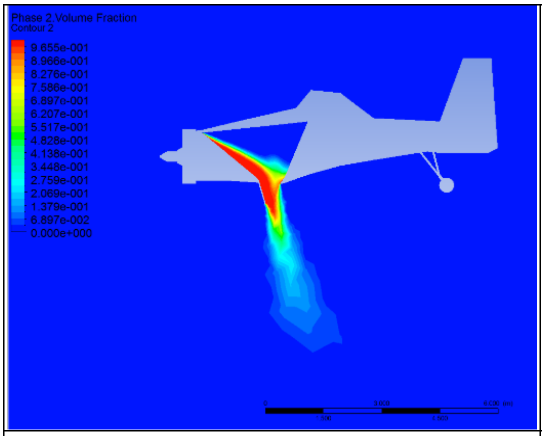
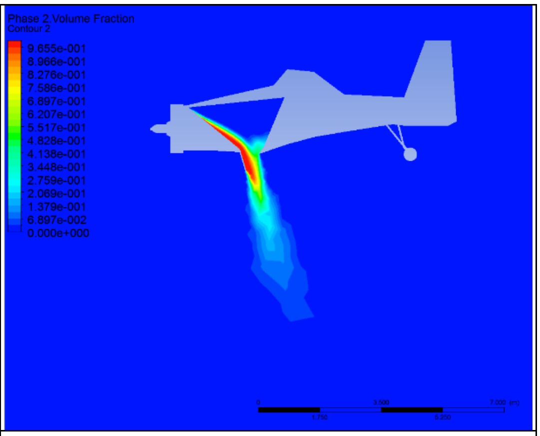
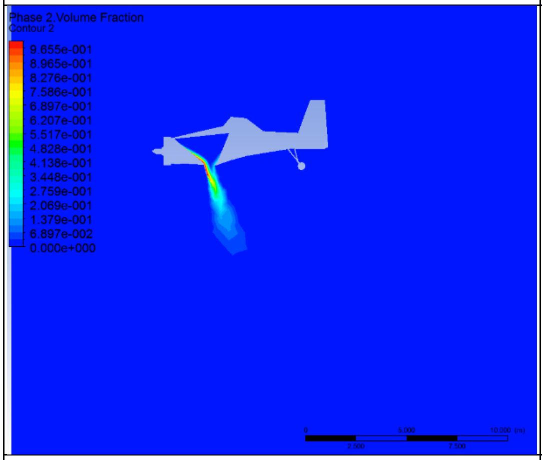
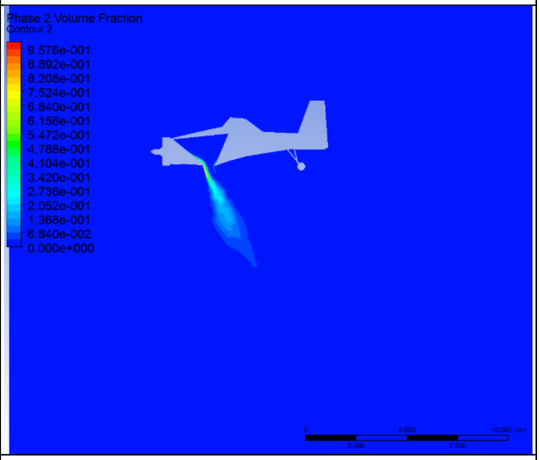
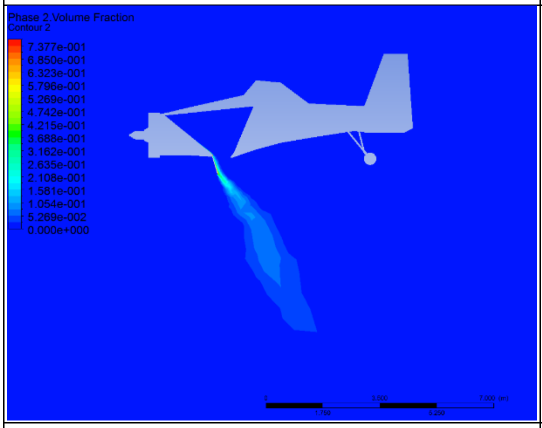
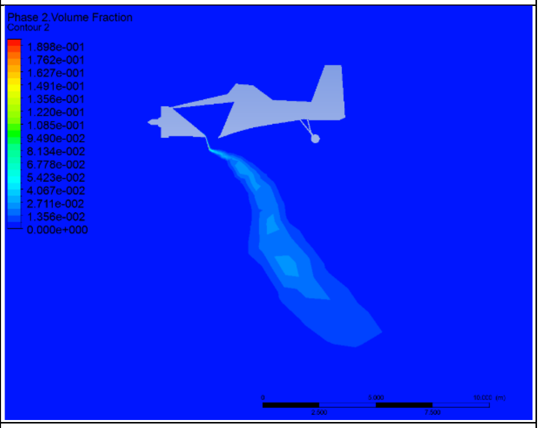
0.8 s

0.9 s

1.0 s

1.2 s

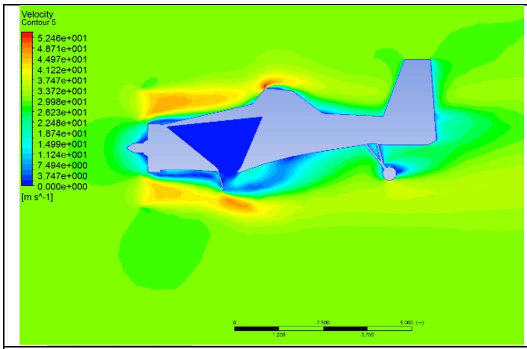
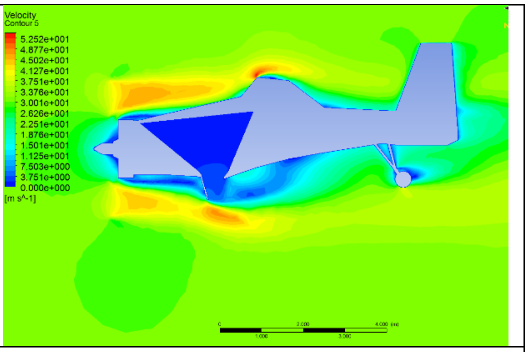
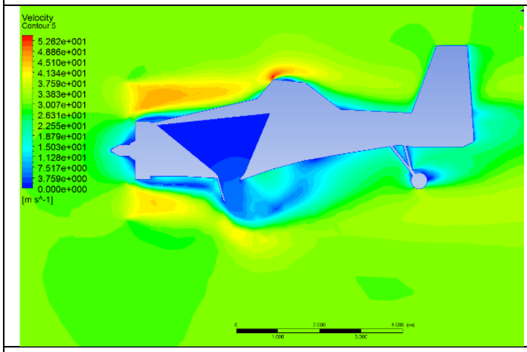
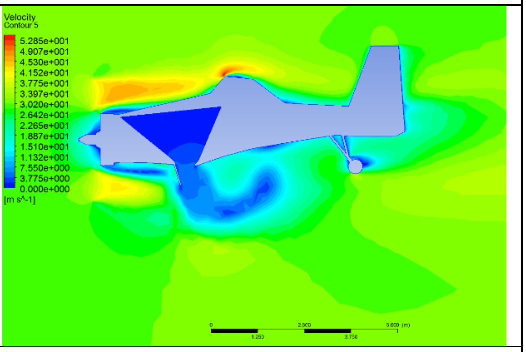
Tab. 1. Variation of the volume fraction distribution of water in the computational domain over time (continued)

	
1.4 s	**1.6 s**
	
1.8 s	**2.0 s**
	
2.2 s	**2.4 s**

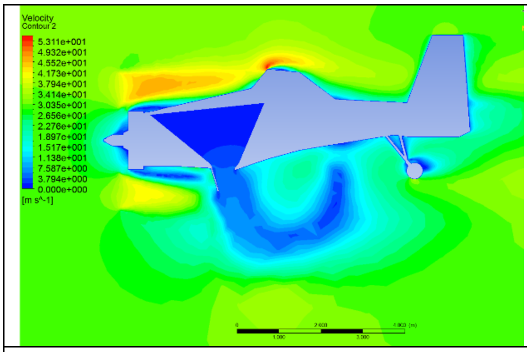
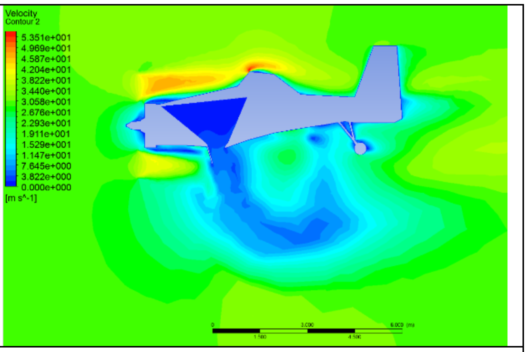
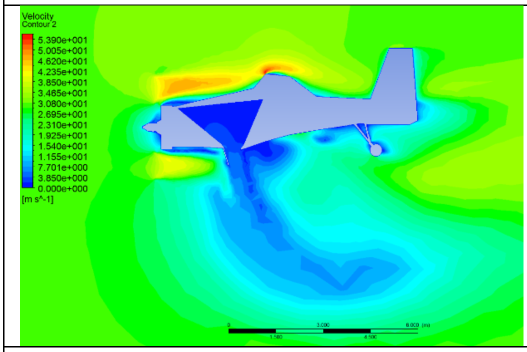
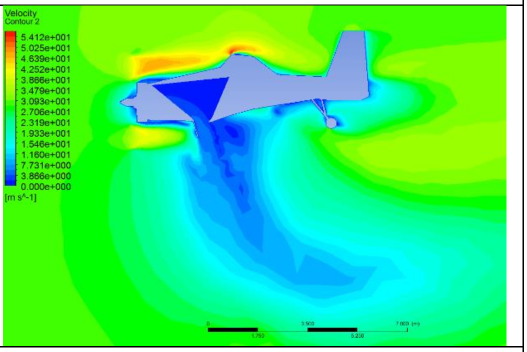
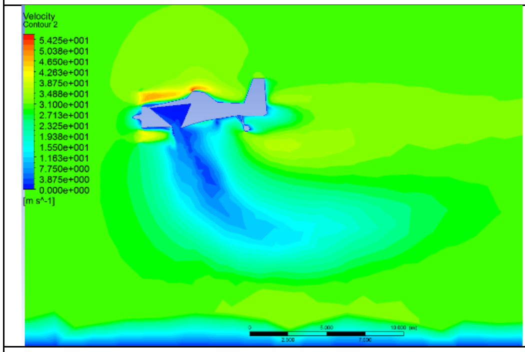
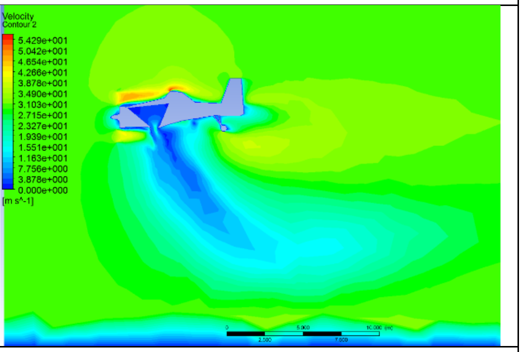
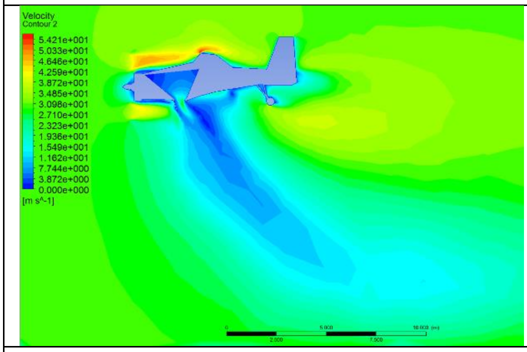
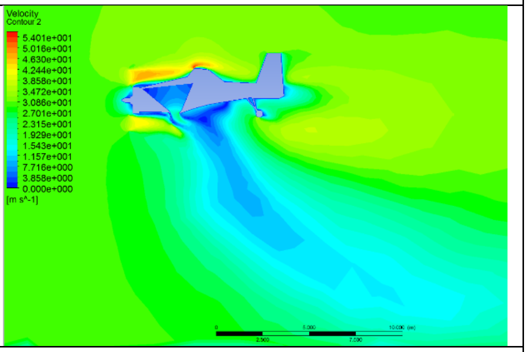
Additionally, the conducted numerical calculations allowed obtaining the distribution of the flow velocity in the computational domain for the specified time points (Tab. 2).

It should be emphasized that water flowing from the tank located in the front part of the aircraft can negatively impact its aerodynamics. As a result of the outflow of the formed liquid jet, the airflow velocity in the lower part of the fuselage decreases. Consequently, the pressure distribution in this area changes, which may lead to a reduction in generated lift and an increase in aerodynamic drag due to the local turbulence of the passing air. Water flowing gravitationally from the tank has an initial velocity close to zero. Then, due to gravity and the aircraft's motion relative to the air, its velocity increases to several meters per second. The resulting swirling causes the falling jet to bend upwards towards the lower surface of the fuselage (1.0-1.2 s). Subsequently, the swirling diminishes, and the jet once again directs vertically downwards. After 2 s, the outermost part of the jet, especially its outer surface, reaches a speed of up to 20 m/s. The flow velocity in the core of the jet is the lowest, averaging a few meters per second. It reaches its minimum value, close to zero, near the lower part of the aircraft's fuselage.

Tab. 2. Change in the velocity distribution of the fluid flow in the computational domain over time

	
0.6 s	**0.7 s**
	
0.8 s	**0.9 s**

Tab. 2. Change in the velocity distribution of the fluid flow in the computational domain over time (continued)

	
1.0 s	**1.2 s**
	
1.4 s	**1.6 s**
	
1.8 s	**2.0 s**
	
2.2 s	**2.4 s**

Additionally, Fig. 7 and Fig. 8 present the boundary layer where air is no longer mixed with water at the simulation time *t* = 2.6 s. The maximum width of the water drop at this moment reaches up to 20 m. The analysis of the boundary layer in the symmetry plane showed that the zone of water-air mixing near the ground begins at a horizontal distance of about 4.5 m from the water drop valve.

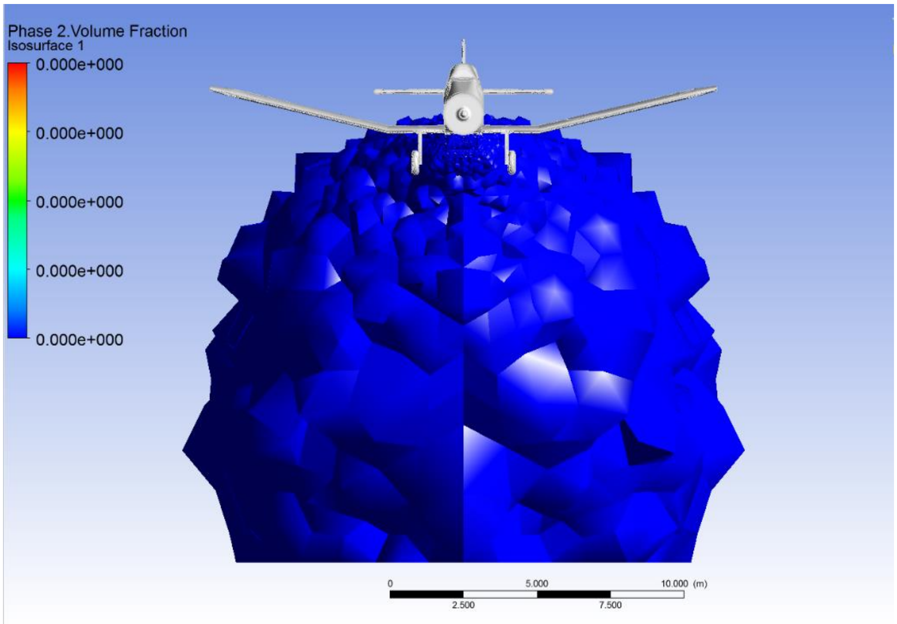


Fig. 7. Observed boundary layer during the final phase of water drop at *t* = 2.6 s – front view of the aircraft

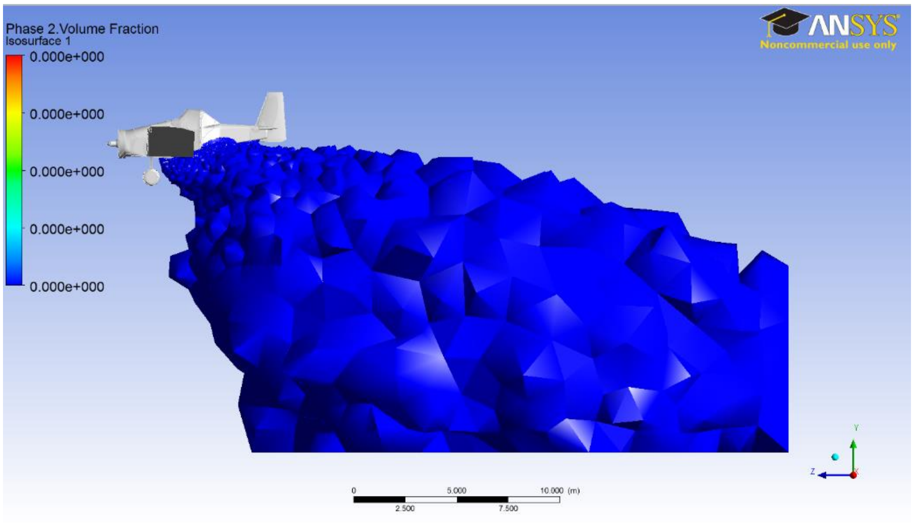


Fig. 8. Observed boundary layer during the final phase of water drop at *t* = 2.6 s – side view of the aircraft

6. CONCLUSION

The research conducted allowed a preliminary determination of the firefighting process by the M18B Dromader aircraft from a height of 15 m. This issue is particularly important due to the fact that such an aircraft is commonly used for firefighting by state services in Poland and other countries. The issue of waterbombing forest and grassland fires is still relevant, especially in the context of the ongoing climate change and heat waves affecting some regions of the world, including large and difficult to access areas of Southern Europe, Central Asia and North America.

The presented numerical analysis of water drop from the PZL M18 Dromader aircraft confirms the proper functioning of the developed model. It allows for changing the basic flight parameters such as flight altitude and speed. It is considered that the main objective of the study, i.e. a numerical analysis of water drop from the firefighting aircraft, has been achieved. Furthermore, the created model of the aircraft is suitable not only for the analysis presented in this study but also for flow analysis when the aerodynamic properties of the entire aircraft or its parts are examined. As a result of the analysis, the water drop time was determined to be 1.5 s, at a minimum aircraft speed of 30.3 m/s. The drop was performed from a height of 15 m, and the tank was filled with 2 078 dm^3 of water.

The results of the calculations made it possible to analyze the evolution of the water drop stream over time. In general, the behavior of the stream was mostly impacted by gravity and the motion of the aircraft relative to the air. The aerodynamic interactions make the speed of the water stream increase to several meters per second. The maximum jet velocity that was obtained was 20 m/s, and this velocity occurred at the surface of the stream about 2 seconds after the water discharge. It was observed that the flow velocity in the core of the stream was the lowest and on average was a few meters per second.

The results made it possible to assess the evolution of the stream in consecutive seconds after the water discharge and to analyze the decay of the stream into a cloud of droplets. Consequently, it is possible to identify the size and location of the area between the aircraft and the ground where water mixes with air.

In addition, the calculations made it possible to determine the width of the boundary layer at which air does not mix with water and the horizontal distance of the mixing zone from the tank flap. These quantities equaled 20 and 4.5 meters, respectively.

The changes in the weight of the aircraft due to water discharge or the level of water in tanks do not affect the numerical analysis performed. Of course, as the amount of water in the tank decreases, the position of the center of gravity of the aircraft slightly changes. However, this change in position must be in accordance with the design assumptions of the aircraft. That is, it must proceed so that the stability of the aircraft is preserved in all typical states of flight, including altitude changes and maneuvers.

In the future, the created model can be used to research the impact of air temperature resulting from extinguished fire on the water drop process and the effect of water drop height on the extinguished area.

Conflicts of Interest

The authors have no conflicts of interest to declare.

REFERENCES

Ahlgren, L. (2020, September 3). Boeing 747 Supertanker - The World's Largest Fire-Fighting Plane. *Simple Flying*. https://simpleflying.com/boeing-747-supertanker/

Amorim, J. H. (2008). Numerical modelling of the aerial drop of products for forest firefighting. *Doctoral dissertation*, Universidade de Aveiro (Portugal).

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*(3), 384-393. https://doi.org/10.1071/WF09122

Amorim, J. H. (2011). Numerical modelling of the aerial drop of firefighting agents by fixed-wing aircraft. Part II: model validation. *International Journal of Wildland Fire, 20*(3), 394-406. https://doi.org/10.1071/WF09123

Amorim, J. H., Borrego, C., & Miranda, A. I. (2008). An operational dropping model towards efficient aerial firefighting. *WIT Transactions on Ecology and the Environment*, *119*, 51-60. https://doi.org/10.2495/FIVA080061

Ardil, C. (2023). aerial firefighting aircraft selection with standard fuzzy sets using multiple criteria group decision making analysis. *International Journal of Transport and Vehicle Engineering*, *17*(4), 136-145. https://publications.waset.org/10013052/pdf

Chaikalis, D., Evangeliou, N., Tzes, A., & Khorrami, F. (2022). Design, modelling, localization, and control for fire-fighting aerial vehicles. *2022 30th Mediterranean Conference on Control and Automation (MED)* (pp. 432-437). IEEE. https://doi.org/10.1109/MED54222.2022.9837053

Clifford, R. M. S., Hoermann, S., Marcadet, N., Oliver, H., Billinghurst, M., & Lindeman, R. W. (2018). Evaluating the effects of realistic communication disruptions in VR training for aerial firefighting. *2018 10th International conference on virtual worlds and games for serious applications (VS-Games)* (pp. 1-8). IEEE. https://doi.org/10.1109/VS-Games.2018.8493423

Clifford, R. M. S., Khan, H., Hoermann, S., Billinghurst, M., & Lindeman, R. W. (2018). The effect of immersive displays on situation awareness in virtual environments for aerial firefighting air attack supervisor training. *2018 IEEE Conference on Virtual Reality and 3D User Interfaces (VR)* (pp. 1-2). IEEE. https://doi.org/10.1109/VR.2018.8446139

Czyż, Z., & Karpiński, P. (2020). Aerodynamic characteristics of the X-tail stabilizer in a hybrid unmanned aircraft. *International Journal of Simulation Modelling*, *19*(4), 631-642. https://doi.org/10.2507/IJSIMM19-4-534

Czyż, Z., Karpiński, P., Skiba, K., & Wendeker, M. (2022). Measurements of Aerodynamic Performance of the Fuselage of a Hybrid Multi-Rotor Aircraft with Autorotation Capability. *International Review of Aerospace Engineering (IREASE)*, *15*(1), 12-23. https://doi.org/10.15866/irease.v15i1.21319

Goraj, Z., Frydrychewicz, A., Ransom, E. C. P., Self, A., & Wagstaff, P. (2001). Aerodynamic, dynamic and conceptual design of a fire-fighting aircraft. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, *215*(3), 125-146. https://doi.org/10.1243/0954410011533121

Han, Y., Liu, H., Tian, Y., Chen, Z., & Nie, Z. (2018). Virtual reality oriented modeling and simulation of water-dropping from helicopter. In *Proceedings of the 2018 International Conference on Artificial Intelligence and Virtual Reality* (pp. 24-29). https://doi.org/10.1145/3293663.3293669

Ito, T., Kato, H., Goda, Y., Tagawa, S., & Negishi, E. (2010). Water-dropping aerodynamics for fire-fighting amphibian. *27th International Congress of the Aeronautical Sciences (ICAS)* (pp. 1-10). http://icas.org/ICAS_ARCHIVE/ICAS2010/PAPERS/333.PDF

Kal'avský, P., Petríček, P., Kelemen, M., Rozenberg, R., Jevčák, J., Tomaško, R., & Mikula, B. (2019). The efficiency of aerial firefighting in varying flying conditions. *2019 International Conference on Military Technologies (ICMT)* (pp. 1-5). IEEE. https://doi.org/10.1109/MILTECHS.2019.8870050

Kliment, L. K., Rokhsaz, K., Nelson, J., Terning, B., & Weinstein, E. M. (2015). usage and flight loads analysis of king airs in aerial firefighting missions. *Journal of Aircraft*, *52*(3), 910-916. https://doi.org/10.2514/1.C032877

Konishi, T., Kikugawa, H., Iwata, Y., Koseki, H., Sagae, K., Ito, A., & Kato, K. (2008). Aerial firefighting against urban fire: Mock-up house experiments of fire suppression by helicopters. *Fire Safety Journal*, *43*(5), 363-375. https://doi.org/10.1016/j.firesaf.2007.10.005

Oleksiak, J. et al. (1975) Preliminary design of agricultural airplane M-18, PZL Mielec - Research and Development Center, Poland. https://pzlmielec.pl/en/company/company-profile-and-history

Qureshi, S., & Altman, A. (2018). Studying fluid breakup and dispersion to predict aerial firefighting ground drop patterns. *2018 AIAA Aerospace Sciences Meeting*, 1047. https://doi.org/10.2514/6.2018-1047

Satoh, K., Kuwahara, K., & Yang, K. T. (2004). A numerical study of forest fire progression and fire suppression by aerial fire fighting. *ASME International Mechanical Engineering Congress and Exposition (IMECE2004)* (pp. 79-86). https://doi.org/10.1115/IMECE2004-60679

Satoh, K., Maeda, I., Kuwahara, K., & Yang, K. (2005). A numerical study of water dump in aerial fire fighting. *Fire Safety Science – Proceedings of the Eighth International Symposium*, *8*, 777-787. https://publications.iafss.org/publications/fss/8/777/view/fss_8-777.pdf

Satoh, K., Sagae, K., Kuwahara, K., & Yang, K. T. (2000). Experiments and Numerical Simulations of Flow Patterns of Water Droplets From Fire-Fighting Helicopters. *ASME International Mechanical Engineering Congress and Exposition (IMECE2000)*, *5*(5-10), 57-64. https://doi.org/10.1115/IMECE2000-1560

Tsujimura, H., Kubota, K., & Sato, T. (2022). Numerical Analysis of Aerial Firefighting Using Grid–Particle Coupling Method. In *AIAA SCITECH 2022 Forum,* 0450. https://doi.org/10.2514/6.2022-0450

Varner, D., Kliment, L. K., & Rokhsaz, K. (2019). The aerodynamics of a civil transport in aerial firefighting. *AIAA Aviation 2019 Forum,* 3697. https://doi.org/10.2514/6.2019-3697

Wang, X., Liu, H., Tian, Y., Chen, Z., & Cai, Z. (2021). A fast optimization method of water-dropping scheme for fixed-wing firefighting aircraft. *IEEE Access*, *9,* 120815-120832. https://doi.org/10.1109/ACCESS.2021.3106538

Walton, B. (2018, July 30). Classic MD-87s Still Earning a Living as Aerial Firefighting Tankers. Avgeekery.com. https://avgeekery.com/classic-md-87s-still-earning-a-living-as-aerial-firefighting-tankers/

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 Congress of the International Council of the Aeronautic Sciences*, 1-10. https://www.icas.org/ICAS_ARCHIVE/ICAS2018/data/papers/ICAS2018_0474_paper.pdf

Zhou, T., Lu, J., Wu, C., & Lan, S. (2022). Numerical calculation and analysis of water dump distribution out of the belly tanks of firefighting helicopters. *Safety*, *8*(4), 69. https://doi.org/10.3390/safety8040069

Zohdi, T. I. (2021). A digital twin framework for machine learning optimization of aerial fire fighting and pilot safety. *Computer Methods in Applied Mechanics and Engineering, 373,* 113446. https://doi.org/10.1016/j.cma.2020.113446