Numerical simulation of splashing landing gear on an amphibious aircraft air cushion

A V Fevralskikh

Moscow Aviation Institute (National Research University), 4 Volokolamskoe shosse, Moscow 125993, Russia

a.fevralskih@gmail.com

Abstract. The paper presents the basics of the methodology for modeling the splatter formation of landing gear on an amphibious aircraft hovercraft based on the numerical solution of the Navier-Stokes equations for a two-phase incompressible flow with a free interface (VoF approach) and the disperse medium motion equations (DPM approach). The results of numerical simulation of the water jet collapse in the transverse air flow in comparison with the experimental results are given, as well as recommendations and results of application of numerical simulation in forecasting the air cushion splash formation. The results of the simulation are in good correspondence with the results of the experiments published earlier, which demonstrates the prospects of applying the technique in the development and design of new landing gear layout solutions on an amphibious air cushion aircraft.

1. Introduction

The advantages of hovercrafts are related to the properties of amphibious aircraft, which allow for takeoff and landing on water or snow surface, unequipped sandy shore or shallow water, as well as to approach the place of passengers' disembarkation, using the bearing properties of the air cushion. The capabilities of amphibious vehicles allow to consider them as an actual component of an off-road module to ensure the interconnection of the Russian Federation territory when forming complex scientific and technical projects [1]. At the same time, the operation of amphibious aircraft involves the risk of icing on takeoff, just as it is typical for hovercrafts, because the flow of air from under the air cushion fence provokes an intense spray. Icing of an amphibious aircraft can lead to changes in wing bearing properties, increased takeoff weight, damage to propellers and a number of other negative effects.

At the same time, the prediction of splashability and icing of an amphibious aircraft on takeoff is a nontrivial task, to solve which, in general, it is necessary to take into account the nature of the airflow and droplets from the air cushion, and the external aerodynamic flow of the fuselage, and the interaction of droplets with the wing surface and the fuselage, and the interaction of splashes with the external aerodynamic flow. In order to study the aerohydrodynamics of designed amphibians, large scale models are traditionally tested in hydro canals [2–4]. Despite a number of limitations associated with the effect of scaling, hydrochannel testing can determine with sufficient reliability for design purposes characteristics such as the relative resistance of an air cushion and the running differential at different Froude numbers. At the same time, the numerical simulation of aerohydrodynamics (CFD — computational fluid dynamics), based on the Navier-Stokes equations solution by reference volume
method, with the use of Euler's Volume of Fluid (VoF) approach, is increasingly being used for such tasks [5, 6]. CFD modeling tools have been implemented in CAD software products, which determines their availability and ease of use, and the validation results confirm the effectiveness of CFD for amphibian design tasks.

At the same time, the application of the control volume method together with VoF approach has such limitations as iterative process instability, high requirements to the quality and quantity of the computational grid elements (with the corresponding computational resource costs). To predict splash dynamics using the VoF approach, generally speaking, it is desirable to know in advance the approximate distribution of droplet sizes, on the basis of which you can set the initial distribution of control volume sizes and set up the grid model. These requirements encourage the development of new methods of numerical simulation of aero-hydrodynamics and spatter formation, which would combine such qualities as reliability and robustness, the possibility of application in automated design. For development of such methods it is supposed to consider Lagrangian and pseudo-Lagrangian approaches based on representation of droplets as discrete particles.

In the present work the possibilities of the computational method of aero-hydrodynamics simulation of VoF-to-DPM for the development of the method of numerical simulation of the air-cushion chassis splash formation are investigated. The VoF-to-DPM method is based on the Euler fluid representation with the possibility of converting its individual fragments into the Lagrangian Discrete Phase Method, i.e. it is a hybrid approach. The paper presents the basics of the method of modeling the spray formation of an air cushion using the DPM method, presents the results of numerical simulation of the water jet collapse in the transverse air flow in comparison with the results of experiments, as well as recommendations and results of the method in modeling the spray formation of an air cushion.

2. Fundamentals of DPM method
DPM (Discrete Phase Method) refers to the Euler-Lagrangian approaches, where the dynamics of a solid medium are modeled using the reference volume method using Navier-Stokes equations and the motion of discrete particles is described according to the law of pulse change (1).

$$m_p \frac{d \tilde{u}_p}{dt} = m_p \tilde{u} - \tilde{u}_p + \frac{m_p}{\tau_r} g(\rho_p - \rho) + \bar{F}. \quad (1)$$

In the equation (1): $m_p$ — particle mass, $\tilde{u}_p$ — particle velocity, $\tilde{u}$ — rate of flow, $g$ — free-fall acceleration, $\rho$ and $\rho_p$ — respectively the density of the solid medium and the density of the particle, $\bar{F}$ — additional forces influencing the motion of a particle, $\tau_r$ — time of relaxation (2).

$$\tau_r = \frac{\rho_p d_p^2}{18 \mu} \frac{24}{C_d Re}.$$

Relaxation time $\tau_r$ is determined by the molecular viscosity of the solid medium $\mu$, particle size $d_p$, particle resistance coefficient $C_d$ and Reynolds' number $Re$ by the velocity of the particle relatively solid medium.

Approach described by the equations (1) and (2) in general, it is quite universal, and can be complemented by relations describing the influence of particles on the heat transfer, turbulence and other characteristics of the solid medium — on the one hand, and the relations that determine the forces acting on the particle side of the solid medium — on the other hand. Separately, equations can be introduced to describe the dynamics of particles in terms of degrees of freedom of rotation. With this approach, it is assumed that spatter (droplets) have a spherical shape, the drop size does not
exceed 10% of the characteristic size of the reference volume, and the processes of flow on the surface of the drop do not have a noticeable effect on the motion characteristics. In this paper, to determine the resistance coefficient $C_d$ approach was taken Dynamic Drag Model (3), implemented in Ansys Fluent software, which allows to take into account the change in particle shape during drop destruction [7].

\[
C_{d,\text{sphere}} = \begin{cases} 
0.424 & \text{Re} > 1000, \\
24 \left(1 + \frac{1}{6} \text{Re}^{2/3}\right) & \text{Re} \leq 1000, 
\end{cases}
\]

\[
C_d = C_{d,\text{sphere}}(1 + 2.632y),
\]

\[
\frac{d^2 y}{dt^2} = \frac{C_F}{C_k} \frac{\rho_k}{\rho} \frac{u^2}{r^2} = \frac{C_k \sigma}{\rho r^2} y - \frac{C_d \mu}{\rho r^2} \frac{dy}{dt}.
\]

Model represented by a system of equations (3), is based on Taylor's analogy [8]. In the system of equations (3) $y$ — drop deformation value (deviations from the spherical shape), $r$ — nondeformant drop radius. Model factors $C_k = 1/2$, $C_F = 1/3$, $C_d = 5$, $C_k = 8$ are determined by the results of experiments.

Madabhushi model with Ansys Fluent settings offered by default is used for modeling decomposition of liquid fragments (Figure 1).

3. Simulation of a water jet collapse in a transverse air flow
The water jet spray in the transverse air flow is simulated (Figure 5). The characteristics of the modeled flow are given in Table 1.

![Figure 1](image1.png)

**Figure 1.** Settings of the decomposition model of liquid fragments in Ansys Fluent.

![Figure 2](image2.png)

**Figure 2.** Modeling the collapse of the water jet in the transverse air flow.

| Water Density $\rho_l$, kg/m³ | Air density $\rho_a$, kg/m³ | Weber Number $We$ | Dynamic pressure ratio $q = \frac{\rho_l V_l^2}{\rho_a V_a^2}$ | Jet diameter in injection area, mm | Surface tension factor $\sigma$, N/m |
|-----------------------------|--------------------------|-----------------|--------------------------|-------------------------------|--------------------------|
| 998.2                       | 1.225                    | 248             | 7.1                      | 0.381                         | 0.0719                   |
The joint turbulent flow of two incompressible liquids — air and water — with free interface of variable topology is investigated. It is accepted that turbulization of the aerodynamic flow caused by the appearance, destruction and movement of droplets under these flow conditions does not have a noticeable effect on the velocity distribution. Numerical modeling of control volumes using DPM approach was performed in Ansys Fluent software. A polyhedron mesh model with the number of control volumes of 30108 units was used, a turbulence model SST k-ω and a turbulence damping option were connected to close the system of Navier-Stokes averaged Reynolds equations [6]. Figure 3 shows the input (for air) and the lower boundary of the simulation area with a splash cloud obtained from the simulation results.

\[ \frac{y}{D} = 1.55q^{0.53} \ln \left( 1 + 1.66 \frac{z}{D} \right) \]

In relation (4) \( y \) — vertical co-ordinate of the particle, \( z \) — co-ordinate of the particle in the direction of aerodynamic flow, calculated from the point of injection (injection) of water jet, \( D \) — jet diameter at the injection point. The mean square deviation of the obtained correlation is 1.83 units.

The results of DPM numerical modeling performed in Ansys Fluent compared to the results of the experiment in the form of discrete phase coordinate distribution (spatter/droplets) are shown in the figure below 4.

![Figure 3](image-url)

**Figure 3.** Displaying the boundaries and results of spray modeling in Ansys Fluent.

At the input boundary of the modeling area, the boundary condition of aerodynamic flow velocity was set, at the output boundary — pressure condition, at the upper, lower and side walls — impermeability condition with slippage. Also, at the lower boundary is given a source of injection of spray — water as a discrete phase, according to the conditions given in Table 1.

The computation time was about 18 hours on a computer with 4 physical cores of 3.4 GHz Intel Core i7-6700 processor, 64 GB of RAM and an NVIDIA Quadro K620 graphics card.

The results of the simulation of the particle trajectory were compared with the correlation of the results of experimental studies [9–14].

![Figure 4](image-url)

**Figure 4.** Comparison of the results of DPM water jet decay simulation with the results of experiments.
The results of the numerical simulation are in good agreement with the results of the correlation of experimental data in the area of jet injection and for most DPM particles in the area of jet disintegration. However, the default settings of the Madabhushi model offered in Ansys Fluent seem to be subject to additional calibration to achieve a better match with the experimental results.

4. Numerical simulation of hovercraft spraying

Numerical simulation of the spray formation of the amphibious aircraft air cushion section, shown in Figure 5, was performed. The graphical image of the calculated area is presented in Figure 6.

![Figure 5. Hovercraft with chassis on a hover cushion on takeoff mode from water.](image)

![Figure 6. Scheme of the calculation area of air cushion section modeling.](image)

The results of modeling are given in Figure 7 in comparison with the results of experiments and calculations according to the empirical model [15] at different values of air flow rate and initial draft of the vessel. Quantitative analysis of modeling results is presented as a dependency of dimensionless volumetric spray rate \( \dot{Q}_\text{m} = q_\text{m} / Q_a \) (\( q_\text{m} \) — volumetric spray rate, m\(^3\)/s; \( Q_a \) — volumetric airflow rate, m\(^3\)/s) against the criterion \( U \) (5).

\[
U = \ln \left( 10^9 \frac{Q_a r_m}{H_n} \right),
\]

\[
\dot{Q}_\text{m} = \frac{Q_a}{H_n \sqrt{2P / \rho}}.
\]

In relation (5) \( H_n \) — the depth of the air cushion trough, m; \( P \) — average air cushion pressure, Pa; \( \rho \) — air density, kg/m\(^3\); \( r_m \) — average median drop radius, determined by the results of numerical modeling.

The computation time for each calculation case using the same computer as in the jet disintegration simulation was about 36 hours.
Figure 7. Results of numerical simulation of the air cushion spray using the VoF-to-DPM method: (a) — comparison of the values of dimensionless spray flow at different values of the criterion obtained during CFD modeling (round markers) with the data of experiments and empirical model; (b) — visualization of free surface and spray near the air cushion body.

The data in Figure 7 show that the application of VoF-to-DPM method in CFD modeling allows to reliably determine the amount of water carried as a spray from the air cushion area. In this case, there is no modeling is carried out in a full-size setting and, thus, it is possible to avoid the theoretical assumptions accepted when setting the model experiments in experimental pools. In addition, when performing this kind of calculations, it is not necessary to specify the depth of the trough, the pressure value and accept M.S. Volynsky's condition as the initial data, as proposed in the method of [15]. In addition, if sufficient computational resources are available, the application of this technique can be extended to the case of a complete aircraft layout, thus analyzing the interaction of spray with propellers, sprayability of the fuselage and other effects. However, despite the high level of automation of the described method of modeling spatter formation, its requirements to computational resources remain quite high, and in the future, it seems expedient to consider the use of meshless methods of modeling disperse hydrodynamics.

5. Conclusion
The basics of the method of numerical simulation of splash formation of landing gear on the air cushion of amphibious aircraft are described. The technique is based on the numerical solution of the equations of viscous turbulent flow of incompressible fluid by the method of control volumes together with the equations of motion of disperse medium. The VoF-to-DPM method used in this case, implemented in Ansys Fluent software, allows to take into account in one statement of the computational experiment the deformation of the free water surface and the dynamics of splashes represented by discrete particles. The results of the method verification in the course of numerical simulation of water jet disintegration in the air current demonstrate good agreement with the experimental results. The reliability of the results of the air cushion section modeling is confirmed by comparison with the results of the empirical model calculations. Among advantages of the stated technique it is necessary to carry high automatization, possibility of application together with software complexes of the automated designing, and also possibility to leave a number of theoretical assumptions and to apply technique in the decision of problems of interaction of full configuration of an aircraft with splashes.

References
[1] Pogosyan M A, Strelets D Y and Vladimirova V G 2019 Territorial Connectivity of the Russian Federation: From the Statement of Complex Problems to Drawing up Integrated Scientific and Technical Projects Herald of the Russian Academy of Sciences 89 (2) 179–84
[2] Fevralskikh A V 2017 Development of methods for designing the aero-hydrodynamic arrangement of an amphibious aircraft on an air cushion with aerodynamic unloading on the basis of numerical modeling Cand. (Nizhny Novgorod: R.E. Alekseev NGTU) 175 p
[3] Diachenko N V 2013 Spattering of amphibious air-cushion vessels Dissertation of Doctor of Technical Sciences (St. Petersburg State Maritime Technical University) 301 p
[4] Demeshko G V 1992 Design of ships. Amphibious FPVs Textbook Vol 1 (St. Petersburg: Shipbuilding) 269 p
[5] Kal'yasov P S, Fevralskikh A V and Shabarov V V 2014 Mathematical modeling of the amphibian vessel aero-hydrodynamics on an air cushion J. Problems of Strength and Plasticity 76 (3) 263–8
[6] Gramuzov E M and Fevralskikh A V 2018 Application of the turbulence damping method for calculation of the water surface shape in the task of determining the resistance of an air cushion vessel Sci. J. Marine intellectual technologies 4 (42) 242–9
[7] Liu A B, Mather D and Reitz R D 1993 Modeling the Effects of Drop Drag and Breakup on Fuel Sprays SAE Technical Paper 930072
[8] Taylor G I 1963 The Shape and Acceleration of a Drop in a High-Speed Air Stream Technical Report Scientific Papers ed G K Batchelor
[9] Tambe S, Jeng S-M, Mongia H and Hsiao G 2005 Liquid Jets in Subsonic Crossflow 43rd AIAA Aerospace Sciences Meeting and Exhibit. doi:10.2514/6.2005-731
[10] Becker J and Hassa C 2002 Breakup and Atomization of a Kerosene Jet in Crossflow at Elevated Pressure J. Atomization and Sprays 11 49–67
[11] Inamura T, Nagai N, Hirai T and Asano H 1991 Disintegration Phenomena of Metalized Slurry Fuel Jets in High Speed Air Stream Proc. of the 5th Int. Conf. on Liquid Atomization and Spray Systems Gaithersberg, MD USA p 839
[12] Chen T H, Smith C R and Schommer D G 1993 Multi-Zone Behavior of Transverse Liquid Jet in High-Speed Flow AIAA Paper 93–0453
[13] Wu P-K, Kirkendall K A, Fuller R P and Nejad A S 1997 Breakup Processes of Liquid Jets in Subsonic Crossflows J. of Propulsion and Power 13 (1) January-February 64–72
[14] Yates C L 1971 Liquid Injection into Supersonic Airstreams AIAA Paper 71–724
[15] Anosov V N and Diachenko N V 2009 Calculation of the amount of water carried into the atmosphere by an air jet from an air cushion Proc. of the Central Res. Inst. of Acad. A N Krylov 49 (333) p 123