Numerical simulation study on flow field distribution and load characteristics of trans-media aerial underwater vehicle during its water-exit process

Tan Junyi Tan, Junhua Hu, Yang Ge *, Guoming Chen, Qimin Yan
School of Aeronautics Engineering, Air Force Engineering University, Xi’an, China.

*Corresponding author e-mail: 1914385238@qq.com

Abstract. The oblique water-exit process of a trans-media aerial underwater vehicle is a highly non-linear, unsteady and dramatic process of flow field change. Based on the underwater configuration of a variable trans-media aerial underwater vehicle, the water-exit process of this configuration under typical operating conditions was simulated by CFD fluid simulation software, and the flow field characteristics and load distribution of the vehicle at different stages of the water-exit process were analyzed. The simulation results show that when the vehicle crosses the water-air interface obliquely, the phenomenon of liquid surface deformation and uplift, liquid surface attachment wrapping up the water-exit and wake dragging fracture will occur sequentially, which will lead to sudden change and continuous drastic change of the flow field around the vehicle, forming a backflow zone with different radius and direction on both sides. High-frequency positive and negative alternative oscillation of the body load of the vehicle wrapped by water produces pitching moment.

1. Introduction
Trans-media Aerial Underwater Vehicle (TMAUV) is a new type of vehicle that can autonomously and repeatedly cross the water-air interface and stabilize flight and navigation in the air and water[1][2]. It can be used as an offshore detector. It can quickly approach the target area by air flight, detect after entering the water, and return after detection. In the military aspect, this kind of vehicle can be used as a reconnaissance platform to enhance the system’s anti-information perception ability, or as an attack carrier platform to enhance the penetration ability of weapons and complete the attack task.

Vehicle water-exit is a very complex process, involving a series of complex physical problems, such as solid-liquid-gas coupling. In the process of water-exit, the liquid level will rise and deform, resulting in the phenomenon of water mound, and then break up accompanied by the generation, collapsing and splash of droplets[3]. These strong non-linear and unsteady phenomena will make the flow field around the vehicle change dramatically, and the surface pressure gradient will change abruptly, which puts forward higher requirements for the stability control of crossing the water-exit and the variant flight after the water-exit.

At present, the research on the characteristics of the water-exit process of an vehicle is generally focused on the flow field characteristics analysis and cavitation effect of the water-exit of a simple object (cylinder, sphere, etc.). Liao Jianhui et al. [4] used time-varying additional mass theory and finite element method to simulate the underwater vehicle water-exit process. Yao Xiongliang [5] analyzed the
unsteady flow field characteristics of a slender pointed cylinder when it came out of water by improving the second-order double asymptotic line method. Sun Shili et al. [6] considered the mathematical model of cylindrical water-exit movement based on free surface effect and analyzed the phenomenon of water mound. Hu Junhui et al. [7] used simulation software to simulate the whole process of water-exit with cavitation, and studied the formation process of supercavitation and its interaction with free surface. Zou Xing et al. [8] studied the flow field and cavitation evolution of the structure during its water-exit process. Peng Libing et al. [9] combined simulation with experiment, studied the law of the process of the slender body water-exit, and analyzed the relationship between the shape of the vehicle head and the law of cavitation formation. At present, most of the literatures focus on the study of the vertical water-exit characteristics of simple structures, and there is no numerical simulation study on the inclined water-exit process of a specific torpedo configuration at different pitch angles and attack angles. Under different conditions, the flow field characteristics and the load distribution of the body during the oblique water-exit process of a specific configuration are studied in this paper, which can provide theoretical support for a trans-media vehicle across the water-air interface steady and repeatedly.

In this paper, CFD fluid simulation software is used to simulate the oblique water-exit process of a two-dimensional underwater configuration of a trans-media aerial underwater vehicle. The gas-liquid two-phase distribution of the water-exit of the vehicle under typical operating conditions is studied. The instantaneous pressure, velocity and vector fields of the typical position of the water-exit process are compared. The flow field changes, regularity and vehicle load characteristics of the oblique water-exit process of the vehicle are analyzed, which provide theoretical support for the study of trans-media aerial underwater vehicle water-exit control law and the formulation of the optimal water-water-exit scheme.

2. Geometric configuration and simulation conditions

2.1. Geometric Configuration and Shape Parameters

![Figure 1. Geometric configuration of Trans-media Aerial Underwater Vehicle](image)

The geometric configuration of the trans-media aerial underwater vehicle is shown in Figure 1, (a) in the air, (b) in the underwater, (c) in the two-dimensional underwater configuration. In this paper, the underwater configuration is simplified appropriately, the rudder is omitted and the protuberance of the back of the wing is not considered.

2.2. Computational Domain and Simulation Conditions

FLUENT fluid simulation software was used to simulate the flow field. ICEM CFD software is used to draw computational domain grids. The free surface in computational domain and the grids around the vehicle are refined. The number of grids is about 80,000. The computational range is 50m *40m. The altitudes of water and air are 20m each. The head of the vehicle is 1m away from the water surface, as shown in Figure 2. The multiphase flow model based on the finite volume method is adopted. The k-ε turbulence model is adopted. The upper boundary of the computational domain is set as the boundary condition of pressure outlet, the other boundary conditions of the vehicle and the computational domain are set as the boundary condition of solid wall. The SIMPLEC algorithm is used to solve the control equation in the transient calculation process. The momentum, turbulent kinetic energy and dissipation rate are all solved by the second-order upwind discrete format, and the pressure interpolation method
use PRESTO discrete format. The dynamic mesh updating method uses elastic approximate smoothing algorithm and local reconstruction algorithm. Through C language, UDF is written to customize the water-exit velocity of the vehicle, and ultimately realize the water-exit of the vehicle.

![Figure 2. Computational Domain Diagram of water-exit process](image)

3. Flow field and load analysis of oblique water-exit process of TMAUV

3.1. Gas-liquid two-phase distribution

![Figure 3. Gas-liquid two-phase distribution](image)

Fig. 3 is a gas-liquid two-phase distribution map obtained by the oblique water-exit of the vehicle under the conditions of $\alpha = 0^\circ$, $\theta = 45^\circ$, $v = 10\text{m/s}$. A total of eight different moments were selected, with blue as the water and red as the air. The water-exit process can be roughly divided into three parts: liquid surface deformation ($t_2$, $t_3$), liquid bread wrapped water-exit ($t_4$, $t_5$, $t_6$, $t_7$) and wake fracture ($t_8$). From the two-phase diagram, the water surface of will gradually rise and deform in the process of the vehicle approaching the water surface. Due to inertia and viscous action, the liquid around the head has a certain speed and adheres to the surface of the vehicle, forming a thin liquid film to wrap the vehicle. With the constant speed of the water-exit vehicle, the pressure of the attached liquid on both sides decreases, and the viscous force is gradually less than the gravity of the water, resulting in the rupture of the liquid film on the vehicle, and sliding and splashing along the vehicle, which exposes the vehicle to the air. As the vehicle continues to rise, the liquid slips to the tail, forming a wake, which gradually minifies, and the middle part is concave, which breaks and completes the water-exit.

3.2. Variation of Velocity and Flow Field

In order to study the changing trend of liquid velocity and flow field around the vehicle in the process of water-exit, under the condition of 3.1 simulation, 12 instantaneous moments in the process of water-exit were selected to obtain streamline diagram of the flow field around the vehicle, as shown in Fig. 4.
From the change of streamline chart, when the vehicle is sailing near the surface of water, due to the head's squeezing effect on the liquid, the liquid velocity vector near it is radial, and a large recirculation zone is formed on both sides of the body. When the head is near the liquid level, the vortices near the junction of the body and the head are generated. The direction of the upper vortices is counterclockwise, while the direction of the lower vortices is clockwise. At the moment when the head is out of water, the highest point of the vehicle is higher than the far-field undisturbed free surface. The liquid wraps the head and the vehicle is about to carry on air. Because of the great difference of the physical properties of water and air, the velocity distribution is asymmetrical. The velocity distribution on both sides of water and air is very different, and there is a velocity discontinuity on the interface. The axial and radial radius of the side vortices on the fuselage increases rapidly, while the axial radius of the lower side vortices increases rapidly. After the head of the vehicle is out of water, the stable recirculation area near the head of the vehicle is destroyed. The lower side vortices of the vehicle are separated firstly, forming several vortices of different sizes and intensities. Then the upper part of the vehicle is separated and the standing vortices appear, which makes the flow field around the vehicle change dramatically. When the vehicle is completely out of the water, the two sides of it gradually restore a stable recirculation zone.

3.3. Fluid Force Distribution

When the vehicle is at constant velocity in the process of water-exit, the flow field around the transmedia aerial underwater vehicle will change dramatically because of the sudden change of the media. The change of the flow field will cause the change of the pressure on the surface of the vehicle, and even lead to the damage and breakage of the slender body. The pressure monitor can be used to monitor the fluid force acting on the surface of the vehicle. A total of 110 sets of data are obtained. All points are marked on the coordinate axis and fitted into a curve. The inertia spindle of the vehicle is taken as the horizontal axis and the direction is directed to the head, indicating the position of the surface of the vehicle. The longitudinal axis is perpendicular to the x-axis in the longitudinal symmetry plane, indicating the fluid force, and the direction is oblique upward and positive.

Fig. 5 chooses 12 instantaneous pressure nephograms and fluid force curves of the same 12 instantaneous moments as 3.2 to study the force law on the surface of the vehicle. Above is the pressure nephograms, below is the corresponding force curve of the vehicle wall.
Figure 5. Force curve of flow field in water-exit process

From the overall trend of change, there is always a high pressure area in the middle of the tail of the vehicle during the process of water-exit, and there is a minimum pressure at the junction between the tail and the fuselage. Its value varies with the location of the outlet water level. In underwater navigation, due to the hydrostatic effect, there is a pressure difference on the surface, because the selected points on the upper and lower surfaces of the fuselage have different water depths. The position of the upper surface is shallow, the pressure is smaller, the position of the lower surface is deeper and the pressure is larger. When crossing the water-air interface, due to the sudden change of media, the generation and breakdown of liquid film, the splash of droplets and other reasons, the force on the upper and lower surfaces of the fuselage produces oscillations of different magnitudes, and the phenomenon that the pressure on the upper surface is greater than that on the lower surface appears locally. The part of the vehicle out of water is subjected to the pressure of the air media and restores to a stable pressure difference. The pressure on the upper surface is greater than that on the lower surface.

When the vehicle navigates underwater ($t_1 = 0.16s$), the pressure on the upper surface of the tail is greater than that on the lower surface, and the peak difference is smaller, and the pressure difference between the upper and lower surfaces of the fuselage remains stable. When it approaches the water-air interface gradually ($t_2 = 0.28s$), the peak pressure difference at the tail increases gradually, and the pressure difference between the upper and lower surfaces of the fuselage decreases, and there is a positive and negative alternation. In the process of the vehicle head water-exit ($t_3 = 0.36s$, $t_4 = 0.44s$), when the body head has exceeded the far-field undisturbed water surface, at the moment of liquid film wrapping and breaking, the tail pressure peak difference continues to increase, the upper surface pressure of the fuselage is greater than the lower surface, the pressure difference of the fuselage changes dramatically. At the junction of the fuselage and the head, there is a minimum pressure, a negative pressure on the lower surface and a pitching moment. Because the liquid level on the upper side of the head rises more than that on the lower side, the pressure drop is moderated. In the process of fuselage water-exit ($t_5 = 0.50s$, $T_6 = 0.54s$, $T_7 = 0.60s$, $T_8 = 0.66s$, $T_9 = 0.70s$), the peak pressure difference at the tail is always large. Due to the interference of liquid attachment in fuselage section, the pressure difference appears positive and negative alternation of different magnitudes, and the air flow field at the water-exit part has been developed steadily. When the tail of the vehicle is higher than the far-field undisturbed surface ($t_{10} = 0.76s$, $T_{11} = 0.80s$, $T_{12} = 0.90s$), due to the attachment of the liquid wake to the tail, the pressure on the upper and lower surface of the tail oscillates and two peaks appear, and then
the difference of the peak pressure gradually decreases. When the vehicle is completely out of water, the force restores to stability.

In summary, the force on the surface of the vehicle varies dramatically during the process of water exit, and there will be different magnitude and positive and negative oscillations at different stages, which will produce additional moments, which will have a serious negative impact on the stability of the vehicle and the structure of the body.

4. Conclusion
Based on CFD fluid simulation technology, UDF is used to compile user-defined functions to control the motion of the vehicle, and the oblique water-exit process of the vehicle under typical operating conditions is analyzed. Through simulation comparison, the flow field variation law and the vehicle load characteristics in the oblique water-exit process of the vehicle are analyzed. The following conclusions are drawn: when the vehicle crosses the water-air interface obliquely, the phenomenon of liquid surface deformation and uplift, liquid surface attachment wrapping up the vehicle and wake dragging fracture will occur sequentially, which will lead to sudden change and continuous drastic change of the flow field around the vehicle, forming a backflow zone with different radius and direction on both sides. The high frequency positive and negative oscillations occur in the part of the body loaded by water, resulting in pitching moment.

References
[1] Qi D, Feng J F, Liu A, et al. Overview on the development and key technologies of water-air UAV[C]/IEEE Conference on Control, Automation and Robotics (ICCAR). Hong Kong: IEEE, 2016, 213-218.
[2] Hu J H, Xu B W, Feng J F, et al. Research on its water-exit and take-off process for morphing unmanned submersible aerial vehicle [J]. China Ocean Engineering, 2017, 31(2): 202-209.
[3] Wu Q G. Study on the Deformation of Free Surface and the Variation of Fluid Loading During Water Exit of a Revolving Body[D]. Harbin: Harbin Engineering University, 2017.
[4] Liao J H, You X C, Lv H B, et al. Development of a time-varying added mass method in the simulations of the water-exit process of underwater vehicles [J]. Engineering Mechanics, 2012, 29 (4): 202-209,223.
[5] Yao X L, Zhang C, Yang H, et al. Numerical calculation method for the unsteady flow field induced by missiles flying out of the water surface [J]. Journal of Harbin Engineering University, 2012, 33 (6): 677-683.
[6] Sun S L, Sun Y L, Hu J Z, et al. Free Surface Effect and Spike of a Cylinder Piercing Water Surface [J]. Chinese Journal of Computational Physics, 2013, 30 (2): 187-193.
[7] Hu J H. The research of hydrodynamic mechanism of formation collapse and interaction between free surface of surface of supercavitation in the process of water exit of a vertically launched underwater vehicle [D]. Zhe Jiang: Zhejiang Sci-Tech University, 2013.
[8] Zou X. Numerical Simulation of the Water Exiting Process of Structure [D]. Da Lian: Dalian University of Technology, 2013.
[9] Peng L B. Experimental and numerical study on hydrodynamic characteristics in water exit of slender body [D]. Zhe Jiang: Zhejiang Sci-Tech University, 2014.