Numerical Simulation of Cavitation and Ventilation Phenomena During Aircraft Ditching

CHEN Xingyi[1], *LI Meng*[1], *TONG Mingbo*[1*], *WU Bin*[1,2], *ZHANG Xianzheng*[3]

1. College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing 210016, P.R.China;
2. China Special Vehicle Research Institute, Jingmen 448000, P.R.China;
3. AVIC Jiangxi Hongdu Aviation Industry Group Company Ltd., Nanchang 330000, P.R.China

(Received 15 April 2022; revised 20 June 2022; accepted 10 July 2022)

Abstract: The large eddy simulation (LES) method combined with the global dynamic grid strategy is used to numerically study the influence of pitch angles on cavitation and ventilation phenomena in the rear of fuselage during ditching. The volume of fluid (VOF) method is used to capture the interface. By analyzing the normal force, moment, pressure distribution and nephograms, we find that the pitch angle has a great influence on the total load, the cavitation occurrence time and its transition to ventilation, thus affects the longitudinal load distribution and the controlling of aircraft dynamics.

Key words: aircraft ditching; cavitation; large eddy simulation (LES); ventilation; volume of fluid (VOF)

CLC number: V211.3 **Document code:** A **Article ID:** 1005-1120(2022)S-0009-07

0 Introduction

Aircraft ditching is an emergency procedure consisting in a controlled landing on water. Different from impacting on the ground, a ditching event involves various complex fluid-structure interaction dynamics, leading to phenomena like cavitation and ventilation. Cavitation is a phase change from liquid to vapor as a result of a sharp drop of pressure in liquid to or its vapor pressure. When cavitation occurs, a low pressure zone forms in the rear part of the fuselage, resulting in air sucked in cavity and the pressure back up to atmospheric levels, causing ventilation. According to the results of some model tests, for a ditching event with forward velocity, cavitation and ventilation may have great influence on aircraft dynamics by strongly affecting the load and pressure distribution[1].

Theoretical analysis on water impact were first carried out by Karman et al.[2] and Wagner et al.[3] and has been supplemented and developed subsequently. Experimentally, due to the aircraft dimensions and high horizontal velocities, scaled model tests are often conducted for aircraft ditching. The UK Company Cape Engineering performed the ditching tests on the 1∶8 scaled model of EADS-CASA CN235-300M[4]. Some speculations based on the results suggest that cavitation and ventilation may also be significant for aircraft ditching. However, scaled model tests on the full aircraft do not properly reproduce cavitation and ventilation phenomena that might take place at the rear, and therefore do not accurately reproduce the bottom fuselage pressures and kinematics of the fuselage during the impact phase. With the development of computer technology and numerical simulation methods, the tendency is to use computational tools to simulate the hydrodynamics of ditching. Climent et al. used the smoothed particle hydrodynamics (SPH) method to simulate the ditching of EADS-CASA CN235-300M aircraft[4]. It is pointed out that the suction force (negative pressure) at the rear fuselage is important during ditching. Zhang et al. used general coupling method to calculate the attitude

---

*Corresponding author, E-mail address: tongw@nuaa.edu.cn.

How to cite this article: CHEN Xingyi, LI Meng, TONG Mingbo, et al. Numerical simulation of cavitation and ventilation phenomena during aircraft ditching[J]. Transactions of Nanjing University of Aeronautics and Astronautics, 2022, 39(S): 9-15.
http://dx.doi.org/10.16356/j.1005-1120.2022.S.002

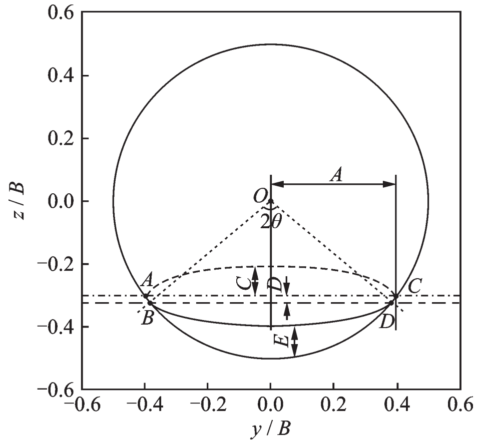
and pressure of large civil aircraft ditching with taking suction force into account[5]. It is concluded that the suction force greatly affects the attitude of plane ditching and that the air model is indispensable for suction force generation. There are also other similar simulations on fluid-structure interaction in the rear fuselage during ditching[6-7]. However, due to the highly nonlinear fluid-structure interaction dynamics during ditching, numerical models available at this time can hardly take all complex effects into account, thus causing errors, low capture accuracy of flow phenomena and high demand of computational resources.

In order to investigate the cavitation and ventilation modalities during ditching and overcome the limits imposed by scaling effects as well as providing valid experimental data for numerical simulations, tests on a double curvature specimen are conducted by Iafrati et al. by using high speed ditching facility (HSDF) at the Institute of Marine Engineering of the Consiglio Nazionale delle Ricerche (CNR-INM)[1].

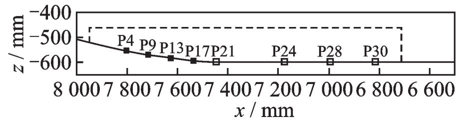
The large eddy simulation (LES) method coupled with global dynamic mesh strategy as well as the volume of fluid (VOF) method is applied to simulate the cavitation and ventilation phenomena during the double curvature shaped body ditching. By analyzing loading and pressure distribution as well as nephograms, the influence of pitch angles on cavitation and ventilation phenomena, and further on aircraft dynamics during ditching is presented.

1 Rear Fuselage Model and Test Condition

Numerical simulation is carried out based on the tests on a double curvature specimen impacting on water conducted by Iafrati et al.[1]. The double curvature specimen is a rear portion of a fuselage shape described by analytical functions[8-9], which has a circular-elliptical cross section shown in Fig.1(a) that is typical of cargo aircrafts. The longitudinal section of the specimen is shown in Fig.1(b). The fuselage portion used in the tests is that with $|y| \leqslant 330$ mm and $6\,710$ mm $\leqslant x \leqslant 7\,950$ mm.



(a) Circular-elliptical cross section



(b) Longitudinal section of the specimen

Fig.1 Schematic plot of specimen sections

The specimen is equipped with a total of 30 pressure probes, as shown in Fig.2. In addition to pressures, the total forces in the longitudinal and normal directions are measured as well. The normal component is measured at the rear and at the front to estimate the center of loads.

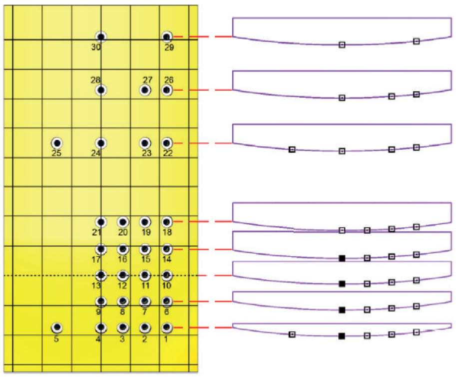


Fig.2 Position of pressure probes

The vertical to horizontal velocity ratio V/U can be achieved in the range of 0.03—0.05. The pitch angle α of the specimen can be varied from 4° to 10° while the maximum horizontal velocity is 47 m/s. The velocity of the specimen is seen as constant during ditching.

More details about the fuselage shapes and the high-speed ditching facility are provided in Ref.[1].

2 Numerical Model and Computational Methods

2.1 Numerical model

The governing equations are established based on the continuity and momentum equations for the three-dimensional homogeneous unsteady multiphase incompressible Navier-Stokes system. The finite volume method (FVM) is used to discretize the governing equations. The second-order implicit temporal discretization is chosen and the convection term is discretized with bounded central difference scheme.

The LES approach with WALE (Wall-adapting local eddy-viscosity) subgrid-scale (SGS) model is adopted in the simulation. The mass transfer cavitation model used in this paper was developed by Schnerr et al.[10]. The model is based on a reduced Rayleigh-Plesset equation[11] and neglects the influence of bubble growth acceleration, viscous effects and surface tension effects.

The VOF method is used to capture the interface and the global dynamic mesh strategy is used to simulate the rear portion of the fuselage ditching.

2.2 Mesh generation strategy

Structured hexahedral meshes are used to simulate the flow field. As shown in Fig.3, refined boundary layer meshes are generated near the wall of the rear fuselage to accurately capture the flow information of the boundary layer, ensuring $y^+ = u_* y/v \approx 1$, where y is the thickness of the first cell from the rear fuselage surface, u_* the wall frictional velocity and v the kinematic viscosity of the fluid. Local mesh refinement is performed near the rear fuselage and near the water surface in order to capture the multiphase flow phenomena accurately. The adaptive mesh refinement model is adopted as well.

The dimensions of the computational domain are 5.8 m×4.0 m×2.0 m. The total number of cells is more than 16 760 000. The time step is set to 2×10^{-4} s and adaptive time step model is used with the minimum time step is 5×10^{-5} s.

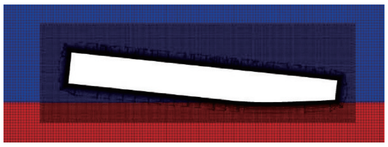


Fig.3 Structural meshes

2.3 Boundary conditions

The boundary conditions are shown in Fig.4. A no-slip boundary condition is imposed on the rear fuselage surface, velocity inlet boundary conditions are imposed on the front and bottom boundaries of the domain and pressure outlet boundary conditions are set at the rest of the boundaries.

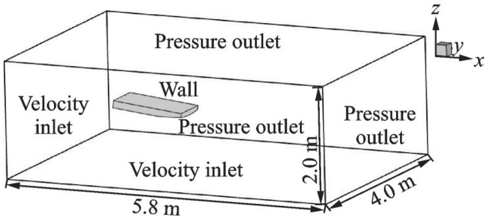
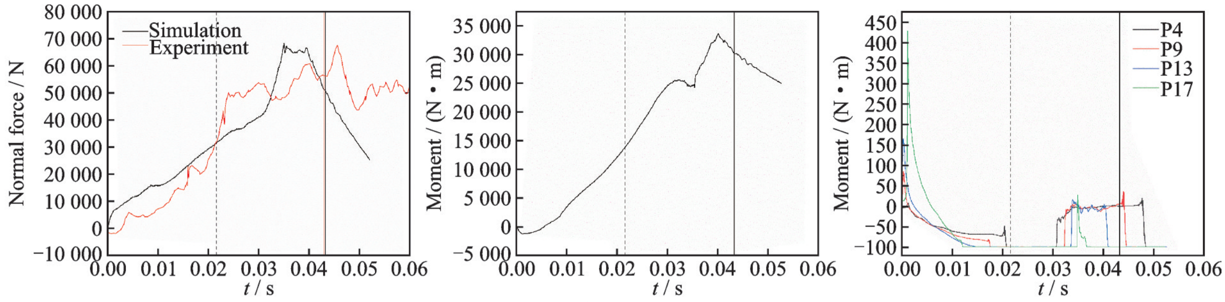


Fig.4 Boundary conditions

3 Numerical Results and Analysis

The LES method coupled with global dynamic mesh strategy as well as the VOF method is applied to simulate the cavitation and ventilation phenomena during the double curvature shaped body ditching. The simulations are performed at V/U ratio of 0.037 5, horizontal velocity of 45.2 m/s and pitch angles of 4°, 6° and 8°. In the simulation conditions, the condition that $\alpha = 6°$ is chosen to be compared with the experimental results and then the role played by pitch angles on cavitation and ventilation phenomena is studied. The moment when the specimen first touches the water surface is considered as t=0 while the time when the spray root reaches the leading edge is seen as the end of the impact phase.

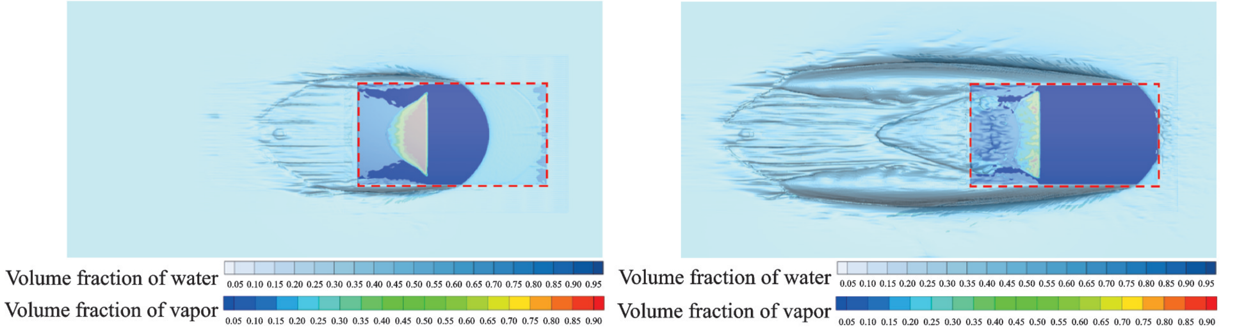
The solid black vertical lines and red lines in Fig.5(a,b,c) indicate the end of the impact phase in the simulation and in the test, respectively. The dashed vertical black lines and red lines denote the middle of the impact phase in the simulation and in



(a) Time histories of total force exerted normal to rear fuselage

(b) Time histories of force moment about the midpoint of the midline on upper surface of rear fuselage

(c) Time histories of relative pressure at positions where probes P4, P9, P13, and P17 are located



(d) Phase distribution on bottom surface of rear fuselage and water surface deformation in middle of impact phase

(e) Phase distribution on bottom surface of rear fuselage and water surface deformation at end of impact phase

Fig.5 Time histories of measured parameters and schematic plot of phase distribution when $\alpha = 6°$

the test, respectively. The edges of the rear fuselage model are marked by the red dashed lines in Fig.5(d,e).

The comparison of the total force exerted by the fluid normal to the specimen between test and simulation at $\alpha = 6°$ is shown in Fig.5(a). The normal force generally shows an increasing trend and the maximum value of the normal force in the simulation is in good agreement with that in the test while it appears earlier in the simulation than in the test. This is related to velocity reduction in the test and as a result, it costs more time for the spray root to get to the leading edge. The force moment about the midpoint of the midline on the upper surface of the specimen is measured as well in the simulation. As is shown in Fig.5(b), the specimen is generally subjected to a nose-up pitching moment, which exhibits an overall rising trend but drops a little at about t=0.032 5 s when the cavitation ends and ventilation begins, indicating a backward movement of the center of loads while the overall trend is forward. Similarly, a sudden increase in the force at the rear is observed in the test when ventilation enters into play, leading to a sharp rise of the total force[1], which is also observed in the simulation at t=0.032 5 s. Overall, the numerical simulation results agree well with the test results.

The time histories of the pressure measured at the positions where the pressure probes P4, P9, P13 and P17 are located are shown in Fig.5(c). At the first contact with the water surface, pressure at the positions where probes P17, P13 and P9 are located rises up sharply and the peak value at P17 is the largest, which is consistent with the test results. Subsequently, a pressure reduction shows at all probe locations, with P17 first dropping to the saturated vapor pressure, then P13, P9, and finally P4. The delay of the pressure drops at different probe locations indicates that the cavitation bubble needs some time to get to different probe locations.

With cavitation ends and ventilation begins, pressure values at all probe locations rise up to the ambient pressure, which corresponds to the time histories of the normal force and the force moment. Different from the time history of the pressure measured at different probes in the test, a new cavitation bubble is developed in the simulation, which is probably due to the velocity reduction in the test.

The phase and pressure nephograms at the bottom of the double curvature shaped rear fuselage at when $\alpha = 6°$ is shown in Fig.6. The curves correspond to the time histories of the pressure and the total loading.

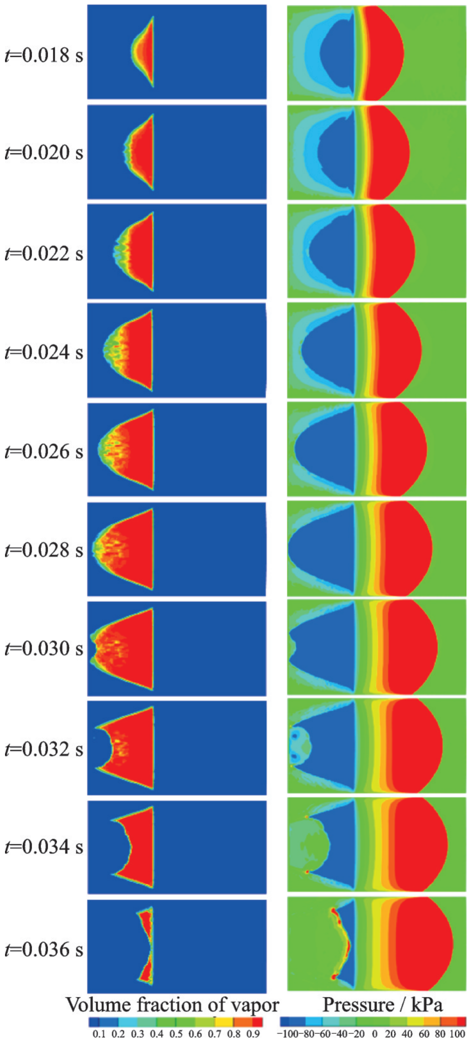


Fig.6 Phase and pressure distribution at bottom of rear fuselage when $\alpha = 6°$

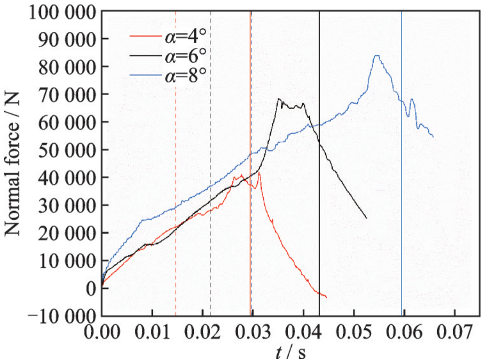


Fig.7 Time histories of total force exerted normal to rear fuselage at different pitch angles

The time histories of the total force exerted by the fluid normal to the rear fuselage at different pitch angles is shown in Fig.7. A sudden rise of the normal force when cavitation ends and ventilation begins is also observed in the cases of $\alpha = 4°$ and 8°. The normal force reaches its peak first at $\alpha = 4°$ as the vertical distance from the leading edge of the specimen to the water surface is the shortest in this case and as a result the least time is needed for the spray root to reach the leading edge, reducing the time duration of the impact phase. The peak value of the normal force increases with increasing the pitch angle. The time histories of the force moment about the midpoint of the midline on the upper surface of the specimen for the cases at different pitch angles are shown in Fig.8. The trend of the time histories of the force moment at $\alpha = 4°$ and 8° is similar to that at $\alpha = 6°$. The peak value of the force moment increases with the increase of the pitch angle and the peak comes earlier at smaller pitch angles.

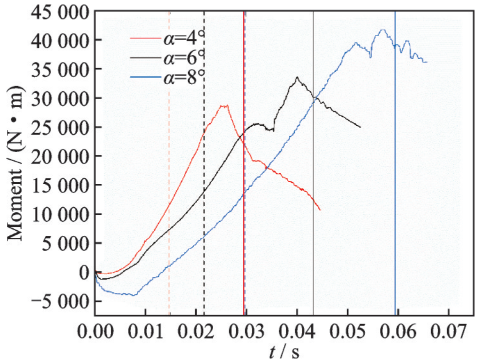
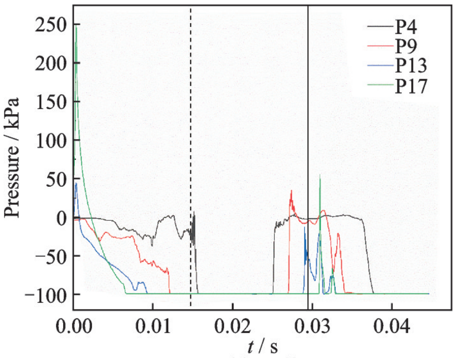
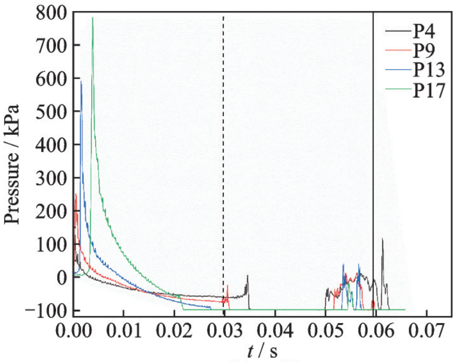


Fig.8 Time histories of force moment about midpoint of midline on upper surface of rear fuselage at different pitch angles

The time histories of the pressures for the cases at pitch angles of 4° and 8° are shown in Fig.9. By comparing the time histories of the pressures for the different cases, it is seen that the cavitation occurs earlier and the time duration of the cavitating phase shrinks with reducing the pitch angle. As a result,



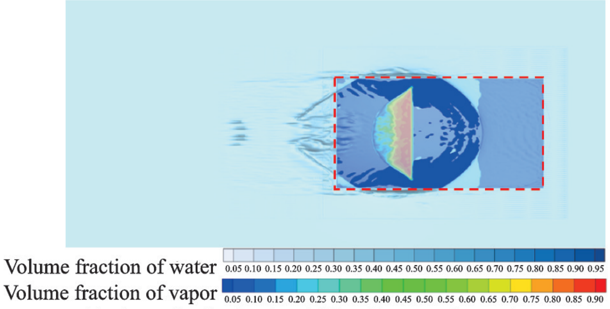
(a) α=4°



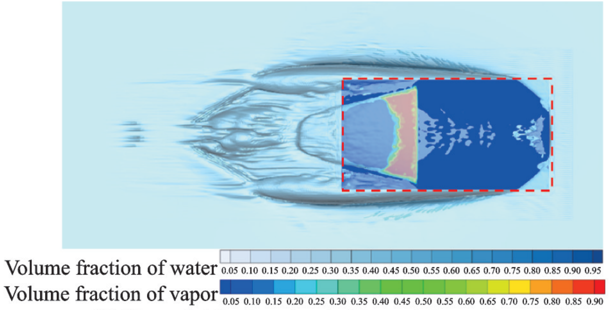
(b) α=8°

Fig.9 Time histories of relative pressure at positions where probes P4, P9, P13, and P17 are located

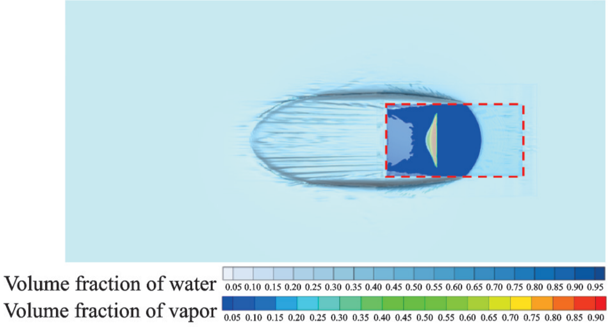
the occurrence of ventilation is also earlier in the impact phase at small pitch angles. Also, the pressure values at the same pressure probe location are bigger at larger pitch angles. The corresponding phase nephograms of the cases at $\alpha = 4°$ and 8° in the middle and at the end of the impact phase are shown in Fig.10.



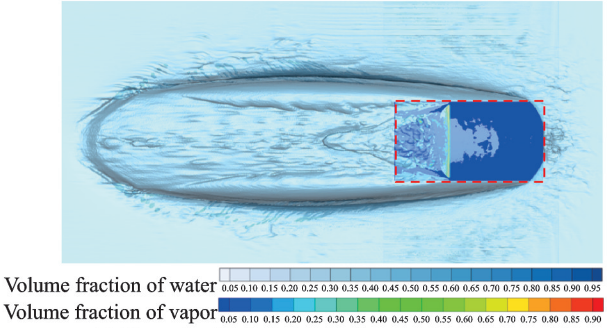
(a) Phase distribution in middle of impact phase when α=4°



(b) Phase distribution at end of impact phase when α=4°



(c) Phase distribution in middle of impact phase when α=8°



(d) Phase distribution at end of impact phase when α=8°

Fig.10 Phase distribution on bottom surface of rear fuselage and water surface deformation

4 Conclusions

The cavitation and ventilation phenomena taking place in the rear part of the fuselage during ditching have been numerically investigated and a good agreement is achieved between numerical and experimental results. The influence of pitch angles on cavitation and ventilation phenomena, and further on aircraft dynamics during ditching is studied as well by analyzing loading and pressure distribution as well as nephograms. The main conclusions are:

(1) When cavitation occurs, pressure will drop to the vapor pressure and there is a delay of pressure reduction at different pressure probe locations as the cavitation bubble needs some time to propagate. Then ventilation starts and the pressure rises to the ambient level.

(2) Cavitation and ventilation affect the total force exerted normal to the rear fuselage and its distribution. When cavitation ends and ventilation enters into play, there is a sharp rise of the total force and a sudden drop of the nose-up pitching moment, indicating a backward movement of the center of loads. The transition from cavitation to ventilation affects the longitudinal distribution of loading that governs the aircraft dynamics.

(3) The pitch angle has influence on the total force exerted normal to the rear fuselage and the force moment. The normal force reaches its peak earlier at smaller pitch angles due to shorter distance from the leading edge to the water surface and the peak value increases with the increase of the pitch angle. The trend of the force moment is similar to that of the normal force.

(4) The pitch angle affects the occurrence time of cavitation and the time duration of the cavitating phase. To be specific, cavitation occurs earlier and its duration is shorter at smaller pitch angles, causing the transition from cavitation to ventilation earlier in the impact phase.

References

[1] IAFRATI A, GRIZZI S. Cavitation and ventilation modalities during ditching[J]. Physics of Fluids, 2019, 31(5): 1-14.

[2] KARMAN T H. The impact on seaplane floats during landing: NACA-TN-321[R]. Washington D C, USA:NACA,1929.

[3] WAGNER H. Phenomena associated with impacts and sliding on liquid surfaces[J]. Journal of Applied Mathematics and Mechanics, 1932, 12: 193-215.

[4] CLIMENT H, BENITEZ L, ROSICH F, et al. Aircraft ditching numerical simulation[C]//Proceedings of the 25th International Congress of the Aeronautical Sciences. Hamburg, Germany:[s.n], 2006.

[5] ZHANG T, LI S, DAI H. The suction force effect analysis of large civil aircraft ditching[J]. Science China Technological Sciences, 2012, 55: 2789-2797.

[6] TOSO N S. Contribution to the modelling and simulation of aircraft structures impacting on water[D]. Stuttgart: University Stuttgart, 2009.

[7] WANG C, WU Z. Water entry numerical study of a simplified aircraft model in ditching[J]. Advances in Aeronautical Science and Engineering, 2016, 7(1): 51-61.

[8] IAFRATI A, OLIVIERI F. Definition of the conditions for guided aircraft ditching tests[C]//Proceedings of the SARAH Deliverable D5.1. Rome, Italy: [s.n.],2017.

[9] IAFRATI A, GRIZZI S, OLIVIERI F. Experimental investigation of fluid-structure interaction phenomena during aircraft ditching: AIAA-2019-2030[R]. [S.l.]:AIAA, 2019.

[10] SCHNERR G H, SAUER J. Physical and numerical modeling of unsteady cavitation dynamics[C]//Proceedings of the 4th International Conference on Multiphase Flow. New Orleans, USA:[s.n.], 2001.

[11] PLESSET M S, PROSPERETTI A. Bubble dynamics and cavitation[J]. Annual Review of Fluid Mechanics, 1977, 9: 145-185.

Authors Ms. **CHEN Xingyi** received the B.S. degree and the M.S. degree in aircraft design from Nanjing University of Aeronautics and Astronautics in 2019 and 2022, respectively. Her research interests include hydrodynamics and multiphase flow.

Prof. **TONG Mingbo** received his Ph.D. degree in solid mechanics from Beijing University and Aeronautics and Astronautics in 1995. His research interest includes fatigue, fracture and damage tolerance studies, hydrodynamic studies, design and analysis of composite materials, and CAD technology and software development.

Author contributions Ms. **CHEN Xingyi** set up and debugged the working conditions, conducted the analysis and wrote the manuscript. Mr. **LI Meng** helped to postprocess the working conditions. Prof. **TONG Mingbo** contributed to the overall planning and proofreading of the manuscript. Mr. **WU Bin** helped to conduct literature research on domestic and foreign research status. Mr. **ZHANG Xianzheng** helped to summarize the numerical methods and principles. All authors commented on the manuscript draft and approved the submission.

Competing interests The authors declare no competing interests.

(Production Editor:SUN Jing)

飞行器水上迫降空化和通风效应数值分析

陈星伊[1],李 勐[1],童明波[1],吴 彬[1,2],张宪政[3]

(1.南京航空航天大学航空学院,南京210016,中国;2.中国特种飞行器研究所高速水动力航空科技重点实验室,荆门448000,中国;3.江西洪都航空工业集团有限责任公司,南昌330000,中国)

摘要:采用大涡模拟(Large eddy simulation, LES)方法,结合整体动网格技术,模拟飞行器水上迫降过程中,俯仰角对机身后部出现的空化和通风现象的影响。采用流体体积法(Volume of fluid, VOF)捕捉两相交界面。对法向力、力矩、压力分布情况和云图进行分析发现,俯仰角对后机身段所受总载荷、空化现象出现的时间以及空化向通风阶段的转变存在重要影响,进而影响机体所受载荷的纵向分布,从而对飞机动力学产生影响。

关键词:水上迫降;空化效应;大涡模拟;通风效应;流体体积法