



HAL
open science

Advanced numerical simulation of oblique hydrodynamic impacts of geometries representative of aeronautical structures

Mathieu Goron, Bertrand Langrand, Nicolas Jacques, Alain Tassin, Thomas Fourest

► To cite this version:

Mathieu Goron, Bertrand Langrand, Nicolas Jacques, Alain Tassin, Thomas Fourest. Advanced numerical simulation of oblique hydrodynamic impacts of geometries representative of aeronautical structures. Aerospace Structural Impact Dynamics International Conference 2023, Jun 2023, WICHITA, United States. hal-04645323

HAL Id: hal-04645323

<https://hal.science/hal-04645323v1>

Submitted on 11 Jul 2024

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Advanced numerical simulation of oblique hydrodynamic impacts of geometries representative of aeronautical structures

M. Goron^{a,c}, B. Langrand^{a,b}, N. Jacques^c, A. Tassin^d, T. Fourest^a

a. DMAS, ONERA, F-59014 Lille - France

b. *Univ. Polytechnique Hauts-de-France*, LAMIH, UMR CNRS 8201, Valenciennes, F-59313, France

c. ENSTA Bretagne, UMR CNRS 6027, IRDL, 29806 Brest CEDEX 09, France

d. Ifremer, RDT, F-29280 Plouzané, France

Key words: Ditching, fluid-structure interaction, numerical simulation, coupled Eulerian-Lagrangian, suction, cavitation, ventilation

Abstract:

This communication deals with the emergency water landing of an aircraft (ditching). During an aircraft ditching, the structure experiences critical loads and eventually fails. The appropriate design allows minimizing the risks of occupants' injuries. Therefore, the behaviour of an aircraft in a ditching situation has been thoroughly studied and demonstrated using scale model tests, comparison with existing designs, analytical investigations and numerical simulations. See [Toso, 2009] for an extensive review of these different approaches. The recent progress in computation capabilities led to the increased use of numerical simulations in the certification process of aircraft. However, modelling such complex hydrodynamic impacts is challenging, even with state-of-the-art numerical approaches. For example, during oblique and high-velocity hydrodynamic impacts, complex hydrodynamic phenomena tend to develop. Among these phenomena, we can cite air cushion, suction forces, cavitation, ventilation, and aeration. These phenomena result from the interaction between the structure and the fluids, they occur simultaneously and influence each other. The objective of this work is to demonstrate the capability of the present numerical method to model hydrodynamic impacts in realistic ditching conditions and involving suction forces, cavitation, and ventilation.

In the present work, numerical simulations are carried out using the explicit Finite Elements solver Radioss, an Eulerian formulation for the fluid and a Lagrangian formulation for the structure. The fluid-structure interaction is based on an immersed contact interface (penalty method). This study is based on a series of experiments performed at the High-Speed Ditching Facility of the CNR-INM (see Figure 1), in Rome, in the framework of the European project SARA (Safety And Robust certification for ditching of Aircraft and Helicopters). These experiments, described in [Iafrafi and Grizzi, 2019], consist of oblique hydrodynamic impacts of a double curvature fuselage section (specimen P3 or SP3) at different horizontal velocities. The results presented here concern three horizontal impact velocities: $U = \{21, 34.5, 45.2\}$ m/s, with a constant ratio between the horizontal velocity and the vertical velocity: $V/U = 0.0375$, and with the SP3 tilted at 6° . The rigid body velocity is considered constant during the impact.

The numerical results are compared to the experimental results in terms of hydrodynamic force and pressure. The hydrodynamic force exerted by the fluid on the whole structure is presented in terms of non-dimensional force $f = \frac{F_z}{\rho_w U^2 S_{el}}$, where F_z is the vertical hydrodynamic force, ρ_w is the water volumetric mass, U is the horizontal velocity of the SP3, and S_{el} is the characteristic elements' area the SP3. The hydrodynamic pressure measured at gauge $p9$ is presented in terms of relative pressure, $Pr_9 = P_9 - P^0$, where P_9 is the pressure obtained at the gauge $p9$ and P^0 is the atmospheric pressure. The $p9$ sensor is located at the rear of the SP3 along the longitudinal plane of symmetry (see Figure 1b). The evolution of f and Pr_9 is presented Figure 2, as a function of the non-dimensional time, $t^* = \frac{Ut}{dH}$, where H is the height between the lowest point and the front edge of the SP3 (see Figure 1b). The time $t^* = 0$ corresponds to the beginning of the impact phase, *i.e.* the instant when the structure enters in contact with the water. The data stops at the end of the impact phase when the limit of the wetted surface reaches the front edge of the structure.

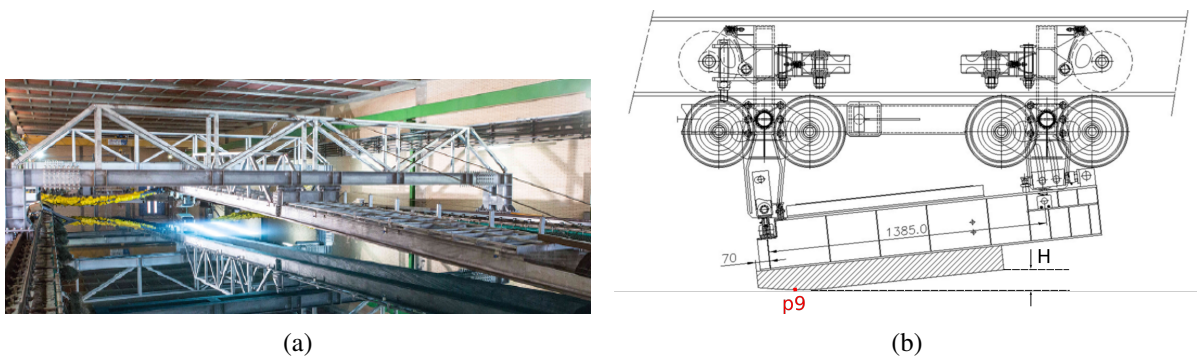


Figure 1: From [Iafrazi and Grizzi, 2019]: (a) *High-Speed Ditching facility*: the trolley moves along the guide suspended above the water. (b) Side view of the SP3 (hatched part), the acquisition box above it, and the trolley. The position of the pressure gauge $p9$ is specified and H corresponds to the vertical distance between the lowest point and the leading edge of the SP3.

Suction forces are measured at the rear of the SP3 for all impact speeds. For $U = 21$ m/s, the relative pressure Pr_9 is negative. Under the effect of suction, a cavitation bubble develops at the rear of the SP3 for the cases $U = 34.5$ m/s and $U = 45.2$ m/s during the entire duration of the impact. Due to cavitation, the relative pressure Pr_9 does not go under 0.1 MPa. This pressure level corresponds to the relative saturation vapour pressure at an ambient temperature of approximately 20°C. At the maximum impact velocity, $U = 45.2$ m/s, the cavitation bubble expands until it reaches the back of the SP3 and opens to the open air. The pressure inside the cavitation bubble thus increases until it reaches the relative atmospheric pressure $Pr_9 = 0$ MPa, at around $t^* = 12$: this is the ventilation phenomenon. An increase in the total hydrodynamic force is also observed at this moment. Indeed, the increase in pressure leads to a decrease in the suction forces exerted at the rear of the SP3, and thus to an increase in the total force.

The evolution of the hydrodynamic force and pressure seems to be satisfactorily predicted by the numerical model. The ventilation phenomenon does not seem to be captured well by the numerical model. Indeed, for $U = 45.2$ m/s, the relative pressure $Pr_9 \simeq -0.1$ MPa from $t^* = 7$ until the end of the simulation.

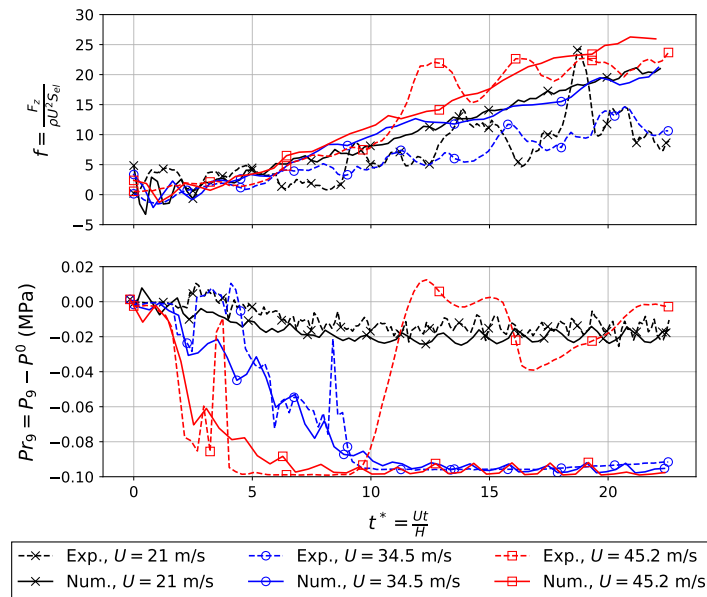


Figure 2: SP3 oblique hydrodynamic impact for $U = \{21, 34.5, 45.2, \}$ m/s. Evolution of (top) the non-dimensional hydrodynamic force f and (bottom) the relative pressure at gauge p_9 , Pr_9 . Both physical values are given depending on the non-dimensional time t^* .

These encouraging results show that the proposed numerical method is promising for the study of hydrodynamic impacts in conditions representative of an aircraft ditching. Although the numerical results are consistent with the experimental results, the ability of the numerical method to model a cavitation phenomenon is not yet proven. The pressure variations observed previously could be related to flow separation at the back of the structure due to the high impact speeds. This hypothesis seems to be supported by the fact that the model is not able to model the ventilation phenomenon in this case. Numerically, the relative pressure at p_9 does not go under 0.1 MPa. This is probably due to the fact that the air phase is modelled using a perfect gas equation, which does not allow for negative pressures. Sensitivity studies are underway to verify these assumptions.

References

- [Iafrati and Grizzi, 2019] Iafrati, A. and Grizzi, S. (2019). Cavitation and ventilation modalities during ditching. *Physics of Fluids*, 31(5):052101. Publisher: American Institute of Physics.
- [Toso, 2009] Toso, N. R. S. (2009). *Contribution to the modelling and simulation of aircraft structures impacting on water*. PhD thesis, Institute of Aircraft Design, Universität Stuttgart. Accepted: 2009-12-18.