ELSEVIER

# Contents lists available at ScienceDirect

# Journal of Fluids and Structures



journal homepage: www.elsevier.com/locate/jfs

# Numerical simulation of the oblique water impact of double curvature bodies involving suction and cavitation phenomena

M. Goron <sup>a,b,c,\*</sup>, A. Tassin <sup>c</sup>, B. Langrand <sup>a</sup>, E. Spinosa <sup>d</sup>, A. Del Buono <sup>d</sup>, N. Jacques <sup>b</sup>, T. Fourest <sup>a</sup>, A. Iafrati <sup>d</sup>

<sup>a</sup> DMAS, ONERA, Lille, F-59014, France

<sup>b</sup> ENSTA Bretagne, UMR CNRS 6027, IRDL, Brest CEDEX 09, F-29806, France

<sup>c</sup> Ifremer, RDT, Plouzané, F-29280, France

<sup>d</sup> CNR-INM, Roma, Via di Vallerano 139, 00128, Italy

# ARTICLE INFO

Keywords: Numerical simulation Fluid–structure interaction Coupled Eulerian–Lagrangian approach Aircraft ditching Suction Cavitation

# ABSTRACT

The present study aims to assess the capability of a numerical method to model hydrodynamic impacts representative of an aircraft ditching. The considered numerical method is based on the Finite Element explicit solver *Radioss* and a Coupled Eulerian–Lagrangian approach. The fluid–structure interaction is dealt with using an immersed contact interface and a penalty coupling method. The oblique water impacts of three different fuselage sections have been studied based on the experimental campaigns carried out during the European project SARAH at the High-Speed Ditching Facility of CNR-INM in Rome, Italy. The results are presented in terms of force coefficient, local relative pressure, and free surface elevation. The effect of the coupling stiffness and size of the fluid elements on the numerical results is analysed to assess the robustness of the numerical method. The numerical method shows a satisfying capability to reproduce most of the experimental results. Particular attention is given to the capability of the numerical method to describe the suction and cavitation phenomena. The effect of the specimens' transversal cross-section, the specimens' longitudinal curvature and the development of cavitation phenomenon on the hydrodynamic loads are also investigated.

# 1. Introduction

An aircraft ditching is a planned emergency procedure to land an aircraft on water. At the moment of impact, the aircraft experiences potentially critical hydrodynamic loads due to its high impact velocity. From the manufacturer and regulation administration point of view, the main objective during such an event is to maximise the survivability of the passengers and crew members (see, for instance, EASA (2021)). For these reasons, the ditching problem and hydrodynamic impacts in general have been widely studied in the past and are still studied nowadays.

Recently, advanced numerical approaches have been widely used to address water impact problems due to their capability to consider complex impact conditions (*e.g.* structural and flow behaviour), sometimes at the cost of high computation times. For instance, the ditching simulation presented by Jackson and Putnam (2020) took approximately 79 days to run. These advanced numerical approaches might rely on explicit or implicit solvers. The structure is often modelled using the Finite Element Method (FEM) with a Lagrangian formulation. The fluid flow can be modelled using various numerical methods, such as the FEM (with a Lagrangian, Eulerian or Arbitrary Lagrangian–Eulerian formulation), the Finite Volume Method (FVM), or meshless methods such

\* Corresponding author at: DMAS, ONERA, Lille, F-59014, France. *E-mail address:* mathieugoron@yahoo.fr (M. Goron).

https://doi.org/10.1016/j.jfluidstructs.2025.104322

Received 16 September 2024; Received in revised form 9 March 2025; Accepted 2 April 2025

Available online 18 April 2025

<sup>0889-9746/© 2025</sup> The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC license (http://creativecommons.org/licenses/by-nc/4.0/).



Fig. 1. Illustration of the hydrodynamic phenomena occurring during an aircraft ditching. Modified from Langrand and Siemann (2018).

as Smoothed Particle Hydrodynamics (SPH), to cite a few. The coupling of the fluid and structure models is of first importance and can rely on different numerical schemes, such as penalty methods or Lagrangian multipliers. For more details on coupling methods, the reader can refer to the work of Souli and Sigrist (2010), Casadei et al. (2011) and Valsamos et al. (2015).

A challenging aspect of modelling this type of water impact is linked to the complex hydrodynamic phenomena potentially occurring during ditching due to the interaction between the structure and the fluids (air and water), such as suction (McBride and Fisher, 1953), cavitation (Brennen, 1995) and ventilation (Judge et al., 2004). These phenomena are illustrated in Fig. 1. In a ditching context, the suction phenomenon corresponds to the development of pressures lower than the atmospheric pressure, named hereafter negative relative pressures, at the rear of the aircraft fuselage. The horizontal velocity of the structure and the longitudinal curvature of the fuselage in the contact zone can highly influence the magnitude of this phenomenon. The suction phenomenon, in addition to the overpressure developing further forward of the structure during ditching, can lead to a pitch-up motion of the aircraft and increase the damage sustained by the structure during the subsequent pitch-down induced water entry. In some cases, suction can lead to cavitation, *i.e.* the development of a gaseous phase within the water when the pressure reaches the vapour pressure. At ambient temperature, the relative vapour pressure is  $P_{vap} = P_{vap} - P^0 \in [-0.0993; -0.0986]$  MPa, where  $P^0$  is the atmospheric pressure, set to 0.101325 MPa. The pressure drop can also lead to a flow detachment and air being sucked below the fuselage in this area. This phenomenon is called ventilation.

These hydrodynamic phenomena can highly influence the ditching process, as outlined in studies by Climent et al. (2006), Toso (2009), and Langrand and Siemann (2018). Therefore, recent efforts have been dedicated to studying these phenomena during impacts that are representative of an aircraft ditching. For instance, this has been one of the objectives of the recent European projects SMAES (Smart Aircraft in Emergency Situations) and SARAH (Increased safety and robust certification for ditching of aircraft and helicopters). Several studies have been conducted in the framework of these European projects. Siemann and Langrand (2017) studied the impact of deformable flat plates with the Coupled Eulerian-Lagrangian (CEL) and SPH methods. Iafrati et al. (2020) experimentally studied the effect of the fuselage section thickness and curvature on the loads, pressure and strains. Some of the configurations studied featured suction, cavitation, and ventilation phenomena. Iafrati and Grizzi (2019) experimentally investigated the influence of horizontal velocity on the development of cavitation and ventilation during the impact of a double curvature fuselage section. Spinosa et al. (2024) extended this analysis to other double curvature fuselage sections. The authors insisted on the influence of horizontal velocity, pitch angle and body curvature on the hydrodynamic phenomena (cavitation and ventilation) observed during these impacts. Spinosa et al. (2022) experimentally and numerically studied the landing phase of reduced scaled fuselages, more specifically, the suction phenomena observed at the rear of the fuselages and the influence of the longitudinal curvature on this phenomenon and the hydrodynamic forces. Hammani (2020) used an SPH approach with adaptative particle refinement to study the fuselage impacts introduced by Iafrati and Grizzi (2019). The author investigated the capability of the SPH method to model cavitation.

The objective of the present work is to assess the capability of a numerical method relying on the FE explicit solver *Radioss*, a CEL approach and a penalty coupling method to model hydrodynamic impacts that are representative of an aircraft ditching, *i.e.* oblique water impacts of fuselage sections with a high horizontal velocity, and involving suction and cavitation phenomena. The capability of the present numerical method to model suction during a vertical impact with varying velocity has been demonstrated by Goron et al. (2023). The same method is now applied to study the oblique water impact of different generic fuselage sections, referred to hereafter as specimens, based on the experiments carried out in the framework of the European project SARAH and partially presented by Iafrati and Grizzi (2019) and Spinosa et al. (2024).

This paper is organised as follows: Section 2 briefly describes the experiments, associated numerical models, and analysis methods. The influence of some key numerical parameters on the numerical results is discussed in Section 3. Section 4 compares the numerical and experimental results for each specimen. The effect of the variation of the transversal cross-section and longitudinal curvature of the specimens on the impact loads is discussed in Section 5. The effect of cavitation on the hydrodynamic loading is discussed in Section 6. Finally, conclusions are drawn, and orientations for future research are discussed in Section 7.

# 2. Description of the experiments and numerical models

The present work focuses on the numerical simulation of the oblique hydrodynamic impacts of different specimens. It relies on experiments led in the framework of the European project SARAH.



Fig. 2. Illustration of the SP2 and SP3 (a) longitudinal profile, and (b) transversal cross-section.



# Impact direction

Fig. 3. Bottom view of the SP3 and location of the pressure probes, modified from lafrati et al. (2019). The location of the probes is given in millimetres and is similar for SP2.

# 2.1. Description of the experiments

The oblique water impact experiments have been carried out at the High-Speed Ditching Facility of CNR-INM in Rome. This experimental facility is extensively described in the following articles: Iafrati et al. (2015) and Iafrati and Grizzi (2019). The oblique water impacts of different specimens have been studied experimentally with different impact velocities, vertical-to-horizontal velocity ratios, pitch angles, etc. In this work, we focus on the impacts of two of these specimens (named SP2 and SP3) with a vertical-to-horizontal velocities ratio V/U = 0.0375, a pitch angle of 6°, and an initial impact velocity ranging from 21 m/s to 46.19 m/s. The impact velocity variation is small enough to be considered constant during the entire impact duration (Iafrati and Grizzi, 2019). This range of impact velocities corresponds to the one experienced by small to mid-size aircraft during ditching. Specimens SP2 and SP3 have the same longitudinal curvature but different transversal cross-sections. Indeed, SP2 has a circular transversal cross-section, while SP3 has an elliptical one, as illustrated in Fig. 2. The evolution of the hydrodynamic load exerted by the fluid on the specimens is measured using load cells and pressure probes. The pressure probes are located on the bottom surface of the specimens, as illustrated in Fig. 3 for SP3 (the position of the sensors is similar for SP2). For more details on the specimen's geometry and pressure probes' position, refer to the following reports and articles: Iafrati and Olivieri (2017), Iafrati et al. (2019, 2020) and Spinosa et al. (2024).

Let us estimate the characteristic values of the Reynolds number  $Re = (UL)/v_w$ , the Weber number  $We = (\rho_w U^2 L)/\sigma$  and the Froude number  $Fr = U/\sqrt{gL}$  associated with the SARAH water impact experiments. Where U is the characteristic velocity of the body, L is its characteristic length,  $v_w$  is the kinematic viscosity of water,  $\sigma$  is the surface tension coefficient (air-water interface), and g is the acceleration due to gravity. With U ranging between 21 and 46.2 m/s, L = 1.2 m,  $v_w = 10^{-6}$  m<sup>2</sup>/s,  $\sigma = 7.2 \times 10^{-2}$  N/m, we find that Re ranges from  $2.5 \times 10^7$  to  $5.5 \times 10^7$ , We from  $7.4 \times 10^6$  to  $3.6 \times 10^7$  and Fr from 6.1 to 13. The large values of the Reynolds and Weber numbers suggest that viscosity and surface tension have a negligible influence on the flow, except in a thin boundary layer near the body (see for instance Korobkin and Pukhnachov (1988), Zhu et al. (2006) and Moore et al. (2014)), and on the hydrodynamic loads. The value of the Froude number suggests that gravity is likely to have a mild effect on the flow and generated loads (Sun and Faltinsen, 2007). However, note that gravity has been considered in the simulations presented hereafter since it can be easily included in the numerical model and does not lead to any increase in computation time.



Fig. 4. Illustration of the rigid body model for SP3. The kinematic links between the structural nodes (green points •) and the primary node (red point •) are represented by the red lines -. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

### 2.2. Numerical model

The present water impact problem is modelled using the explicit FE solver *Radioss* (version 2020) developed by Altair. This solver has been used and validated in Goron et al. (2023) to study vertical water impacts with varying velocity and involving a suction phenomenon. The structure and fluid domains are three-dimensional. However, only half of the impact problem is modelled owing to symmetry. Symmetry allows us to reduce the model size and the associated computation time. The characteristics of the cluster used to perform the computations are given in Table 1.

#### 2.2.1. Structure modelling

The structure is discretised using Mindlin–Reissner four-node bi-linear shell elements of 15 mm thickness. The characteristic structural element size is  $35 \times 35$  mm<sup>2</sup>. The size of the structural elements should be defined to respect the following ratio:  $l_{elem} \ge 2 \times l_f$ , where  $l_f$  is the size of the fluid elements near the structure. According to the *Radioss* documentation, compliance with these proportions ensures the continuity of loading of the structure by the fluid elements. The normal direction of the structural elements is oriented outward (*i.e.* towards the water). The specimens are modelled as rigid bodies, meaning that the nodes of the structure are kinematically linked to a primary node (see Fig. 4). The different geometries studied in this article are given as supplementary material in ".step" format.

# 2.2.2. Fluid modelling

The fluid flow is described by an Eulerian multi-material formulation (*Radioss* law 51). The effects of fluid viscosity and surface tension are neglected. The validity of these assumptions has been discussed in Section 2.1. Adiabatic conditions are also assumed. Two phases are considered: air and water. The fluid mixture is modelled based on the 6-equation model described by Saurel et al. (2009). More details about the numerical method can also be found in the following articles: (Saurel et al., 2007; Petitpas et al., 2007). The transport equation for the air volume fraction  $\alpha_a$  is given by:

$$\frac{\partial \alpha_a}{\partial t} + \vec{V} \cdot \nabla \alpha_a = 0, \tag{1}$$

where  $\vec{V}$  is the fluid velocity. The water volume fraction is then obtained by  $\alpha_w = 1 - \alpha_a$ . Note that, with the present model, the interface between the different phases is a diffuse zone. Therefore, there is no sharp free surface interface within the numerical model; instead, it is a zone spread over a few elements within which the proportion of air and water varies gradually.

The evolution of the mass density for each phase is given by Eqs. (2) and (3):

$$\frac{\partial (\alpha_a \rho_a)}{\partial t} + \operatorname{div} \left( \alpha_a \rho_a \vec{V} \right) = 0,$$
(2)
$$\frac{\partial (\alpha_w \rho_w)}{\partial t} + \operatorname{div} \left( \alpha_w \rho_w \vec{V} \right) = 0,$$
(3)

where  $\rho_a$  and  $\rho_w$  are the air and water mass density, respectively.

A single velocity field is used to describe the motion of the different phases. The momentum equation for the mixture is written as:

$$\frac{\partial(\rho\vec{V})}{\partial t} + \operatorname{div}\left(\rho\vec{V}\otimes\vec{V}\right) + \nabla P = 0,\tag{4}$$

where  $\rho = \alpha_a \rho_a + \alpha_w \rho_w$  is the mass density of the mixture, and *P* is an equilibrium pressure (between the two phases) defined later on.

The specific internal energies of the air  $(e_a)$  and water  $(e_w)$  are given by Eqs. (5) and (6), respectively:

$$\frac{\partial(\alpha_a \rho_a e_a)}{\partial t} + \operatorname{div}(\alpha_a \rho_a e_a) + \alpha_a P_a \operatorname{div} \vec{V} = 0,$$
(5)

Table 2								
Parameters	for	the	air	equation	of	state:	ideal	gas.

Parameters	Values
$\gamma_a$	1.4
$\rho_a^0$	1.22 · 10 <sup>-6</sup> g/mm <sup>3</sup>
$p_a^0$	0.101325 MPa

#### Table 3

m-11- 0

Parameters for the water equation of state: stiffened gas.

Parameters	Values
$\gamma_w$	4.4
$\rho_w^0$	$1.0 \cdot 10^{-3} \text{ g/mm}^3$
$P_w^0$	0.101325 MPa
	500 m/s

$$\frac{\partial(\alpha_w \rho_w e_w)}{\partial t} + \operatorname{div}(\alpha_w \rho_w e_w) + \alpha_w P_w \operatorname{div} \vec{V} = 0,$$

where  $P_a$  and  $P_w$  are the air and water pressure, respectively.

The system composed of Eqs. (1)-(6) is closed by two equations of state (one for each phase). The air behaviour is modelled using a perfect gas equation of state:

$$P_a = (\gamma_a - 1)\rho_a e_a,\tag{7}$$

where  $\gamma_a$  is the heat capacity ratio for air at ambient temperature. The values of the mentioned parameters are given in Table 2. The water behaviour is modelled using a stiffened gas equation of state:

$$P_w = (\gamma_w - 1)\rho_w e_w - \gamma_w P^*, \tag{8}$$

where  $\gamma_w$  is an empirical parameter that defines the non-linearity of the equation, and  $P^*$  is a pressure coefficient that defines the liquid's initial compressibility.  $P^*$  is related to the speed of sound in the material,  $c_{sw}$ , by the following equation:

$$P^* = \frac{\rho_w^0 c_{sw}^2}{\gamma_w}.$$

The values of the mentioned parameters are given in Table 3. Note that such values are commonly used to model water with a stiffened gas equation of state in the framework of fluid or fluid-structure interaction problems (e.g. Saurel et al. (2009) and Faucher et al. (2017)). Also, note that the speed of sound in the water is defined as  $c_{sw} = 500 \text{ m/s}$  for most of the numerical simulations presented in this article. This value is different from the physical speed of sound in the water, which is close to 1500 m/s. There are numerical reasons for using a different value. Indeed, for the problems studied, the compressibility of water plays almost no role because the pressures reached are not high enough. Therefore, decreasing the value of  $c_{sw}$  makes it possible to increase the size of the stable time step of the numerical simulation (via the CFL condition) and, thus, reduce computation times while ensuring the incompressibility condition of the liquid. This hypothesis has been verified numerically by running simulations with different values of  $c_{sw}$  (see Appendix A).

The equilibrium pressure P used in Eq. (4) is computed as follows. The air and water masses are computed for given values of  $\alpha_a, \alpha_w, \rho_a, \rho_w$  in an element:

$$m_a = \alpha_a \rho_a,$$

$$m_w = \alpha_w \rho_w.$$
(10)

Then the values for P,  $e_a$ ,  $e_w$ ,  $\rho_a$ ,  $\rho_w$ , described by the system of five equations given below (Eq. (11)), are computed using a Newton–Raphson iterative method and considering  $m_a$  and  $m_w$  constant:

$$\begin{cases} \frac{m_a}{\rho_a} + \frac{m_w}{\rho_w} - 1 = 0, \\ e_a - e_a^0 + P \cdot \left(\frac{1}{\rho_a} - \frac{1}{\rho_a^0}\right) = 0, \\ e_w - e_w^0 + P \cdot \left(\frac{1}{\rho_w} - \frac{1}{\rho_w^0}\right) = 0, \\ P_a(\rho_a, e_a) = P, \\ P(a, -e_b) = P, \\ P(a, -e_b) = P, \end{cases}$$
(11)

The initial vertical position of the interface between the air and water sub-domains is z = 0 mm. In the air sub-domain (initially for  $z \ge 0$ ), the air volume fraction is initialised with a value  $a_a^0 = 1$  and the water volume fraction with  $a_w^0 = 0$ . In the water sub-domain (initially for  $z \le 0$ ), the air volume fraction is initialised with a value  $\alpha_a^0 = \epsilon$  and the water volume fraction with  $\alpha_{u}^{0} = 1 - \epsilon$ , with  $\epsilon = 10^{-4}$ . Introducing a small fraction of air into the water guarantees P > 0 and  $c_{s}^{2} > 0$ , thus the hyperbolicity of the problem, while ensuring that the mixture behaviour remains similar to the one of pure water. Note that  $c_s$  refers to the air-water mixture speed of sound and is defined as follows:

$$c_s^2 = \frac{(\alpha\rho)_w}{\rho}c_{sw}^2 + \frac{(\alpha\rho)_a}{\rho}c_{sa}^2$$
(12)

(6)



**Fig. 5.** Dimensions (in mm) and mesh of the fluid domain. Outside the impact zone, the size of the fluid elements scales with a factor of 1.2, *i.e.*  $l_{f_{n+1}} = 1.2 \times l_{f_n}$ , with *n* increasing with distance from the impact zone. The structure is represented in green and  $L_{structure} = 1240$  mm. The illustration is not to scale. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

where  $c_{sa}$  is the speed of sound in the air.

The fluid domain is discretised using 3D continuum 8-node elements with one integration point. The fluid elements close to the structure, a zone referred to hereafter as the impact zone, are of uniform size. For the reference mesh, the size of the fluid elements is equal to  $5 \times 5 \times 5$  mm<sup>3</sup> in the impact zone. Other meshes have also been considered, and the influence of the size of the fluid elements in the impact zone is discussed in Section 3.2. The structure is located in the impact zone from the beginning to the end of the numerical simulation. The dimensions of the fluid domain (larger than the impact zone) are chosen large enough, and absorbing boundary conditions are applied at the boundaries of the fluid domain to limit border effects such as acoustic wave reflections at the boundaries of the fluid domain. The dimensions of the fluid domain are given in Fig. 5. The origin of the coordinate system is also indicated in this figure.

#### 2.2.3. Cavitation modelling method

The *Radioss* solver used in this work does not feature a cavitation model that can describe all the physical phenomena occurring during cavitation (such as phase change). However, it can describe the mechanical consequences of this phenomenon, namely pressure saturation and changes in mass density. This type of approach is referred to as barotropic cavitation models and is commonly used in the literature (Dellanoy and Kueny, 1990; Goncalves and Patella, 2009; Egerer et al., 2013; Goncalvès, 2014; Fuster, 2019; Sarkar, 2019; Sarkar et al., 2021). The presented model works as follows:

- The air behaviour is modelled using a perfect gas equation of state. By definition, the perfect gas equation of state imposes a positive absolute pressure within the gas phase, which means that the relative pressure in a fluid element containing air cannot decrease below  $-P^0$ .
- A "small" air volume fraction  $\alpha_a^0 = 10^{-4}$  is initially introduced into the water sub-domain for numerical stability. Thus, a specific volume of air is present in every fluid element of the model at every time step.
- The pressure is assumed to be in equilibrium within a fluid element containing an air–water mixture ( $P_w = P_a$ ). Thus, the relative pressure in the elements containing an air–water mixture cannot decrease below  $-P^0$ . When the mixture's relative pressure approaches  $-P^0$ , the mixture strongly expands, and its mass density decreases. This behaviour is illustrated in Fig. 6, obtained using the method presented by Elhimer et al. (2017).

In conclusion, the *Radioss* solver can model a phenomenon similar to cavitation through a barotropic approach. For convenience, in the following article, we will refer to this phenomenon as cavitation. Note that more complex cavitation models that explicitly describe the phase change can be found in the literature (Folden and Aschmoneit, 2023). However, barotropic models are not necessarily less accurate than models that include phase change (Frikha et al., 2009).

# 2.2.4. Fluid-structure interaction

The fluid–structure interaction is modelled using a penalty method (*Radioss* interface *type 18*). The Lagrangian structural elements are immersed in the Eulerian fluid grid. The structure and fluid domains are meshed independently and superimposed. The coupling algorithm uses an influence zone defined for each structural element over a distance  $h_c$  (coupling thickness) in their normal direction (see Fig. 7). The *Radioss* documentation suggests specifying this coupling thickness using Eq. (13):

$$h_c = 1.5 \times l_f, \tag{13}$$

where  $l_f$  is the size of the fluid elements in the impact zone.



**Fig. 6.** Evolution of the normalised mass density  $\rho/\rho_w^0$  in an air-water mixture in equilibrium as a function of relative pressure  $\mathcal{P} = P - P^0$  for different initial values of the air volume fraction  $\alpha_v^0$ . The initial (or atmospheric) pressure  $P^0$  is set to 0.101325 MPa.

When detected inside the influence zone, a coupling force is applied to a fluid node. A force of the same amplitude and opposite direction is reciprocally applied to the structure. This coupling force is oriented along the local normal direction of the structure, and its amplitude is computed using Eq. (14):

$$F = \frac{k_c}{h_c} d \cdot \tilde{d},\tag{14}$$

where  $k_c$  is the coupling stiffness, d is the penetration distance of a fluid node inside the influence zone of the structure, and  $\tilde{d}$  is the displacement of a fluid node once it is detected inside the influence zone of the structure (see Fig. 7). Variables d and  $\tilde{d}$  are computed using the following equations:

$$d = \max(0, h_c - |(\vec{r}_{fluid} - \vec{r}_{lag}) \cdot \vec{n}|),$$
(15)

$$\begin{cases} \frac{d\vec{a}}{dt} = (\vec{V}_{fluid} - \vec{V}_{lag}) \cdot \vec{n}, & \text{if } d > 0 \\ \frac{d\vec{a}}{dt} = 0, & \text{if } d \le 0 \end{cases}$$
(16)

where  $\vec{r}_{fluid}$  is the position of the fluid node,  $\vec{r}_{lag}$  is the projected position of the fluid node on the structural element,  $\vec{V}_{fluid}$  is the velocity of the fluid node,  $\vec{V}_{lag}$  is the velocity of the structure at the projected position of the fluid node, as defined in Fig. 7. The *Radioss* documentation suggests to use the following value for  $k_c$ :

$$k_c = \frac{\rho_w^0 U^2 S_{el}}{h_c},$$
(17)

where U is the structure initial horizontal velocity, and  $S_{el}$  is the mean surface of the structural elements. As discussed in Section 3.1, the value of  $k_c$  has a strong influence on the numerical results and should be selected with care depending on the application studied.

#### 2.2.5. Initial and boundary conditions

An initial velocity, oriented in the guide direction, is applied to the structure. The velocity variation during each numerical simulation remains around 1%, as observed during the experiments (see section III-Results and figure 4 in Iafrati and Grizzi (2019)).

Gravity is applied to all the nodes of the model in the  $\vec{z}$  direction ( $\vec{g} = -9.81 \ \vec{z} \ m/s^2$ ). Gravity is used to initialise the pressure field in the fluid domain using the following relation:

$$P = P^0 + \rho^0 \, gz, \tag{18}$$

where  $P^0$  is the initial (or atmospheric) pressure at z = 0,  $\rho^0$  is the initial fluid mass density, and z is the vertical coordinate (recall that z = 0 mm corresponds to the initial air–water interface). Unless otherwise stated, the initial pressure is set as  $P^0 = 0.101325$  MPa (atmospheric pressure at sea level).

At the boundaries of the fluid domain corresponding to the symmetry plane, the velocity in the direction normal to the fluid domain is set at zero. Non-reflecting boundary conditions are applied to the other boundaries of the fluid domain based on the pressure formulation given in Bayliss and Turkel (1982).



**Fig. 7.** Illustration of the penetration of a fluid node inside the influence zone of the structure, d(t). The relative velocity of the fluid node regarding the Lagrangian (structural) element is  $(\vec{V}_{fluid} - \vec{V}_{lag}) \cdot \vec{n}$ .

# 2.3. Results analysis

For the sake of the numerical approach validation, the numerical and experimental results are compared in terms of force coefficient and relative hydrodynamic pressure measured at the different probes.

The force coefficient  $C_f$  is defined as:

$$C_f = \frac{F_n}{\rho_w^0 U^2 S},\tag{19}$$

where  $F_n$  is the normal component of the hydrodynamic force (in the load cells coordinate system) acting on the specimen due to the fluid-structure coupling algorithm, and  $S = L \times W = 1240 \times 660 \text{ mm}^2$  is the projected area of the specimens on the horizontal plane (global coordinate system). The evolution of these physical quantities is given as a function of the non-dimensional time  $t^* = \frac{t-t_1}{4t}$ ,

with *t* the time,  $t_i$  the initial time of the impact, and  $\Delta t$  the experimental impact duration. Experimentally,  $t_i$  corresponds to the contact between the lowest point of the specimens and the water. The end of the impact corresponds to the jet root reaching the leading edge of the specimens during the experiments. Numerically,  $t_i$  corresponds to the contact between the influence zone of the specimens and the water. The choice of this numerical initial time is justified because it has been noticed that the water started moving before the structure formally reached the altitude z = 0 mm during the simulations.

Numerically, different probes have been used to monitor the evolution of the hydrodynamic pressure and mass density. These probes have been located at positions similar to those during the experiments. Note that the "numerical" pressure probes are located outside the influence zone of the structures at a distance of  $\frac{4}{3}h_c$  from the structure in the local normal direction (see Fig. 8). Indeed, due to the coupling method used in the present study, the pressure signals obtained inside the influence zone of the structure are noisy and difficult to analyse. The mass density in the elements located in the influence zone rapidly evolves over time, between  $\rho_w^0 = 10^{-3}$  g/mm<sup>3</sup> and  $\rho_a^0 = 10^{-6}$  g/mm<sup>3</sup>. This evolution of the mass density may lead to oscillations of the pressure signals. The numerical results are presented in terms of relative pressure,  $\mathcal{P} = P - P_0$  (MPa).

The position of the air–water interface is presented along the symmetry plane of the numerical model, based on the same postprocessing method as presented in Goron et al. (2023). It is assumed that the air–water interface position corresponds to the iso-line position where the air volume fraction is equal to 0.5. However, note that the present numerical model uses a multi-material fluid approach with a diffuse interface between the different fluids: strictly speaking, no sharp air–water interface exists in the model.

# 3. Investigation of numerical parameters affecting the oblique water impact simulations without cavitation

As observed in Goron et al. (2023), the results obtained with the present numerical method are sensitive to several numerical parameters, particularly the coupling stiffness and the size of the fluid elements in the impact zone. It has been concluded that assessing the influence of these parameters is a critical point when using this numerical method. To do so, for the case of oblique water impacts, the effect of the coupling stiffness and size of the fluid elements is investigated through the numerical simulation of SP3 entering the water with an initial horizontal velocity U = 21 m/s. The parametric studies are presented for this impact



Fig. 8. Illustration (not to scale) of the influence zone of the specimens and pressure probes position along the midline of the specimens.

Table 4

Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: effect of the coupling stiffness on the mean time step and computation time. 128 CPUs have been used for all computations.

<i>k</i> <sub>c</sub> (N/mm)	$0.01 \times k_{c_0}$	$0.1 \times k_{c_0}$	$k_{c_0} = 72$	$4 \times k_{c_0}$	$8 \times k_{c_0}$	$16 \times k_{c_0}$
Mean time step (ms)	0.00126	0.00126	0.00071	0.00036	0.00025	0.00018
Computation time (DD-hh:mm:ss)	0-15:01:45	1-04:49:13	2-06:29:07	4-03:21:56	5-14:52:37	8-20:41:41

configuration only to avoid redundancy with the similar results obtained for higher-velocity impacts. The convergence of the results is based on the analysis of the hydrodynamic force and relative pressure, mass density and position of the air–water interface. For the sake of brevity, only the pressure and mass density results associated with probe  $p^9$  are presented in this section. This probe is located on the symmetry plane at the rear of the specimen, where negative relative pressures (suction) are observed (see Fig. 3).

#### 3.1. Effect of the coupling stiffness

The effect of the coupling stiffness  $k_c$  on the numerical results is investigated in this section. The numerical results obtained for the SP3 oblique water impact, with U = 21 m/s and a fluid element size of  $l_f = 5$  mm, are compared for different coupling stiffness values:  $k_c = \{0.01; 0.1; 4; 8; 16\} \times k_{c0}$ . The recommended coupling stiffness, obtained using Eq. (17), is  $k_{c0} = 72$  N/mm.

The mean time step and the computation time for each case are presented in Table 4. Increasing the coupling stiffness decreases the mean time step, thus increasing the computation time. For instance, the time required to perform the numerical simulation for  $k_c = 16 \times k_{c0}$  is approximately 4 times greater than for the case with  $k_c = k_{c0} = 72$  N/mm.

The evolution of the relative pressure  $\mathcal{P} = P - P^0$  and normalised mass density  $\rho/\rho_w^0$  obtained at probe p9 for the different coupling stiffnesses considered is presented in Fig. 9. For  $k_{c_0}$  and  $k_c = 0.1 \times k_{c0}$ , similar pressure evolutions are observed. The pressure gradually decreases, then remains at similar (and negative) levels for both coupling stiffnesses. At  $t^* = 1$ ,  $\mathcal{P} \simeq -0.019$  MPa for  $k_{c_0}$  and  $k_c = 0.1 \times k_{c0}$ . For  $k_c \ge 4 \times k_{c0}$ , the tendencies observed begin to diverge during the impact: the pressure gradually increases towards the initial pressure ( $\mathcal{P} = 0$  MPa) and the mass density of the air–water mixture gradually decreases towards the air mass density ( $\rho/\rho_w^0 \simeq 0$ ). This pressure increase to the atmospheric pressure level is observed earlier for higher coupling stiffnesses. For instance for  $k_c = 8 \times k_{c0}$ , the pressure increase occurs around  $t^* \simeq 0.6$ , whereas for  $k_c = 16 \times k_{c0}$  it occurs around  $t^* \simeq 0.5$ . These pressure variations are associated with the flow detachment observed at the rear of SP3 during the impact (see the discussion below).

The evolution of the force coefficient  $C_f$  for the different coupling stiffnesses considered is presented in Fig. 10. The oscillations observed before  $t^* = 0.2$  are non-physical and result from the initialisation of the fluid-structure interaction algorithm. The shaded areas represent the envelope of the high-frequency oscillations observed numerically during the hydrodynamic impact. Note that the envelope of the high-frequency oscillations grows larger after  $t^* \simeq 1$ , for all the coupling stiffnesses considered. This is due to the development of horizontal forces opposed to the structure displacement at the end of the impact caused by water pilling up in front of the structure. The solid lines correspond to the filtered numerical signals obtained using a low pass second order Butterworth filter with a 50 kHz cut-off frequency. For  $k_{c0}$  and  $k_c = 0.1 \times k_{c0}$ , the evolution of  $C_f$  is nearly linear. The maximum value of the force coefficient is similar in both cases:  $C_{f max} = 0.06$ . On the one hand, decreasing the coupling stiffness ( $k_c = 0.01 \times k_{c_0}$ ) leads to a global decrease of the force coefficient magnitude. The position of the air-water interface along the symmetry plane of the numerical model at  $t^* = 1$ , presented in Fig. 11, shows that this reduction in force is accompanied by fluid penetration through the structure, which is a sign of too small coupling stiffness. On the other hand, increasing the coupling stiffness increases the force coefficient from the first moment of impact. It should also be noted that the evolution of  $C_f$  is no longer linear for  $k_c \ge 4 \times k_{c_0}$ . The greater the coupling stiffness, the earlier the change in the slope of  $C_f$  occurs during the impact. Indeed, the change of slope is observed around  $t^* \simeq 0.7$  for  $k_c = 4 \times k_{c0}$ ,  $t^* \simeq 0.6$  for  $k_c = 8 \times k_{c0}$ , and  $t^* \simeq 0.5$  for  $k_c = 16 \times k_{c0}$ . This change in the slope of  $C_f$ is associated with the flow detachment occurring at the rear of the SP3. Indeed, when flow detachment occurs, a reduction of the low-pressure zone located at the rear of the SP3 is observed (the pressures measured in this zone become predominantly positive or null), leading to a reduction in the (negative) suction forces and, therefore, to an increase in the positive hydrodynamic forces exerted by the fluid on the structure.

The air–water interface profile along the symmetry plane of the numerical model at  $t^* = 1$  is depicted in Fig. 11 for the different coupling stiffnesses considered. The penetration of the fluid through the structure mentioned above for  $k_c = 0.01 \times k_{c_0}$  is clearly



Fig. 9. Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: evolution of the relative pressure  $\mathcal{P}$  and normalised mass density  $\rho/\rho_w^0$  obtained at probe p9 as a function of the non-dimensional time  $t^*$  for different coupling stiffnesses  $k_c$ .



Fig. 10. Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: evolution of the force coefficient  $C_f$  as a function of the non-dimensional time  $t^*$  for different coupling stiffnesses  $k_c$ .

visible. At this instant, like the rest of the impact, the positions of the air–water interface observed for  $k_c = k_{c_0}$  and  $k_c = 0.1 \times k_{c_0}$  are almost identical. Finally, flow detachment is visible at the rear of SP3 for  $k_c \ge 4 \times k_{c_0}$ : the probe *p*9 is almost entirely above the air–water interface, *i.e.* out of the water, for these three configurations. Note that, apart from the case with  $k_c = 0.01 \times k_{c_0}$ , the position of the air–water interface is similar at the front of the SP3 for all the coupling stiffnesses considered.

In conclusion, no clear convergence of the numerical results is observed regarding the coupling stiffness for the SP3 oblique water impact with an initial horizontal velocity U = 21 m/s. In contrast to what has been observed in Goron et al. (2023) for the case of a cone entering and exiting the water, this absence of convergence seems to be related to the flow detachment observed at the rear of the specimen if a too-high value of  $k_c$  is used. Based on these observations, it has been decided to select a coupling stiffness high enough to ensure the impermeability of the specimen, also leading to a flow behaviour representative of the one observed during the corresponding experiments. Experimentally, no flow detachment has been observed for this impact configuration (U = 21 m/s) via the pressure probes or the underwater images (see Iafrati and Grizzi (2019) and Section 4). Numerically, for  $k_{c_0} = 72$  N/mm, no fluid penetration across the specimen nor flow detachment at the rear of the specimen is observed. Therefore, to study this type of oblique water impact, the coupling stiffness recommended by the *Radioss* user manual  $k_{c_0}$  appears as a suitable value of coupling



Fig. 11. Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: air-water interface profile at  $t^* = 1$  for different coupling stiffnesses  $k_c$ .

stiffness to ensure an evolution of the hydrodynamic load and flow behaviour representative of the one observed experimentally. Similar conclusions have been drawn from additional investigations with higher impact velocities (U = 34.5 and U = 45.2 m/s).

# 3.2. Effect of the size of the fluid elements in the impact zone

The effect of the size of the fluid elements on the numerical results is investigated in this section. The numerical results obtained for the SP3 oblique water impact with U = 21 m/s and  $k_c = 72$  N/mm are presented for different fluid element sizes in the impact zone:  $l_f = \{3; 4; 5; 7.5\}$  mm.

The evolution of the relative pressure  $\mathcal{P} = P - P^0$  and normalised mass density  $\rho/\rho_w^0$  obtained at probe p9 for the different fluid element sizes considered is presented in Fig. 12. The evolution of these two physical quantities is overall similar for  $l_f \leq 5$  mm: the pressure decreases at the start of the impact until it reaches the value  $\mathcal{P} \simeq -0.022$  MPa around  $t^* = 0.4$ , and remains close to this value until the end of the impact.

The evolution of the force coefficient  $C_f$  for the different fluid element sizes considered is presented in Fig. 13. The evolution of the force coefficient is similar for all element sizes tested, except for  $l_f = 7.5$  mm, for which the force evolution presents some oscillations. For  $l_f \le 5$  mm, the evolution of the force coefficient is linear, and the maximum force coefficient  $C_f \simeq 0.056$  is reached around  $t^* = 0.98$ , then rapidly decreases.

The air–water interface profile along the symmetry plane of the model at  $t^* = 1$  is depicted in Fig. 14 for the different fluid element sizes considered. A similar flow behaviour is observed for all values of  $l_f$ . Note that the formation of a jet is observed for  $l_f = 3$  mm and not for the simulations performed with larger fluid elements. Smaller fluid elements allow a more precise representation of the free surface, particularly of the jet generated at impact, as mentioned in Battistin and Iafrati (2003), Del Buono et al. (2021) and Del Buono (2022).

In conclusion, for the SP3 oblique water impact at U = 21 m/s, the size of the fluid elements in the impact zone has little influence on the evolution of the impact loads on the specimen (force and pressure). Converging tendencies are observed for  $l_f \le 5 \text{ mm}$ . Even if using smaller fluid elements yields a more precise description of the free surface evolution, it hardly affects the hydrodynamic load but strongly increases the computation times. Therefore, the size of the fluid elements has been set to  $l_f = 5 \text{ mm}$  in the following numerical simulations, which appears as a good compromise between computation time and accuracy.

# 4. Comparisons with the experimental results

In this section, the oblique water impact of the SP3 specimen is studied for three horizontal velocities U = [21; 34.5; 45.2] m/s. The numerical results are compared to the experimental data obtained by the CNR-INM in the framework of the European project SARAH (Iafrati and Grizzi, 2019) in terms of force coefficient and relative pressure observed at the different probes. The position of the probes is given in Fig. 3. For every impact configuration presented in this section, the numerical results have been obtained using the recommended value for the coupling stiffness ( $k_c = k_{c_0}$ ) and a fluid element size of  $l_f = 5$  mm to discretise the impact zone of the numerical model.



**Fig. 12.** Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: evolution of the relative pressure  $\mathcal{P}$  and normalised mass density  $\rho/\rho_w^0$  obtained at probe p9 as a function of the non-dimensional time  $t^*$  for different fluid element sizes  $l_f$  in the impact zone.



Fig. 13. Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: evolution of the force coefficient  $C_f$  as a function of the non-dimensional time  $t^*$  for different fluid element sizes  $l_f$  in the impact zone.

# 4.1. Impact with suction (U = 21 m/s)

The SP3 oblique water impact with an initial horizontal velocity U = 21 m/s, involving only a suction phenomenon, is considered first. The numerical results in terms of relative pressure  $\mathcal{P} = P - P^0$  are compared to the pressure measured experimentally at probes *p*17, *p*13, *p*9 and *p*4 in Fig. 15. Recall that these probes are located along the midline of the specimen at the rear of the fuselage (Fig. 3). Also, the lowest point of SP3 in this configuration is located close to probe *p*17, near the SP3 curvature change. Thus, the initial instant of impact during the experiments ( $t^* = 0$ ) is determined using the pressure measurements obtained with probe *p*17 (Fig. 15(a)). For this impact velocity, a suction phenomenon is observed at the rear of SP3 (downstream of the point of first contact between the structure and the water), as shown by the negative (relative) pressures observed at all probes in this area. However, the intensity of the suction phenomenon is fairly low, and the pressures measured experimentally or observed numerically at sensors *p*4 and *p*9 remain close to  $\mathcal{P} = 0$  MPa. The pressures measured near probes *p*13 and *p*17 are higher than the ones observed numerically. For example,  $\mathcal{P}_{p17 num}(t^* = 1) = -0.06$  MPa and  $\mathcal{P}_{p17 exp}(t^* = 1) \simeq -0.02$  MPa. This difference in magnitude could be due to the numerical method overestimating the intensity of the suction phenomenon in this area or to measurement errors. Indeed, the pressure spikes measured at probes *p*13 and *p*17 could also be non-physical due to slightly protruding probes generating cavitating



Fig. 14. Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: air-water interface profile at  $t^* = 1$  for different fluid element sizes in the impact zone.

vortices at such speeds (Iafrati et al., 2015) or to slight thermal drifts of the probes occurring when they touch the water, which is at a lower temperature than air (Spinosa et al., 2024). Despite this difference in magnitude, the tendencies observed numerically and experimentally are similar.

The evolution of the force coefficient  $C_f$  is presented in Fig. 16. The force levels predicted numerically and measured experimentally are similar. The force coefficient  $C_f$  evolves linearly during the simulation until reaching a maximum around  $t^* = 1$ , then decreases rapidly. This decrease in force is related to the wetted surface reaching the leading edge of the fuselage section. The experimental measurements follow overall the same tendency but show large amplitude oscillations, probably due to vibrations of the guide during impact (Iafrati et al., 2015). These oscillations are particularly visible for this impact velocity, which is one of the lowest considered during this test campaign. Indeed, for this impact velocity, the magnitude of the hydrodynamic force is of the order of magnitude of the measurement uncertainties (Iafrati et al., 2019).

# 4.2. Impact with suction and cavitation (U = 34.5 m/s)

The SP3 oblique water impact with an initial horizontal velocity U = 34.5 m/s, involving suction and cavitation phenomena, is now considered. The numerical results in terms of relative pressure  $\mathcal{P} = P - P^0$  are compared to the pressure measured experimentally at probes p17, p13, p9 and p4 in Fig. 17. As before, the pressure peak measured at probe p17 is used to identify the initial instant of the impact (Fig. 17(a)). The numerical method satisfactorily predicts the instant at which the pressure peak is detected at probe p17 but significantly underestimates its magnitude:  $P_{p17 num} = 0.054$  MPa compared with  $P_{p17 exp} = 0.51$  MPa. This difference in magnitude is probably due to the location of probes outside the influence zone of the specimen in the numerical model, as explained in Section 2.3. Shortly after the beginning of the impact, a suction phenomenon develops at the rear of SP3, resulting in negative relative pressures measured at the various sensors located in this area. Experimentally, a cavitation pocket starts developing around probe p17 at  $t^* \simeq 0.22$ , then propagates towards the rear edge of SP3. This phenomenon manifests itself by a saturation of the pressures measured in this area around P = -0.1 MPa, first at probe p17, then successively at probes p13, p9 and p4. The cavitation pocket is also visible on the underwater images recorded during the tests (Fig. 19). The pressure measured at probes p17, p13 and p4 varies abruptly from positive values to the saturation vapour pressure. These tendencies could be due to the development of cavitation in this area or to measurement errors. Therefore, caution should be exercised when using the experimental results, particularly regarding estimating the instant at which the cavitation pocket appears during the test. Similarly to the experiments, negative relative pressures are observed at the different probes in the rear part of SP3 in the numerical simulation. Note that the decrease rate of the relative pressure is lower in the numerical simulations than in the experiments. As mentioned in Section 2.2.3, the numerical method does not consider phase change. The pressure saturation at  $\mathcal{P} \simeq -0.1$  MPa, observed at the various probes, is due to the perfect gas equation of state used to model the air. Therefore, no cavitation pocket (i.e. a zone filled with water vapour) is observed during the simulations, but rather a zone with an air-water mixture at low pressure. It should be noted that the pressure observed numerically at probe p17, located near the lowest point of the structure and the limit of the cavitation zone, never reaches  $\mathcal{P} = -0.1$  MPa and



**Fig. 15.** Oblique water impact of SP3 with an initial horizontal velocity U = 21 m/s: evolution of the relative pressure P, obtained at probes (a) p17, (b) p13, (c) p9, and (d) p4, as a function of the non-dimensional time  $t^*$ .



Fig. 16. Oblique water impact of SP3 with an initial horizontal velocity U = 21 m/s: evolution of the force coefficient  $C_f$  as a function of the non-dimensional time  $t^*$ .



**Fig. 17.** Oblique water impact of SP3 with an initial horizontal velocity U = 34.5 m/s: evolution of the relative pressure P, obtained at probes (a) p17, (b) p13, (c) p9, and (d) p4, as a function of the non-dimensional time  $t^*$ . The red dashed lines indicate the onset of the cavitation phenomenon during the experiment.

saturates at  $\mathcal{P} \simeq -0.07$  MPa, indicating that the position of the cavitation zone predicted by the numerical model is not identical to that observed during the tests. Despite this, the pressures recorded at the rear of SP3 during the numerical simulation are overall representative of the experimental measurements.

The evolution of the force coefficient  $C_f$  is presented in Fig. 18. Experimentally, the total hydrodynamic force increases approximately linearly from the first contact with the water, while cavitation occurs, and until the end of the impact. Numerically, the pressure saturation associated with cavitation seems to influence the slope of  $C_f$  compared with the linear evolution observed for the case U = 21 m/s. The fact that the numerical method relies on a barotropic approach to model cavitation could explain the differences in tendency. The numerical method globally overestimates the force coefficient after  $t^* = 0.3$ .

The contact pressure distribution on the impacting surface of SP3 at  $t^* = 0.5$  is compared to an underwater image recorded during the SP3 water impact in Fig. 19. The contact pressure corresponds to the sum of the nodal coupling forces (*F* in Eq. (14)) divided by the area of the corresponding structural element. Numerically, the high-pressure zone is close to the boundary of the wetted surface at the front of the specimen. The numerical method seems to underestimate the size of the wetted surface of the structure at this instant. The size of the low-pressure zone observed at the rear of SP3 during the numerical simulation is comparable to that of the cavitation pocket observed during the experiments. However, this illustration only serves as a qualitative comparison with the experiments.

# 4.3. Impact with suction, cavitation and ventilation (U = 45.2 m/s)

The SP3 oblique water impact with an initial horizontal velocity U = 45.2 m/s, involving suction, cavitation and ventilation phenomena, is now considered. The numerical results in terms of relative pressure  $\mathcal{P} = \mathcal{P} - \mathcal{P}^0$  are compared to the pressure measured experimentally at probes *p*17, *p*13, *p*9 and *p*4 in Fig. 20. Experimentally, the intensity of the suction phenomenon developing at the rear of SP3 is greater than for lower impact velocities. This results in a more rapid decrease in the pressures measured at the rear



Fig. 18. Oblique water impact of SP3 with an initial horizontal velocity U = 34.5 m/s: evolution of the force coefficient  $C_f$  as a function of the non-dimensional time  $t^*$ . The red dashed line indicates the onset of the cavitation phenomenon during the experiment.



Fig. 19. Oblique water impact of SP3 with an initial horizontal velocity U = 34.5 m/s: contact pressure distribution on the impacting surface of SP3 and position of the cavitation pocket observed experimentally at  $t^* = 0.5$ .

of SP3. As a result, the cavitation pocket develops earlier during the impact, at  $t^* \simeq 0.12$ . The simulation accurately describes this phenomenon. The cavitation pocket also grows larger than for lower impact velocities and extends until it reaches the trailing edge of the specimen at  $t^* \simeq 0.42$ . At this instant, the cavitation pocket opens to the open air, and the pressure inside the cavitation pocket goes from the vapour pressure ( $\mathcal{P} \simeq -0.101325$  MPa) to the ambient pressure ( $\mathcal{P} \simeq 0$  MPa). This increase in pressure is first measured at probe p4, the closest to the SP3 trailing edge. Then, it propagates inside the cavitation pocket: the pressure variation is successively measured at probes p9 and p13. Iafrati and Grizzi (2019) described this phenomenon as ventilation. Note that this phenomenon is probably due to the use of a truncated structure and might not be observed during the ditching of a complete structure. Numerically, the ventilation phenomenon is not reproduced: once the pressure observed at probes p4, p9, p13, and p17reaches its saturation threshold  $\mathcal{P} \simeq -0.1$  MPa, it remains at this level until the end of the numerical simulation. The exact reason for the absence of ventilation in the numerical simulation is unknown. However, the use of a rather simple fluid model could be an explanation.

The evolution of the force coefficient  $C_f$  is presented in Fig. 21. A direct consequence of the ventilation phenomenon observed in the experiments is the reduction in the (negative) suction forces at the rear of SP3 and, therefore, an increase in the total hydrodynamic forces. Indeed, after the opening of the cavitation pocket, the relative pressures measured in this zone vary from predominantly negative to predominantly positive or null. This pressure variation leads to the change in slope of  $C_{f exp}$  observed around  $t^* = 0.42$ . Once the cavitation pocket is fully ventilated, at  $t^* \simeq 0.5$ , the hydrodynamic force measured during the test continues to increase until the end of the impact, but at a lower rate. As the ventilation phenomenon is not reproduced numerically, the evolution of  $C_f$  predicted by the numerical method does not display a pronounced slope variation around  $t^* = 0.42$ , as observed with the experimental results. Despite an overestimation of the hydrodynamic force during the early phase of the impact (for instance



**Fig. 20.** Oblique water impact of SP3 with an initial horizontal velocity U = 45.2 m/s: evolution of the relative pressure P, obtained at probes (a) p17, (b) p13, (c) p9, and (d) p4, as a function of the non-dimensional time  $t^*$ . The red dashed line and green dash-dotted line indicate the onset of the cavitation and ventilation phenomena during the experiment, respectively.

 $\Delta C_f(t^* = 0.42) = |\frac{C_{f exp}(t^*=0.42) - C_{f num}(t^*=0.42)}{C_{f exp}(t^*=0.42)}| \simeq 30\%), \text{ the experimental and numerical data are in rather good agreement until the second s$ 

onset of ventilation. Indeed, up to this point, the numerical assumptions are overall representative of the physical phenomena occurring during the impact, particularly the pressure saturation around  $\mathcal{P} = -0.1$  MPa at the rear of the specimen. Despite the differences in tendencies observed for  $t^* \ge 0.42$ , the maximum force coefficient measured experimentally and predicted by the numerical model are reasonably close:  $C_{f exp max} \simeq 0.081$  and  $C_{f num max} \simeq 0.076$ .

The contact pressure distribution on the impacting surface of SP3 at  $t^* = 0.5$  is compared to an underwater image recorded during the SP3 water impact in Fig. 22. The low-pressure zone at the rear of SP3 is more developed than during the impact for U = 34.5 m/s, which is in agreement with the evolution of the pressures recorded in this zone for both cases. The fact that the numerical method is unable to describe the ventilation phenomenon is illustrated by the low-pressure zone not reaching the trailing edge of SP3. Note that beyond  $t^* = 0.8$ , during the numerical simulation, this low-pressure zone is virtually stationary, *i.e.* its position, size and shape no longer change.

#### 5. Effect of the curvature

The present section is dedicated to studying the influence of the body geometry on the hydrodynamic loads during water entry, particularly the effect of the transversal and longitudinal profiles. For all the impact configurations studied, the vertical to horizontal velocities ratio is V/U = 0.0375, and the pitch angle of the specimens is 6°.



Fig. 21. Oblique water impact of SP3 with an initial horizontal velocity U = 45.2 m/s: evolution of the force coefficient  $C_f$  as a function of the non-dimensional time  $t^*$ . The red dashed line and green dash-dotted line indicate the onset of the cavitation and ventilation phenomena during the experiment, respectively.



Fig. 22. Oblique water impact of SP3 with an initial horizontal velocity U = 45.2 m/s: contact pressure distribution on the impacting surface of SP3 and position of the cavitation pocket observed experimentally at  $t^* = 0.5$ .



Fig. 23. Illustration of the transversal cross-sections of SP3 and SP2.

#### 5.1. Effect of the transversal cross-section

The comparison of the SP3 and SP2 results allowed us to estimate the effect of the transversal cross-section on the hydrodynamic loading. Indeed, SP3 and SP2 have the same longitudinal curvature but a different transversal cross-section: SP3 has an elliptical transversal cross-section, whereas SP2 has a circular transversal cross-section (see Fig. 23).

The contact pressure distribution on the impacting surface of specimens SP2 and SP3 at  $t^* = 0.5$  is presented in Fig. 24. For SP2, which presents a more pronounced transverse curvature than SP3, thus a greater local deadrise angle, the wetted surface during



Fig. 24. Oblique water impact of SP2 and SP3 with an initial horizontal velocity U = 34.5 m/s: contact pressure distribution on the impacting surface of the specimens at  $t^* = 0.5$ .



Fig. 25. Oblique water impact of SP2 and SP3 with an initial horizontal velocity U = 34.5 m/s: evolution of the force coefficient  $C_f$  as a function of the non-dimensional time  $t^*$ .

impact is smaller than for SP3, both in the longitudinal and transversal directions. The low-pressure zone at the rear of SP2 is also smaller than for SP3. These results tend to show that a flatter body geometry favours a more rapid extension of the wetted surface in the longitudinal and transversal directions and the development of a larger low-pressure zone at the rear of the structures during this type of impact.

The evolution of the force coefficient  $C_f$  is presented in Fig. 25. Compared with a flatter cross-section (specimen SP3), a circular cross-section (specimen SP2) reduces the wetted surface of the structure and the pressure level at the front of the specimen during impact due to the higher deadrise angle of the section. Therefore, the hydrodynamic load is expected to be lower during the SP2 impact. This tendency is observed both experimentally and numerically. For example, the differences at  $t^* = 1$  between the force coefficients associated with the impacts of specimens SP2 and SP3 are:  $\Delta C_{f exp} = \frac{C_f SP3 \exp^{-C_f SP2 exp}}{C_f SP3 \exp} \approx 82\%$  and  $\Delta C_{f num} = \frac{C_f SP3 \min^{-C_f SP2 num}}{C_f SP3 num} \approx 71\%$ . These observations are in agreement with the work of Iafrati et al. (2020) on the oblique water impact of different single-curvature plates with flat, concave and convex cross-sections.

# 5.2. Effect of the longitudinal curvature

Two other geometries (SP2C and SP2D) are defined numerically to discuss the effect of the longitudinal curvature on the numerical results. These two specimens have the same transversal cross-section as SP2 but a less pronounced longitudinal curvature, as illustrated in Fig. 26. More precisely, SP2C and SP2D have been created by modifying the vertical position of the structural nodes of SP2 from the first point of contact with the water to the trailing edge (linear variation). At the point of first contact with the water



Fig. 26. Illustration of the longitudinal curvature of SP2, SP2C, and SP2D. The position of the pressure probes used in the simulations is also indicated.



Fig. 27. Oblique water impact of SP2, SP2C, and SP2D with an initial horizontal velocity U = 34.5 m/s: contact pressure distribution on the impacting surface of the specimens at  $t^* = 0.5$ .

(near probe *p*17), the structural nodes of SP2, SP2C and SP2D have the same vertical position. The difference in vertical position between the different specimens is maximal at the trailing edge:  $d_z \ _{SP2-SP2C} = 13.2 \text{ mm}$  and  $d_z \ _{SP2-SP2D} = 6.6 \text{ mm}$ . The different geometries are given as supplementary material in ".step" format. The numerical results are compared for an initial horizontal velocity U = 34.5 m/s.

The effect of the longitudinal curvature is particularly visible at the rear of the specimens, where the difference in longitudinal curvature is the most pronounced. The contact pressure distribution on the impacting surface of the three specimens at  $t^* = 0.5$  is presented in Fig. 27. The negative pressure area is smaller when the curvature at the rear of the profile is reduced. This phenomenon is also visible in Fig. 28(a), through the evolution of the relative pressure observed at probe p9: the decrease in pressure occurs later during the impact, and the minimum pressure is higher for profiles with a less pronounced longitudinal curvature (SP2C compared with SP2D and SP2D compared with SP2). One consequence of the reduced suction phenomenon, when the rear of the structure is flatter, is an increase in the total hydrodynamic loading during impact (Fig. 28(b)). These observations are in agreement with the observations of McBride and Fisher (1953).

## 6. Effect of cavitation on the hydrodynamic loading

The effect of the cavitation phenomenon on the evolution of the hydrodynamic load predicted by the numerical method is investigated in this section. Simulations of cavitating impacts, *i.e.* at a given impact velocity, are carried out with different initial pressure:  $P^0 = 0.1$  MPa (reference case, with cavitation) and  $P^0 = 1$  MPa (without cavitation). As discussed in Section 2.2.3, the initial pressure  $P^0$  controls the lower limit that the relative pressure can reach during the simulation, hence influencing the development of the so-called cavitation phenomenon. Recall that the *Radioss* solver can model a phenomenon similar to cavitation through a barotropic approach, but for convenience, we refer to this phenomenon as cavitation. The value  $P^0 = 1$  MPa has been chosen to prevent cavitation from developing (using a higher value would not change the results).



**Fig. 28.** Oblique water impact of SP2, SP2C, and SP2D with an initial horizontal velocity U = 34.5 m/s: evolution of the (a) relative pressure  $\mathcal{P}$  observed at probe p9, and (b) force coefficient  $C_f$  as a function of the non-dimensional time  $t^*$ .



**Fig. 29.** Oblique water impact of SP3 with an initial horizontal velocity U = 34.5 m/s: position of the air–water interface profile and contact pressure distribution along the symmetry plane of the numerical model at  $t^* = 0.5$  for different values of initial pressure  $P^0$ .  $P^0 = 0.1$  MPa corresponds to the reference case with cavitation and  $P^0 = 1$  MPa to a simulation without cavitation.

The contact pressure distribution along the centreline of SP3 and the air–water interface profile along the symmetry plane of the model at  $t^* = 0.5$  obtained for the two values of P0 are presented in Fig. 29. For  $P^0 = 0.1$  MPa, the development of a low-pressure zone (resembling a cavitation pocket) at the rear of the SP3 is indicated by the detachment of the air–water interface from the SP3 surface and the pressure reaching -0.1 MPa. For  $P^0 = 1$  MPa, the pressure remains above -1 MPa at the rear of the SP3. Therefore, no cavitation pocket develops during the impact, as illustrated by the air–water interface remaining close to the surface of the SP3. The contact pressure distribution is similar at the front of SP3, with or without cavitation. However, for 750 mm < x < 1000 mm, the contact pressure is lower without cavitation ( $P^0 = 1$  MPa), with pressure reaching values lower than -0.1 MPa. For 550 mm < x < 750 mm, the contact pressure is lower with cavitation ( $P^0 = 0.1$  MPa). This result shows that cavitation not only clips pressures below  $\mathcal{P} = -P^0$  but globally modifies the pressure distribution on the structure.

The contact pressure distribution on the impacting surface of SP3 at  $t^* = 0.5$  is also presented in Fig. 30. These results emphasise the three-dimensional aspect of cavitation. They also confirm that cavitation globally influences the pressure distribution acting on the structure during the simulations, upstream and downstream of the area where the phenomenon develops.



Fig. 30. Oblique water impact of SP3 with an initial horizontal velocity U = 34.5 m/s: contact pressure distribution on the impacting surface of SP3 at  $t^* = 0.5$  for different values of initial pressure  $P^0$ .  $P^0 = 0.1$  MPa corresponds to the reference case with cavitation and  $P^0 = 1$  MPa to a simulation without cavitation.



Fig. 31. Oblique water impact of SP3 with an initial horizontal velocity (a) U = 34.5 m/s, and (b) U = 45.2 m/s: evolution of the force coefficient as a function of the non-dimensional time  $t^*$  for different values of initial pressure  $P^0$ .  $P^0 = 0.1$  MPa corresponds to the reference case with cavitation and  $P^0 = 1$  MPa to a simulation without cavitation.

The evolution of the force coefficient  $C_f$  is presented in Fig. 31(a) for the two values of  $P^0$  and U = 34.5 m/s. Increasing  $P^0$  leads to a slight variation in the amplitude of the force coefficient between  $t^* = 0.3$  and  $t^* = 0.62$ . Between these instants,  $C_f$  is slightly lower for  $P^0 = 1$  MPa, *i.e.* when the pressure does not reach the saturation threshold during impact. This phenomenon can be explained by the fact that the suction phenomenon, which is not limited by pressure saturation when  $P^0 = 1$  MPa, reduces the total hydrodynamic forces exerted by the fluids on the structure. This phenomenon is more pronounced for a higher impact velocity, U = 45.2 m/s, for which the cavitation phenomenon is more pronounced (Fig. 31(b)). In this case, a clear difference between the numerical results is observed from  $t^* = 0.2$ . This difference remains practically constant between  $t^* = 0.4$  and  $t^* = 1$ , *i.e.* until the end of the impact:  $C_{fmax}^{P0=1} = 0.054$  and  $C_{fmax}^{P0=0.1} = 0.071$ .

The results of the numerical simulations have shown that cavitation has a global influence on the pressure distribution (not only in the low-pressure zone similar to a cavitation pocket). To experimentally verify that cavitation can locally favour the development of lower negative pressures, we suggest comparing the pressures measured at different impact velocities and scaled to the impact configuration U = 34.5 m/s according to the following equation:

$$P_{rescaled} = P_U \times \frac{34.5^2}{U^2},\tag{20}$$

where  $P_U$  is the pressure measured at a given probe for an initial horizontal impact velocity U. The rescaled pressures are expressed as a function of the non-dimensional penetration depth of the structure  $h^* = (t - t_i)V/L$ , where V is the vertical velocity of the structure, and L is its length. During an impact without cavitation, the hydrodynamic pressures are expected to be proportional to the square of the impact velocity. While it is possible to prevent or not cavitation during numerical simulations, this is not the



**Fig. 32.** Oblique water impacts of SP3 with different horizontal impact velocities: comparison of the pressures measured at probes (a) p17, (b) p13, (c) p9, and (d) p4, and scaled to the impact configuration U = 34.5 m/s. The quantities are expressed as a function of the non-dimensional penetration depth of the structure  $h^*$ .

case during experiments. Therefore, the proposed rescaling aims to deduce the changes in pressure that could be observed at higher impact velocities if cavitation had been prevented from the results obtained for a low-velocity test, during which cavitation does not develop.

The evolution of the pressure measured experimentally at probes p4, p9, p13 and p17 is presented in Fig. 32 for U = 21 m/s (without cavitation and rescaled), U = 26.8 m/s (without cavitation and rescaled) and U = 34.5 m/s (with cavitation). Aside from probe p17, the lowest pressures are measured for the case with cavitation, *i.e.* for U = 34.5 m/s. Note that, for U = 26.8 m/s, the measurements associated with probe p17 seem to indicate that cavitation is about to develop (Fig. 32(a)) although no cavitation pocket is visible on the underwater images captured during this impact. This phenomenon seems to result from a local interaction between the flow and this specific probe (not perfectly flush-mounted, see (Iafrati and Grizzi, 2019)). These tendencies are in agreement with those observed numerically and confirm that during this type of impact, at the rear of the specimens, cavitation can favour the development of lower negative (relative) pressures than without cavitation.

# 7. Conclusion

The impacts of two fuselage sections (SP2 and SP3) have been studied numerically based on the experimental campaigns of the European project SARAH. The present numerical method relies on the explicit solver *Radioss* and uses a CEL approach and a penalty method to deal with the fluid–structure interaction. Cavitation is accounted for by describing the mechanical consequences of this phenomenon, *i.e.* the pressure saturation and the change in mass density. This type of approach is referred to as a barotropic cavitation model.

The effect of the coupling stiffness and size of the fluid elements on the numerical results has been assessed. On the one hand, no clear convergence of the numerical results has been observed for the coupling stiffness. However, the value of coupling stiffness recommended by the *Radioss* user's manual ( $k_{c_0}$ ) seems to ensure (i) an evolution of the hydrodynamic load and (ii) a flow behaviour representative of the one observed experimentally. On the other hand, clear converging tendencies have been observed for fluid elements of  $l_f \leq 5$  mm in the impact zone.

The evolution of the hydrodynamic loads predicted by the numerical model has been compared to the experimental measurements for several horizontal impact velocities. The model showed a satisfying capability to predict the evolution of the hydrodynamic load for the impact configurations involving suction and cavitation phenomena. The numerical model does not reproduce ventilation, which influences the evolution of the impact loads. Therefore, the difference between the numerical and experimental results is more pronounced during the oblique water impacts involving a ventilation phenomenon. The comparison of the contact pressure distribution on the specimens and the underwater images recorded during the experiments helped to qualitatively estimate the evolution of the wetted surface and low-pressure zone at the rear of the specimens during the impacts. Despite these satisfying results, the numerical method is not fully representative of the physical phenomena involved during this type of impact. This could explain some of the differences observed between numerical and experimental results for the cases involving cavitation and ventilation.

The effect of the curvatures of the specimen on the numerical results has been studied. A more circular transversal crosssection reduces the size of the wetted surface, resulting in lower hydrodynamic loads during the impact. The longitudinal curvature influences the suction phenomenon and, by extension, the cavitation and ventilation phenomena. Indeed, a more pronounced longitudinal curvature leads to a more pronounced suction phenomenon and the development of cavitation earlier during the impact. Logically, a less pronounced longitudinal curvature reduces the magnitude of the suction phenomenon, thus leading to the development of a higher total hydrodynamic force during the impact.

The influence of cavitation on the hydrodynamic load has also been discussed. Cavitation seems to notably and globally influence the pressure distribution along the specimens, upstream and downstream of where the low-pressure zone starts developing. The suggested extrapolation of the experimental pressure results allowed us to highlight that during such oblique water impacts, cavitation could locally favour the development of lower negative relative pressures than without cavitation at the rear of the specimens.

Suction and cavitation phenomena can highly influence the outcome of a ditching. Therefore, it is relevant to consider such phenomena in the simulation of an aircraft ditching. The literature on the subject is currently pretty scarce, and this study is nearly a first step towards numerical simulations of higher accuracy. Future work will be dedicated to improving several aspects of the numerical model. Firstly, considering a more complex fluid model (for instance, explicitly describing phase change, using three distinct phases, with a non-diffuse interface between phases or considering surface tension and viscosity) could lead to numerical results more representative of the phenomena observed experimentally, in particular regarding the occurrence of ventilation. Secondly, in this study, the structure is considered rigid. Structural deformation has been shown to influence the hydrodynamic load and the flow behaviour during water impacts. Therefore, it would be necessary to consider a more complex structural model to tend towards more realistic ditching simulations. Finally, impact conditions (velocity and pitch angle) influence the hydrodynamic loads experienced by the structure. Moreover, ditching often happens in an agitated sea. Therefore, more realistic impact conditions could be achieved by considering an initially deformed free surface, *i.e.* by considering the effect of waves during an aircraft ditching.

# CRediT authorship contribution statement

**M. Goron:** Writing – review & editing, Writing – original draft, Visualization, Validation, Supervision, Software, Methodology, Investigation, Funding acquisition, Formal analysis, Conceptualization. **A. Tassin:** Writing – review & editing, Supervision, Funding acquisition, Conceptualization. **B. Langrand:** Writing – review & editing, Supervision, Methodology, Funding acquisition, Conceptualization. **E. Spinosa:** Writing – review & editing, Resources, Conceptualization. **A. Del Buono:** Writing – review & editing, Conceptualization. **N. Jacques:** Writing – review & editing, Supervision, Conceptualization. **T. Fourest:** Writing – review & editing, Supervision, Formal analysis, Conceptualization. **A. Iafrati:** Writing – review & editing, Resources, Conceptualization.

# Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

# Acknowledgements

This work has been supported by ISblue project, an Interdisciplinary graduate school for the blue planet (ANR-17-EURE-0015), and by a grant from the ONERA international mobility program and the French government under the program "Investissements d'Avenir" embedded in France 2030. Mathieu Goron acknowledges ONERA and IFREMER for funding his PhD scholarship.

#### Table A.5

Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: effect of the speed of sound in the water  $c_{sw}$  on the computation time. 128 CPUs have been used for all computations.

<i>c</i> <sub><i>sw</i></sub> (m/s)	200	500	1500
Computation time (DD-hh:mm:ss)	0-20:20:11	1-03:11:27	2-01:13:06



Fig. A.33. Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: evolution of the relative pressure  $\mathcal{P}$  and normalised mass density  $\rho/\rho_w^0$  obtained at probe  $p^9$  as a function of the non-dimensional time  $t^*$  for different values of speed of sound in the water  $c_{uw}$ .

# Table A.6

Oblique water impact of the SP3 with an initial horizontal velocity U = 45.2 m/s: effect of the speed of sound in the water  $c_{sw}$  on the computation time. 128 CPUs have been used for all computations.

<i>c<sub>sw</sub></i> (m/s)	200	500	1500
Computation time (DD-hh:mm:ss)	0-21:49:28	0-20:57:27	1-21:43:40

# Appendix A. Effect of the speed of sound in the water

The effect of the speed of sound in the water  $c_{sw}$  on the numerical results is investigated in this appendix to support the fact that it is possible to perform numerical simulations using a value of the speed of sound in the water smaller than the physical one (approximately  $c_{sw} = 1500 \text{ m/s}$ ) to decrease computation times. Indeed, in explicit simulations, the stable time step depends on the speed of sound in the medium (CFL condition). Therefore, using a smaller speed of sound value increases the stable time step of the simulation, thus decreasing the total computation time. The numerical results obtained for the SP3 oblique water impact, with  $U = \{21; 45.2\}$  m/s, a coupling stiffness  $k_{c0}$  (depending on the impact velocity) and a fluid element size of  $l_f = 5$  mm, are compared for different values of the speed of sound in the water:  $c_{sw} = \{200; 500; 1500\}$  m/s.

# A.1. Oblique water impact of specimen SP3 with an initial horizontal velocity U = 21 m/s

The computation time for each case is presented in Table A.5. The evolution of the relative pressure  $\mathcal{P} = P - P^0$  and normalised mass density  $\rho/\rho_w^0$  obtained at probe p9 for the different values of speed of sound considered is presented in Fig. A.33. The evolution of the force coefficient  $C_f$  for the different values of speed of sound considered is presented in Fig. A.34.

A.2. Oblique water impact of specimen SP3 with an initial horizontal velocity U = 45.2 m/s

The computation time for each case is presented in Table A.6. The evolution of the relative pressure  $\mathcal{P} = P - P^0$  and normalised mass density  $\rho/\rho_w^0$  obtained at probe p9 for the different values of speed of sound considered is presented in Fig. A.35. The evolution of the force coefficient  $C_f$  for the different values of speed of sound considered is presented in Fig. A.36.

Overall, for the oblique water impacts considered in this article, the precision loss regarding the evolution of the hydrodynamic pressure and force coefficient for the cases with  $c_{sw} = 500$  m/s, in comparison with  $c_{sw} = 1500$  m/s, is judged reasonable in light of the computation time reduction. Note that the effect of  $c_{sw}$  on the numerical results is slightly more pronounced with U = 45.2 m/s. This



Fig. A.34. Oblique water impact of the SP3 with an initial horizontal velocity U = 21 m/s: evolution of the force coefficient  $C_f$  as a function of the nondimensional time  $t^*$  for different values of speed of sound in the water  $c_{su}$ .



Fig. A.35. Oblique water impact of the SP3 with an initial horizontal velocity U = 45.2 m/s: evolution of the relative pressure  $\mathcal{P}$  and normalised mass density  $\rho/\rho_w^0$  obtained at probe  $p^9$  as a function of the non-dimensional time  $t^*$  for different values of speed of sound in the water  $c_{sw}$ .

could indicate an influence of  $c_{sw}$  on the "cavitation" phenomenon observed numerically. The results obtained with  $c_{sw} = 500$  m/s and  $c_{sw} = 200$  m/s are very similar in terms of hydrodynamic pressure, force coefficient and computation time. Therefore, it seems preferable to set  $c_{sw}$  such as  $c_{sw} > c_{sa}$  (where  $c_{sa}$  is the speed of sound in the air) to ensure that the stable time step is defined by the speed of sound in the water.

# Appendix B. Supplementary data

Supplementary material related to this article can be found online at https://doi.org/10.1016/j.jfluidstructs.2025.104322.

# Data availability

Data will be made available on request.



Fig. A.36. Oblique water impact of the SP3 with an initial horizontal velocity U = 45.2 m/s: evolution of the force coefficient  $C_f$  as a function of the non-dimensional time  $t^*$  for different values of speed of sound in the water  $c_{sw}$ .

#### References

- Battistin, D., Iafrati, A., 2003. Hydrodynamic loads during water entry of two-dimensional and axisymmetric bodies. J. Fluids Struct. 17 (5), 643–664. Bayliss, A., Turkel, E., 1982. Outflow boundary conditions for fluid dynamics. SIAM J. Sci. Stat. Comput. 3.
- Brennen, C.E., 1995. Cavitation and Bubble Dynamics. Oxford Engineering Science Series, Oxford University Press, New York.
- Casadei, F., Leconte, N., Larcher, M., 2011. Strong and weak forms of a fully non-conforming FSI algorithm in fast transient dynamics for blast loading of structures. In: EU Science Hub European Commission. Corfu, Greece, p. 20.
- Climent, H., Benítez, L., Rueda, F., Toso-Pentecôte, N., 2006. Aircraft ditching numerical simulation. In: Undefined. Hambourg, Germany, p. 16.
- Del Buono, A., 2022. Fully Non-Linear Potential Flow Model for Aircraft Ditching Applications (Ph.D. thesis). Roma Tre, Roma.
- Del Buono, A., Bernardini, G., Tassin, A., Iafrati, A., 2021. Water entry and exit of 2D and axisymmetric bodies. J. Fluids Struct. 103, 103269.
- Dellanoy, Y., Kueny, J., 1990. Two phase flow approach in unsteady cavitation modeling. In: ASME FED 98. Toronto, Ontario, Canada., pp. 153-158.
- EASA, 2021. Certification Specifications and Acceptable Means of Compliance for Large Aeroplanes. CS-25 Amendment 27.. Technical Report.
- Egerer, C., Hickel, S., Schmidt, S., Adams, N.A., 2013. Les of turbulent cavitating shear layers. In: Nagel, W.E., Kröner, D.H., Resch, M.M. (Eds.), High Performance Computing in Science and Engineering '13. Springer International Publishing, Cham, pp. 349–359.
- Elhimer, M., Jacques, N., El Malki Alaoui, A., Gabillet, C., 2017. The influence of aeration and compressibility on slamming loads during cone water entry. J. Fluids Struct. 70, 24–46.
- Faucher, V., Bulik, M., Galon, P., 2017. Updated VOFIRE algorithm for fast fluid-structure transient dynamics with multi-component stiffened gas flows implementing anti-dissipation on unstructured grids. J. Fluids Struct. 74, 64–89.
- Folden, T.S., Aschmoneit, F.J., 2023. A classification and review of cavitation models with an emphasis on physical aspects of cavitation. Phys. Fluids 35 (8).
- Frikha, S., Coutier-Delgosha, O., Astolfi, J.A., 2009. Influence of the cavitation model on the simulation of cloud cavitation on 2D foil section. Int. J. Rotating Mach. 2008, e146234.
- Fuster, D., 2019. A review of models for bubble clusters in cavitating flows. Flow, Turbul. Combust. 102 (3), 497-536.
- Goncalvès, E., 2014. Modeling for non isothermal cavitation using 4-equation models. Int. J. Heat Mass Transfer 76, 247-262.
- Goncalves, E., Patella, R.F., 2009. Numerical simulation of cavitating flows with homogeneous models. Comput. & Fluids 38 (9), 1682–1696.
- Goron, M., Langrand, B., Jacques, N., Fourest, T., Tassin, A., Robert, A., Chauveheid, D., 2023. Simulation of water entry-exit problems highlighting suction phenomena by coupled Eulerian-Lagrangian approach. Eur. J. Mech. B Fluids 100, 37–51.
- Hammani, I., 2020. Improvement of the SPH Method for Multiphase Flows Application to the Emergency Water Landing of Aircrafts (Ph.D. thesis). École centrale de Nantes.
- Iafrati, A., Grizzi, S., 2019. Cavitation and ventilation modalities during ditching. Phys. Fluids 31 (5), 052101.
- Iafrati, A., Grizzi, S., Olivieri, F., 2020. Experimental investigation of fluid-structure interaction phenomena during aircraft ditching. AIAA J. 59 (5), 1–14.
- Iafrati, A., Grizzi, S., Siemann, M.H., Benítez Montañés, L., 2015. High-speed ditching of a flat plate: Experimental data and uncertainty assessment. J. Fluids Struct. 55, 501–525.
- Iafrati, A., Olivieri, F., 2017. Definition of the Conditions for Guided Aircraft Ditching Tests. Technical Report SARAH Deliverable D5.1, Institute of Marine Engineering (CNR-INM), Rome, Italy, p. 24.
- Iafrati, A., Olivieri, F., Grizzi, S., Carta, F., Camassa, A., Sale, M., Spinosa, E., 2019. Report on Test Data for Double Curvature Rigid Specimen. Technical Report SARAH Deliverable D5.3, Institute of Marine Engineering (CNR-INM), Rome, Italy, p. 114.
- Jackson, K.E., Putnam, J.B., 2020. LS-DYNA<sup>®</sup> Water Ditching Simulation of a Fokker F28 Fellowship Aircraft. Technical report, U.S. Department of Transportation, FAA, p. 48.
- Judge, C., Troesch, A., Perlin, M., 2004. Initial water impact of a wedge at vertical and oblique angles. J. Engrg. Math. 48 (3), 279-303.
- Korobkin, A.A., Pukhnachov, V.V., 1988. Initial stage of water impact. Annu. Rev. Fluid Mech. 20 (Volume 20, 1988), 159-185.
- Langrand, B., Siemann, M.H., 2018. Full aircraft ditching simulation: a comparative analysis of advanced coupled fluid-structure computational methods. Int. Conf. Impact Load. Struct. Mater. 4.
- McBride, E.E., Fisher, L.J., 1953. Experimental Investigation of the Effect of Rear-Fuselage Shape on Ditching Behavior. Report 2929, NACA, Langley Aeronautical Laboratory, Langley Field, Virginia, USA, p. 36.
- Moore, M.R., Ockendon, H., Ockendon, J.R., Oliver, J.M., 2014. Capillary and viscous perturbations to Helmholtz flows. J. Fluid Mech. 742, R1.

Petitpas, F., Franquet, E., Saurel, R., Le Metayer, O., 2007. A relaxation-projection method for compressible flows. Part II: Artificial heat exchanges for multiphase shocks. J. Comput. Phys. 225 (2), 2214–2248.

Sarkar, P., 2019. Simulation of Cavitation Erosion by a Coupled CFD-FEM Approach (Ph.D. thesis). Université Grenoble Alpes.

Sarkar, P., Ghigliotti, G., Franc, J.-P., Fivel, M., 2021. Fluid-structure modelling for material deformation during cavitation bubble collapse. J. Fluids Struct. 106, 103370.

Saurel, R., Franquet, E., Daniel, E., Le Metayer, O., 2007. A relaxation-projection method for compressible flows. Part I: The numerical equation of state for the Euler equations. J. Comput. Phys. 223 (2), 822–845.

Saurel, R., Petitpas, F., Berry, R.A., 2009. Simple and efficient relaxation methods for interfaces separating compressible fluids, cavitating flows and shocks in multiphase mixtures. J. Comput. Phys. 228 (5), 1678–1712.

Siemann, M.H., Langrand, B., 2017. Coupled fluid-structure computational methods for aircraft ditching simulations: Comparison of ALE-FE and SPH-FE approaches. Comput. Struct. 188, 95–108.

Souli, M., Sigrist, J.-F., 2010. Interaction Fluide-Structure: Modélisation Et Simulation Numérique. Vol. 19, M'hamed Souli.

- Spinosa, E., Broglia, R., Iafrati, A., 2022. Hydrodynamic analysis of the water landing phase of aircraft fuselages at constant speed and fixed attitude. Aerosp. Sci. Technol. 130, 107846.
- Spinosa, E., Grizzi, S., Iafrati, A., 2024. High-speed ditching of double curvature specimens with cavitation and ventilation. http://dx.doi.org/10.48550/arXiv. 2408.06952, ArXiv Preprint (arXiv:2408.06952).

Sun, H., Faltinsen, O.M., 2007. The influence of gravity on the performance of planing vessels in calm water. J. Engrg. Math. 58 (1), 91-107.

Toso, N.R.S., 2009. Contribution to the Modelling and Simulation of Aircraft Structures Impacting on Water (Ph.D. thesis). Institute of Aircraft Design, Universität Stuttgart.

Valsamos, G., Larcher, M., Casadei, F., Faucher, V., 2015. Decoupled Formulation of Constraints on Adaptive Hanging Nodes in EUROPLEXUS. Publications Office of the European Union, LU.

Zhu, X., Faltinsen, O.M., Hu, C., 2006. Water entry and exit of a horizontal circular cylinder. J. Offshore Mech. Arct. Eng. 129 (4), 253-264.