



HAL
open science

Simulation of water entry–exit problems highlighting suction phenomena by coupled Eulerian–Lagrangian approach

Mathieu Goron, Bertrand Langrand, Nicolas Jacques, Thomas Fourest, Alan Tassin, Adrien Robert, Daniel Chauveheid

► **To cite this version:**

Mathieu Goron, Bertrand Langrand, Nicolas Jacques, Thomas Fourest, Alan Tassin, et al.. Simulation of water entry–exit problems highlighting suction phenomena by coupled Eulerian–Lagrangian approach. *European Journal of Mechanics - B/Fluids*, 2023, 100, pp.37-51. 10.1016/j.euromechflu.2023.02.006 . hal-04060396

HAL Id: hal-04060396

<https://ensta-bretagne.hal.science/hal-04060396>

Submitted on 24 Nov 2023

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.

Simulation of water entry-exit problems highlighting suction phenomena by coupled Eulerian-Lagrangian approach

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

^a*DMAS, ONERA, Lille, F-59014, France, mathieu.goron@onera.fr, bertrand.langrand@onera.fr, thomas.fourest@onera.fr*

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

^c*ENSTA Bretagne, UMR CNRS 6027, IRDL, Brest CEDEX 09, F-29806, France, nicolas.jacques@ensta-bretagne.fr,*

^d*Ifremer, RDT, Plouzané, F-29280, France, alan.tassin@ifremer.fr,*

^e*Altair Engineering France, Antony, 92160, France, arobert@altair.com, dchauveheid@altair.com*

Abstract

The present study aims to assess the possibility of describing suction using coupled Eulerian-Lagrangian approach. The water entry and subsequent exit of conical and hemispherical bodies is investigated numerically using the Finite Element simulation software *Radioss*. The numerical method relies on an explicit numerical scheme. An Eulerian and a Lagrangian formulation are considered for the fluid and the structure, respectively. The fluid-structure interaction is based on an immersed contact interface. Particular attention is given to the evolution of the hydrodynamic (positive and negative) force and wetted surface. The numerical results are compared to experimental results from the literature for different impact conditions (maximum velocity and penetration depth). The influence of several parameters of the numerical model is analysed to assess its robustness and improve the numerical results. The numerical model especially shows a satisfying ability to predict suction forces.

Keywords: Numerical simulations, fluid-structure interaction, coupled Eulerian-Lagrangian approach, suction, cone, hemisphere

1. Introduction

Hydrodynamic impact arises when a solid body and a liquid enter into contact due to their relative motion. The study of this phenomenon is motivated by various

11 applications such as hull slamming, spacecraft (capsule) landing in water, aircraft and
12 rotorcraft emergency water landing (ditching) [1, 2]. The hydrodynamic loads arising
13 during water impacts can be among the most severe loads to which the structure can be
14 subjected. Therefore, for some applications, hydrodynamic loads should be considered
15 during the structures' sizing and certification exposed to this kind of event.

16 The phenomena occurring during a hydrodynamic impact are well known for simple
17 impact conditions, such as the two-dimensional, vertical impact of a rigid body with a
18 simple geometry on a quiescent fluid, neglecting gravity. Analytical approaches, often
19 based on the seminal works of von Karman [3] and Wagner [4], have been developed to
20 analyse the pressure distribution acting on the structure. Numerical methods offer the
21 possibility to study these problems with fewer restrictions on the impact conditions.
22 Ribet et al. [5] studied oblique and high-speed impacts of a sphere, wedge and ellip-
23 soid using Lagrangian and ALE formulations, with comparison to experimental results.
24 Faucher et al. [6] studied the vertical water impact of a cylinder using a CEL approach
25 and an adapted anti-dissipative scheme for unstructured meshes. Their results com-
26 pared well with the experiments in terms of cylinder deformation.

27 However, with more complex impact conditions, such as the water entry and sub-
28 sequent exit of a structure or oblique impacts with a large horizontal velocity, more
29 complex hydrodynamic phenomena may appear. Among these hydrodynamic phenom-
30 ena we can cite suction forces [7, 8, 9, 10], cavitation [11], ventilation [12], aeration [13]
31 and air cushioning [14]. These hydrodynamic phenomena are difficult to model indi-
32 vidually with state-of-the-art analytical or numerical approaches. Piro and Maki [8]
33 numerically predicted the suction force during the two-dimensional water entry and
34 exit of rigid and elastic wedges. They used a Finite Volume Method (FVM) with an
35 ALE formulation and a Volume-Of-Fluid (VOF) method to model the interface be-
36 tween air and water. Tassin et al. [9] obtained similar results for the same case using
37 an analytical 2D approach. Del Buono et al. [15] used a hybrid Boundary Element
38 Method-Finite Element Method (BEM-FEM) to model the same two-dimensional wa-

39 ter entry-exit problems. The authors compared their numerical results to analytical
 40 and experimental data. They highlighted the influence of gravity on the evolution of
 41 the hydrodynamic force, which is particularly strong during the exit phase. Both of
 42 these studies are in good agreement with the reference results but had been limited to
 43 two-dimensional and simple cases. During an aircraft ditching, the mentioned hydro-
 44 dynamic phenomena may happen simultaneously and influence each other (see Fig. 1),
 45 the structure is three-dimensional and deformable, etc. Modelling these complex hydro-
 46 dynamic impacts can become challenging for the mentioned analytical and numerical
 47 approaches. Therefore, these approaches require further development to increase their
 48 robustness and use during design or certification procedures.

49 The present work focuses on the numerical modelling of suction forces. In a ditching
 50 context, suction forces develop because of a depression localised near the rear fuselage,
 51 where the first contact with the water occurs, as shown in Fig. 1. This phenomenon
 52 occurs due to the high horizontal velocity of the structure at impact and the longitudinal
 53 curvature of the fuselage. Recent numerical work showed that suction forces could affect
 54 the overall aircraft kinematics during ditching [16, 17]. It is thus crucial to consider
 55 this phenomenon when numerically modelling realistic industrial applications such as
 56 aircraft ditching.

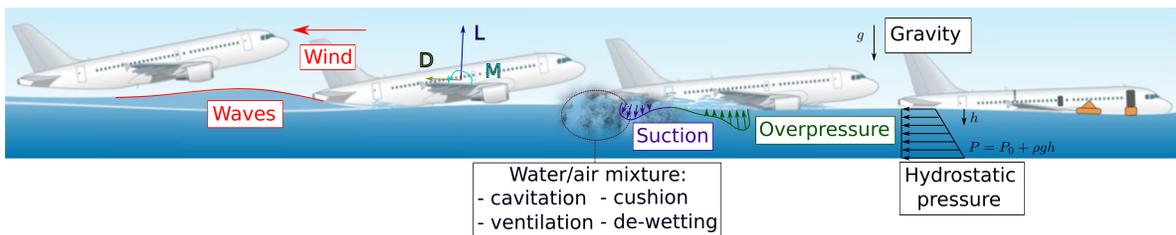


Figure 1: Illustration of the hydrodynamic phenomena occurring during an aircraft ditching.

57 Advanced numerical approaches are required to study realistic industrial applica-
 58 tions such as aircraft ditching. In the literature, the term *advanced numerical ap-*
 59 *proaches* generally refers to high-fidelity and coupled fluid and structure models using,
 60 for instance, explicit Finite Elements (FE) solvers. On the one hand, the structure
 61 is usually described with a Lagrangian approach. On the other hand, the fluid be-

62 haviour can be described using various frameworks: Eulerian, Arbitrary Lagrangian-
63 Eulerian (ALE) [18], or mesh-free methods such as Smoothed Particle Hydrodynamics
64 (SPH) [19, 20]. The coupling between the fluid and structural solutions is a key point
65 because the interdependence between the fluid flow and structural response is impor-
66 tant [21, 22].

67 Advanced numerical approaches have been widely employed to study hydrodynamic
68 impact problems. Stenius et al. [23, 24, 25] studied the hydroelastic response of flat
69 panels during vertical hydrodynamic impact. They used a CEL approach and penalty
70 method to model the fluid-structure interaction. The authors considered different im-
71 pact velocities, deadrise angles, structural masses and boundary conditions. They high-
72 lighted the influence of the structural behaviour on the hydrodynamic response by
73 comparing the structural response and hydrodynamic loading for rigid and hydroelas-
74 tic cases. N. Toso [16] studied the hydrodynamic impacts of spheres, cylinders, wedges,
75 a NACA body (as reported in [7]) and a full-scale helicopter sub-floor. The author
76 compared the experimental results to the results obtained with a FE method, a SPH
77 method and a combined SPH-FE approach. An overall good agreement between the
78 experiments and the simulations had been found, with more difficulty with modelling
79 the more complex cases, particularly in terms of pressure measures. M.H. Siemann and
80 B. Langrand [26] assessed the ability of SPH and Coupled Eulerian-Lagrangian (CEL)
81 methods to model the oblique impact of aircraft panels undergoing large deformations.
82 Recently, several studies [27, 28, 29, 30] presented simulations of complex water impact
83 problems, such as three-dimensional water entry of bodies with a complex shape and
84 aircraft ditching, based on the SPH method.

85 From the literature, it appears that several numerical approaches can deal with wa-
86 ter impact problems, from simple shapes to more realistic industrial applications. In
87 particular, CEL models are commonly applied to these problems. If their capacity to
88 describe impact forces (high positive pressures) is quite well established through de-
89 tailed comparisons with experiments, this is not the case when complex hydrodynamic

90 phenomena occur. Studies dedicated to the modelling of three-dimensional hydrody-
91 namic impacts involving suction forces and the related de-wetting process are scarce.
92 In the present work, the ability of the CEL method presented in [26] to model suction
93 is assessed. The issue of the evolution of the wetted surface is also considered.

94 Different experimental results taken from the literature [31] are used in this work to
95 assess the capacity of the computational method to model (i) suction forces and (ii) the
96 water exit phenomenon. These simple test cases consist of low-velocity water entry and
97 subsequent exit of different structures (cone, hemisphere) at different maximum impact
98 velocities ($U_{max} \in [0.4, 0.6]$ m/s). During the water entry and subsequent exit of the
99 structures, the vertical velocity varies from $-U_{max}$ (the structure enters the water)
100 to U_{max} (the structure exits the water). Suction forces are observed because of this
101 vertical velocity variation, with the maximum deceleration occurring at the end of the
102 entry stage. The numerical results are compared to the experimental results in terms
103 of hydrodynamic force and wetted surface radius. The numerical approach is used to
104 model a "simple" axisymmetric case using a three-dimensional formulation to assess
105 its capacity to model suction loads before modelling more complex test cases closer to
106 an aircraft ditching. Another relevant aspect of these test cases concerns the velocities
107 considered. The order of magnitude of the velocities is close to the ones specified by the
108 airworthiness authorities under aircraft ditching regulations. Indeed, during ditching,
109 the airspeed is set to achieve the minimum rate of descent at touchdown. For example,
110 the certification of the Airbus A320 relies on tests with a descent rate of approximately
111 1 m/s [32]. In [17], the ditching of a generic rigid aircraft body had been modelled with
112 a structural vertical velocity of 1.5 m/s.

113 This paper is organised as follows. Section 2 provides a brief presentation of the
114 water entry-exit experiments, a description of the associated numerical model and the
115 analysis methods. The effect of some key numerical parameters on the numerical results
116 is presented in Section 3. Section 4 compares the numerical and experimental results.
117 Finally, conclusions are drawn and orientations for future research are discussed in

119 2. Water entry and subsequent exit problems and associated computational 120 models

121 The present work focuses on the water entry and subsequent exit experiments de-
122 scribed in [31]. A brief presentation of the experiments is given hereunder. Then
123 follows a description of the fundamentals of the adopted fluid-structure method. Fi-
124 nally, the methods used to obtain the hydrodynamic force and wetted surface radius
125 are presented.

126 2.1. Description of the experiments

127 The water entry and subsequent exit experiments have been conducted in a water
128 tank with a 6 Degrees-Of-Freedom (DOF) motion generator [31]. The 6-DOF motion
129 generator enforces the displacement of the mock-up (vertical translation). During the
130 experiments, the vertical velocity varies from $-U_{max}$ (the structure enters the water)
131 to U_{max} (the structure exits the water). The maximum impact velocity reached during
132 the experiments (U_{max}) ranges from 0.4 to 0.6 m/s. The vertical position of the lowest
133 point of the mock-up is defined by the equation $z = h(t)$ and the function $h(t)$ defined
134 as:

$$\begin{cases} h(t) = -H \sin(2\pi(t - t^0)/T) + \delta_z, & t^0 \leq t \leq T/2 + t^0 \\ \dot{h}(t) = U_{max}, & t \geq T/2 + t^0 \end{cases} \quad (1)$$

135 where H is the maximum submergence depth, t^0 is the instant when the structure starts
136 decelerating (see Fig. 7), $T = 2\pi H/U_{max}$ is the period of the structure kinematics, U_{max}
137 is the maximum velocity, and $\delta_z = 3$ mm is a small parameter introduced in [31] a
138 posteriori to compensate for a vertical offset of the mock-up during the experiments.
139 The acceleration is maximum at the transition between the entry and the exit phase
140 (at $t = T/4 + t^0$).

141 Transparent mock-ups made of polymethyl methacrylate (PMMA) and LED edge-
 142 lighting techniques have been used to improve the visualisation of the wetted surface
 143 (see Fig. 2 and 3), as described in [33].

144 *2.2. Numerical model*

145 The present hydrodynamic impact problem is modelled using the explicit solver
 146 *Radioss*, developed by Altair. The structures and fluid domains are three-dimensional.
 147 However, only a quarter of the impact problem is modelled because the problem is
 148 axisymmetric. This reduces the size of the model and the associated computation time.
 149 Moreover, it was checked that the use of symmetry conditions does not affect the results
 150 of the simulations. The computations have been performed using a cluster available at
 151 ONERA, whose characteristics are given in Table 1.

Central Processing Unit (CPU) type	Intel(R) Xeon(R) CPU E5-2650 v4
Frequency (GHz)	2.20
RAM (GB)	128
Number of CPUs	64
Computing mode	Distributed memory, double precision

Table 1: Description of ONERA’s cluster: hardware and computational settings.

152 *2.2.1. Structure modelling*

153 The dimensions of the mock-ups are given in Fig. 2 and 3 [31]. The structure is dis-
 154 cretized using Mindlin-Reissner four-nodes bi-linear shell elements of 15 mm thickness.
 155 The characteristic structural element size is $10 \times 10 \text{ mm}^2$. The normal of the structural
 156 elements is oriented outward (toward the water). The structure is modelled as a rigid
 157 body: the nodes of the structure are kinetically linked to a primary node (see Fig. 4).

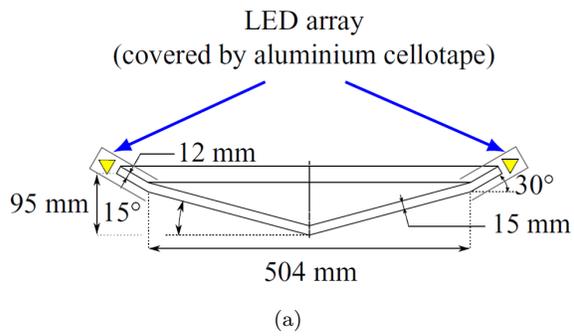


Figure 2: (a) Sketch and (b) photo of the conical mock-up used in the experiments of [31].

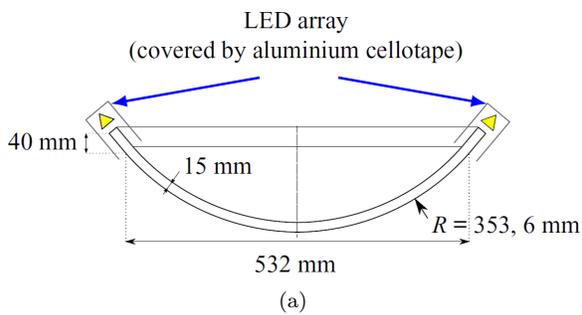


Figure 3: (a) Sketch and (b) photo of the hemispherical mock-up used in the experiments of [31].

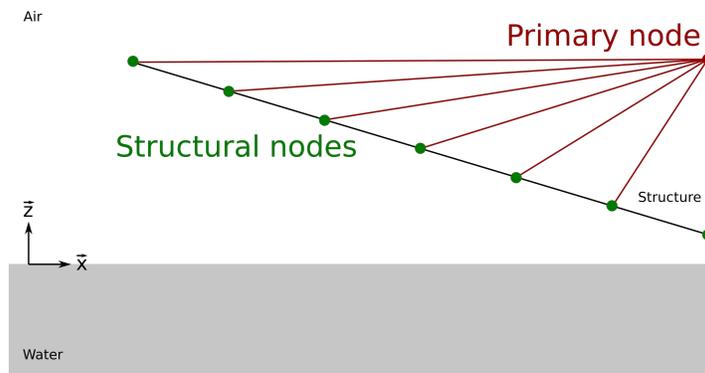


Figure 4: Illustration of the rigid structure model. The kinematic links between the structural nodes (green points ●) and the primary node (red point ●) are represented by the red lines -.

158 *2.2.2. Fluid modelling*

159 The fluid flow is described by an Eulerian multi-material formulation (*Radioss* law
 160 51). Fluid viscosity and surface tension effects are neglected, and adiabatic conditions
 161 are assumed. The validity of these assumptions in the present case is discussed in
 162 section 5.1 of reference [31]. Two phases are considered, air and water. The interface
 163 between the different phases is a diffuse zone. The fluid mixture is modelled through
 164 a so-called six-equation model as described in [34]. The transport equation for the air
 165 volume fraction α_a is given by:

$$\frac{\partial \alpha_a}{\partial t} + \vec{V} \cdot \nabla \alpha_a = 0 \quad (2)$$

166 where \vec{V} is the fluid velocity. The water volume fraction is then obtained by $\alpha_w = 1 - \alpha_a$.
 167 The evolution of the mass density for each phase is given by Eq. (3) and (4):

$$\frac{\partial(\alpha_a \rho_a)}{\partial t} + \text{div}(\alpha_a \rho_a \vec{V}) = 0 \quad (3)$$

$$\frac{\partial(\alpha_w \rho_w)}{\partial t} + \text{div}(\alpha_w \rho_w \vec{V}) = 0 \quad (4)$$

168 where ρ_a and ρ_w are the air and water mass density, respectively. A single velocity field
 169 is used to describe the motion of the different phases:

$$\frac{\partial(\rho \vec{V})}{\partial t} + \text{div}(\rho \vec{V} \otimes \vec{V}) + \nabla P = 0 \quad (5)$$

170 where $\rho = \alpha_a \rho_a + \alpha_w \rho_w$ is the mass density of the mixture and P is an equilibrium
 171 pressure to be determined later on. The specific internal energies of the air (e_a) and
 172 water (e_w) are given by Eq. (6) and (7), respectively:

$$\frac{\partial(\alpha_a \rho_a e_a)}{\partial t} + \text{div}(\alpha_a \rho_a e_a) + \alpha_a P_a \text{div} \vec{V} = 0 \quad (6)$$

$$\frac{\partial(\alpha_w \rho_w e_w)}{\partial t} + \text{div}(\alpha_w \rho_w e_w) + \alpha_w P_w \text{div} \vec{V} = 0 \quad (7)$$

173 where P_a and P_w are the pressure of the air and water, respectively. Eq. (2), (3), (4),
 174 (5), (6) and (7) are closed by two equations of state (one for each phase). The air
 175 behaviour is modelled using an ideal gas equation of state:

$$P_a = (\gamma_a - 1) \rho_a e_a \quad (8)$$

176 where γ_a is the heat capacity ratio for air at ambient temperature. The values of the
 177 mentioned parameters are synthesised in Table 2. The water behaviour is modelled
 178 using a stiffened gas equation of state:

$$P_w = (\gamma_w - 1) \rho_w e_w - \gamma_w P^* \quad (9)$$

179 where γ_w is the heat capacity ratio for water, P^* is a pressure coefficient ensuring a
 180 stable value of the speed of sound in the medium c_s , thus of the water compressibility,
 181 regardless of the pressure variation. In practice, Eq. (10) below is used to define P^* .
 182 The values of the mentioned parameters are synthesised in Table 3.

$$P^* = \frac{\rho_w^0 c_s^2}{\gamma_w} \quad (10)$$

183 The equilibrium pressure P used in Eq (5) is computed as follows. The air and
 184 water masses are computed for given values of α_a , α_w , ρ_a , ρ_w in an element:

$$\begin{aligned} m_a &= \alpha_a \rho_a, \\ m_w &= \alpha_w \rho_w \end{aligned} \quad (11)$$

185 Then the values for P , e_a , e_w , ρ_a , ρ_w , described by the system of five equations
 186 (Eq. (12)), are computed using a Newton-Raphson iterative method and considering
 187 m_a and m_w constant:

$$\left\{ \begin{array}{l} \frac{m_a}{\rho_a} + \frac{m_w}{\rho_w} - 1 = 0, \\ e_a - e_a^0 + P \cdot \left(\frac{1}{\rho_a} - \frac{1}{\rho_a^0} \right) = 0, \\ e_w - e_w^0 + P \cdot \left(\frac{1}{\rho_w} - \frac{1}{\rho_w^0} \right) = 0, \\ P_a(\rho_a, e_a) = P, \\ P_w(\rho_w, e_w) = P \end{array} \right. \quad (12)$$

188 The spatial discretization of the momentum balance equation (Eq. (5)) is based on
189 a Lagrange-projection method [34]. The Lagrange step is dealt with using a Monotonic
190 Upstream-Centered Scheme for Conservation Laws (MUSCL). This second-order ad-
191 vection scheme reduces the diffusion problem at the interface between the two phases
192 (*i.e.* air and water). Fluid nodes with an initial vertical position $z_0 \geq 0$ mm are initially
193 located in the air sub-domain. Similarly, fluid nodes with an initial vertical position
194 $z_0 \leq 0$ mm are initially located in the water sub-domain. As mentioned previously,
195 only a quarter of the fluid domain is modelled. Symmetry conditions are applied to the
196 symmetry planes of the model. For the reference mesh, the size of the fluid elements is
197 equal to $2.5 \times 2.5 \times 2.5$ mm³ near the structure (impact zone). The structure is located
198 within the impact zone at every instant of the computation. The dimensions of the
199 fluid domain are chosen large enough to avoid border effects (*e.g.* reflections of pres-
200 sure waves) and are given in Fig. 5. The fluid domain is discretized with 8 140 520 3D
201 continuum 8-node elements with one integration point. Other meshes have also been
202 built, and the influence of the size of the fluid elements in the impact zone is discussed
203 in Section 3.3.

204 In the air domain, the air fraction is initialised to a value $\alpha_{air} = 1$ and the water
205 fraction to $\alpha_{water} = 0$. In the water domain, the air fraction is initialised to a value
206 $\alpha_{air} = \epsilon$ and the water fraction to $\alpha_{water} = 1 - \epsilon$, with $\epsilon = 10^{-4}$. Introducing a small
207 fraction of air into the water guarantees $P > 0$ and $c_s^2 > 0$, thus it guarantees the
208 hyperbolicity of the problem.

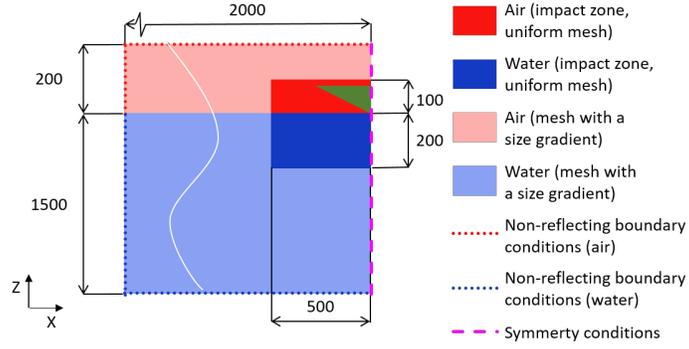


Figure 5: Dimensions and mesh of the fluid domain (in mm). Outside the impact zone, the size of the fluid elements scales with a factor of 1.2. The cone is represented in green.

Parameters	Values
γ_a	1.4
ρ_a^0	$1.22 \cdot 10^{-6} \text{ g/mm}^3$
P_a^0	0.101325 MPa

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

Parameters	Values
γ_w	4.4
ρ_w^0	$1.0 \cdot 10^{-3} \text{ g/mm}^3$
P_w^0	0.101325 MPa
c_s	1500 m/s

Table 3: Parameters for the water equation of state: stiffened gas.

209 *2.2.3. Fluid-structure interaction*

210 The fluid-structure interaction is modelled using a "weak" coupling approach [22].
 211 The structural Lagrangian elements (primary elements) are immersed in the Eulerian
 212 fluid grid (secondary nodes). The structure and fluid domains are meshed indepen-
 213 dently and superimposed. The coupling algorithm uses an influence zone defined over
 214 a distance h_c in the direction normal to the structure (see Fig. 6). When a fluid node is
 215 detected inside the influence zone, a coupling force is applied to it. The coupling force
 216 is computed using Eq. (13):

$$F = \frac{k_c}{h_c} d \cdot \tilde{d} \quad (13)$$

217 where h_c is the contact height, k_c is the contact stiffness, d is the penetration of a fluid
 218 node inside the influence zone of the structure and \tilde{d} is the displacement of a fluid node
 219 once it is detected inside the influence zone of the structure (see Fig. 6). d and \tilde{d} are
 220 computed using Eq. (14) and (15), respectively:

$$d = \max(0, h_c - |(\vec{r}_{fluid} - \vec{r}_{lag}) \cdot \vec{n}|) \quad (14)$$

221 where \vec{r}_{fluid} is the position of the projected fluid node on the Lagrangian surface, \vec{r}_{lag}
 222 is the position of the Lagrangian node.

$$\begin{cases} \frac{d\tilde{d}}{dt} = (\vec{V}_{fluid} - \vec{V}_{lag}) \cdot \vec{n}, & \text{if } d > 0 \\ \frac{d\tilde{d}}{dt} = 0, & \text{if } d \leq 0 \end{cases} \quad (15)$$

223 where \vec{V}_{fluid} is the velocity of the fluid node, \vec{V}_{lag} is the velocity of the structure. The
 224 value of the displacement is null at the instant t^i when the fluid node enters the influence
 225 zone, *i.e.* $\tilde{d}(t^i) = 0$. The *Radioss* documentation suggests to use the following values
 226 for h_c and k_c :

$$h_c = 1.5 \times l_f \quad (16)$$

$$k_c = \frac{\rho_w^0 U_{max}^2 S_{el}}{h_c} \quad (17)$$

227 where l_f is the size of the fluid elements in contact with the structure, U_{max} is the
 228 structure maximum velocity and S_{el} is the mean surface of the structural elements.

229 This method has been applied to different hydrodynamic impact problems, namely
 230 the vertical impact of a wedge [35], the ditching of deformable fuselage sections [26]
 231 and the ditching of a helicopter [36].

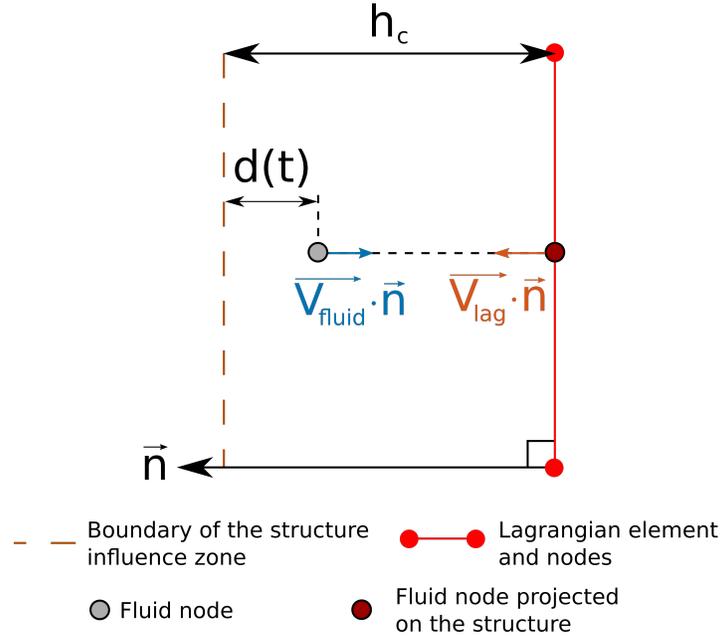


Figure 6: Illustration of the penetration of a fluid node inside the influence zone of the structure, $d(t)$. The relative velocity of the fluid node regarding the Lagrangian (structural) element is $(\vec{V}_{fluid} - \vec{V}_{lag}) \cdot \vec{n}$.

232 2.2.4. Initial and boundary conditions

233 It has been observed that the water is pushed down and significant contact forces are
 234 observed before the structure reaches the undisturbed water level. This phenomenon
 235 is due to the contact algorithm used in the present numerical model (presence of the
 236 structure influence zone, see Section 2.2.3). To compensate for this phenomenon, an
 237 additional vertical offset equal to h_c is given to the structure in the numerical simu-
 238 lations. Therefore, the bottom boundary of the influence zone of the structure in the
 239 simulations is at the same position as the (physical) structure in the experiments (see

240 Fig. 7). Hence, the motion of the structure in the simulations is prescribed according
 241 to Eq. (18), which is obtained by adding the term h_c to Eq. (1):

$$\begin{cases} h(t) = -H \sin(2\pi(t - t^0)/T) + \delta_z + h_c, & t^0 \leq t \leq T/2 + t^0 \\ \dot{h}(t) = U_{max}, & t \geq T/2 + t^0 \end{cases} \quad (18)$$

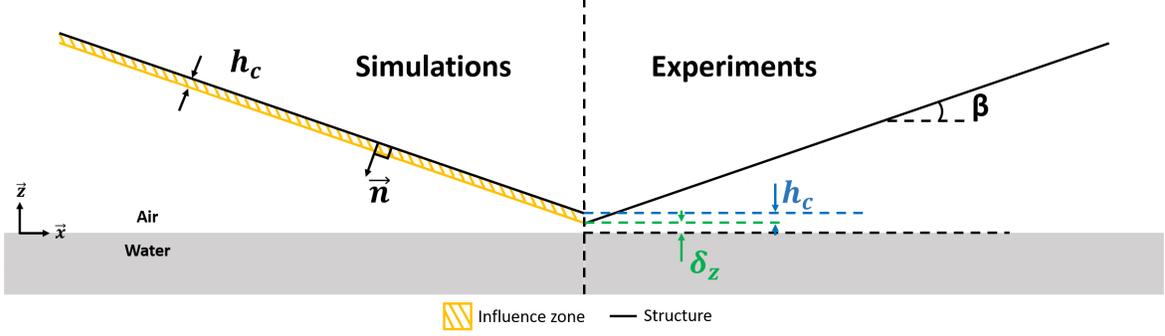


Figure 7: Position of the structure in the numerical simulations (left) and the experiments (right) at t^0 , *i.e.* at the time when the structure starts decelerating. h_c is the contact height, δ_z is the parameter introduced a posteriori in the experiments, and β is the deadrise angle of the cone.

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

$$P = P^0 + \rho^0 g z \quad (19)$$

245 where $P^0 = 0.101325 \text{ MPa}$ is the initial pressure at $z = 0$, g is the gravity acceleration,
 246 ρ^0 is the initial fluid mass density and z is the vertical coordinate (recall that $z = 0 \text{ mm}$
 247 corresponds to the initial air-water interface).

248 At the boundaries of the fluid domain corresponding to symmetry planes, the veloc-
 249 ity in the direction normal to the fluid domain is set at zero. Non-reflecting boundary
 250 conditions, based on the pressure formulation given in [37], are applied to the other
 251 boundaries of the fluid domain.

252 *2.3. Results analysis*

253 For the sake of the numerical approach validation, the numerical results are com-
 254 pared to the experimental ones in terms of non-dimensional hydrodynamic force and
 255 wetted surface radius. The non-dimensional hydrodynamic, f_{adim} , force is defined as:

$$f_{adim} = \frac{F_z}{\rho U_{max}^2 S} \quad (20)$$

256 where F_z is the vertical hydrodynamic force acting on the rigid body due to the fluid-
 257 structure coupling algorithm. $S = \pi R^2$ is the projected area of the structure and R is
 258 the structure radius.

259 The relative difference between the experimental and numerical maximum and min-
 260 imum values of the non-dimensional forces, respectively $\Delta f_{adim \ max}$ and $\Delta f_{adim \ min}$, are
 261 defined as:

$$\Delta f_{adim \ max} = \frac{f_{adim \ exp \ max} - f_{adim \ num \ max}}{f_{adim \ exp \ max}} \quad (21a)$$

$$\Delta f_{adim \ min} = \frac{f_{adim \ exp \ min} - f_{adim \ num \ min}}{f_{adim \ exp \ min}} \quad (21b)$$

262 The wetted surface radius analysis requires a specific post-processing operation of
 263 the computational results. For this purpose, the air volume fraction in the fluid elements
 264 at a symmetry plane is monitored (see Fig. 8a). Then, the iso-line corresponding to a
 265 volume fraction of 0.5 (assumed to correspond to the position of the air-water interface)
 266 is extracted from this data. The wetted surface radius is taken as the radial position of
 267 the highest point of this iso-line (see Fig. 8b).

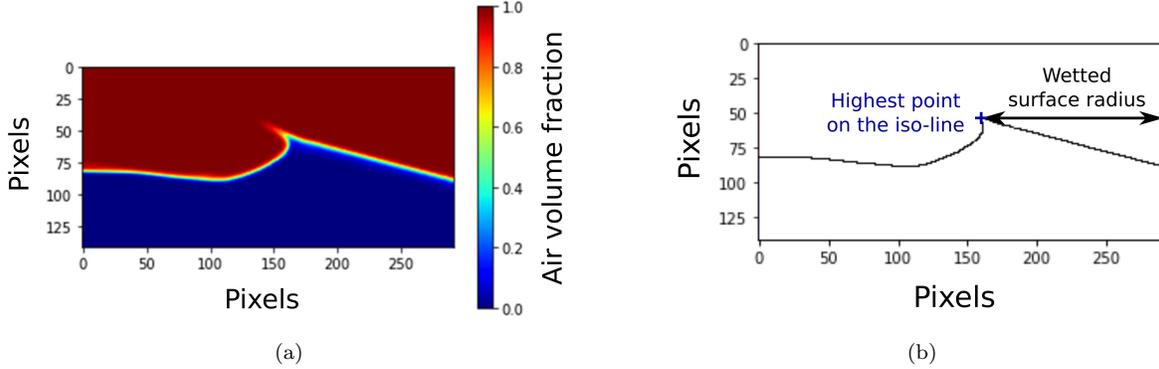


Figure 8: Water entry and subsequent exit of the cone at $t = 245$ ms. (a) Visualisation of the volume fraction in the fluid elements at a symmetry plane. (b) Extraction of the iso-line corresponding to a volume fraction of 0.5, of the position of the highest point on this iso-line (blue cross +) and of the wetted surface radius.

268 The evolution of the pressure has been monitored numerically at different locations
 269 of the structures. The position of the pressure gauges are described for the cone and
 270 the hemisphere in Fig 9a and Fig. 9b respectively. Note that the pressure gauges
 271 are located outside the influence zone of the structures. Indeed, due to the coupling
 272 method used in the present study (penalty method), the pressures obtained inside the
 273 influence zone of the structure are noisy and difficult to analyse. Inside the influence
 274 zone of the structure, the fluid elements contain an air-water mixture. It means that
 275 the volume fraction of the elements in the influence zone rapidly oscillates over time,
 276 between $\rho_w^0 = 10^{-3}$ g/mm³ and $\rho_a^0 = 10^{-6}$ g/mm³. The oscillations (noise) of the
 277 volume fraction induce the oscillations of the pressure results. The numerical results
 278 are presented in terms of pressure coefficient, p_{adim} (see Eq. (22)). The numerical
 279 pressure results are not compared to experimental results because pressures had not
 280 been measured during the experiments.

$$p_{adim} = \frac{P - P^0}{\rho_w^0 U_{max}^2} \quad (22)$$

281 where P is the pressure measured by the gauges and P^0 is the initial pressure.

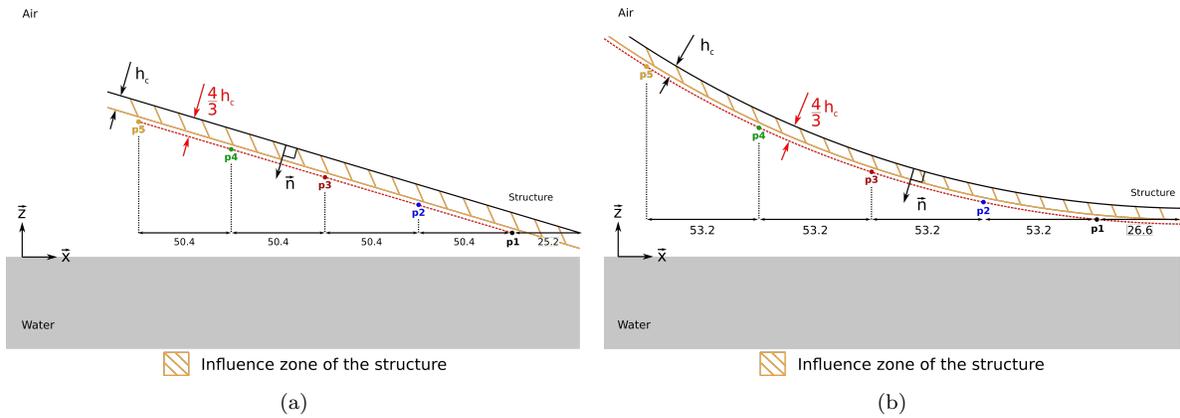


Figure 9: Illustration (not to scale) of the pressure gauges position for (a) the cone and (b) the hemisphere. The distance between the gauges are given in millimetres.

282 3. Investigation of different numerical parameters affecting the water entry 283 and subsequent exit simulations

284 The effect of several simulation parameters had been studied in [35] for the water
285 entry of a wedge. This study helped us define guidelines for the simulation of hy-
286 drodynamic impacts with the numerical method presented in this paper. It has been
287 noted that the fluid elements near the impacting structure (impact zone) should be at
288 least two times smaller than the structure elements to ensure fluid-structure interac-
289 tion continuity. The size of the fluid elements outside the impact zone can be increased
290 gradually to reduce the number of fluid elements in the model (a factor of 1.2 between
291 the sizes of two adjacent elements has been found to be suitable). The impact zone
292 dimensions should be equal to twice the structure dimensions. Also, the dimensions of
293 the (entire) fluid domain should be large enough to avoid boundary effects. Following
294 this preliminary work, the influence of different model parameters is studied in this sec-
295 tion. The following points are discussed: the effect of the speed of sound in the water,
296 the effect of the contact stiffness (used for the fluid-structure interaction) and the effect
297 of the size of the fluid elements in the impact zone (around the structure). The results
298 are presented for the water entry and subsequent exit of the cone with $c_{max} = 250$ mm
299 (maximum wetted surface radius) and $U_{max} = 0.6$ m/s. The maximum wetted surface
300 radius corresponds to the theoretical value obtained with the Wagner theory when the

301 penetration depth is maximum (see section 2.4 in [31]).

302 3.1. *Effect of the speed of sound in the water*

303 As presented in Section 2.2.2, the water equation of state depends on the speed of
304 sound in the medium (c_s), among other parameters. In explicit simulations, the stable
305 time step depends on the sound celerity. Therefore, it can be beneficial in terms of
306 computation time to use a speed of sound value smaller than the real value for the water
307 (approximately $c_s = 1500$ m/s). Simulations have been performed with different speed
308 of sound values: $c_s = \{200; 500; 1500\}$ m/s (other parameters remaining identical).
309 The results in terms of hydrodynamic force are presented in Fig. 10. The results show
310 that decreasing the value of c_s in the computation leads to a slight reduction of the
311 absolute force value. This maximum value is reached at the transition between the
312 entry and exit stages (note that the force is negative at this time). A decrease of 2.3%
313 and 5.5% is observed for $c_s = 500$ m/s and $c_s = 200$ m/s, respectively, in comparison
314 to the case with $c_s = 1500$ m/s. The computation times are given in Table 4. The
315 computation time needed to complete this simulation with $c_s = 1500$ m/s is more than
316 3 times higher than with $c_s = 500$ m/s. The loss of accuracy in terms of hydrodynamic
317 peak force predictions ($\simeq 2.3\%$) had been considered reasonable concerning the gain
318 in computation time (3 times less). The results presented in the following paper have
319 been obtained with $c_s = 500$ m/s.

320 An alternative linear polynomial equation of state has also been tested to describe
321 the water behaviour (see Eq. (23)). The results obtained with both equations of state
322 are very similar (for the same values of c_s). The stiffened gas equation of state (Eq. (9))
323 is used in the following simulations.

$$P = P^0 + \rho_w c_s^2 \quad (23)$$

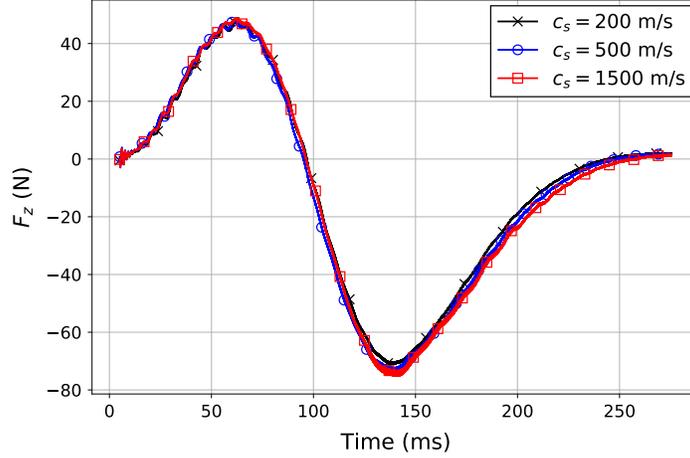


Figure 10: Time history of the hydrodynamic force during the water entry and subsequent exit of the cone, with $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s, for several values of the speed of sound in the water: $c = \{200, 500, 1500\}$ m/s.

c_s (m/s)	200	500	1500
Computation time (DD-hh:mm:ss)	2-08:53:01	3-11:02:05	10-11:27:38

Table 4: Simulation of the water entry and subsequent exit of the cone, with $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s. Comparison of the computation time for $c_s = \{200; 500; 1500\}$ m/s.

324 3.2. Effect of the contact stiffness

325 In this section, the effect of the contact stiffness parameter k_c on the hydrodynamic
326 force is analysed. The numerical results obtained for the water entry and subsequent
327 exit of the cone, with a fluid element size of $l_f = 2.5$ mm, are presented in Fig. 11 for
328 different values of k_c .

329 With the recommended value, $k_{c0} = 0.0096$ N/mm, obtained using Eq. (17), the
330 force time history presents some low-frequency oscillations during the exit stage, start-
331 ing when the force reaches its minimum value. Increasing the value of k_c reduces these
332 oscillations and the amplitude of the negative peak force. For $k_c \geq 4 \times k_{c0}$, the force
333 signal does not oscillate anymore, but the negative peak force value is lower than for
334 higher values of k_c . Convergence of the numerical results is achieved for $k_c \geq 8 \times k_{c0}$.
335 The amplitude of the positive peak force is less influenced by the value of k_c than the
336 amplitude of the negative peak force. The cone results presented in Section 4 have been
337 obtained with a value of $k_c = 16 \times k_{c0} = 0.1536$ N/mm.

338 A similar convergence study has been carried out for the hemisphere case. The

339 convergence of the results has been obtained for $k_c = 160 \times k_{c_0} = 1.536 \text{ N/mm}$. This
 340 higher stiffness value is explained by the difference in geometry. Indeed, the flat bottom
 341 of the hemisphere leads to a more rapid increase of the hydrodynamic force during
 342 the first instants of the impact, requiring a higher contact stiffness for the numerical
 343 modelling. The hemisphere results presented in Section 4 have been obtained with a
 344 value of $k_c = 1.536 \text{ N/mm}$.

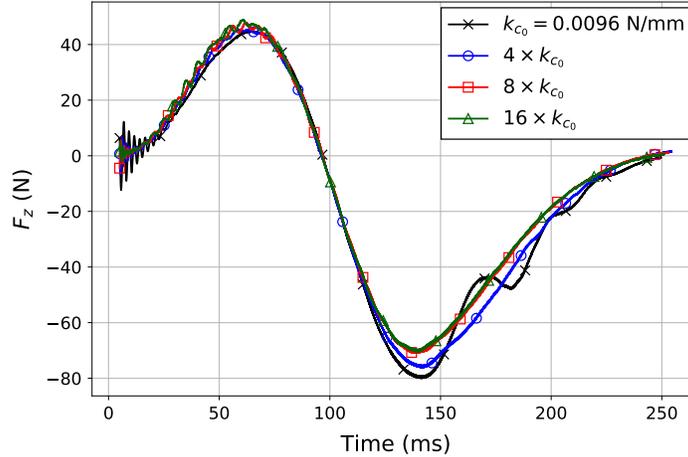


Figure 11: Time history of the hydrodynamic force during the water entry and subsequent exit of the cone, with $c_{max} = 250 \text{ mm}$ and $U_{max} = 0.6 \text{ m/s}$. Different contact stiffness are tested. The contact stiffness $k_{c_0} = 0.0096 \text{ N/mm}$ is obtained when using Eq. (17).

345 3.3. Effect of the size of the fluid elements in the impact zone

346 In this section, the influence of the size of the fluid elements on the numerical
 347 results is investigated. Note that, as the present numerical model uses an explicit
 348 solver, the stable time-step of the computation also depends on the size of the fluid
 349 elements (via the CFL condition). The water entry and subsequent exit of the cone,
 350 with $c_{max} = 250 \text{ mm}$ and $U_{max} = 0.6 \text{ m/s}$, is simulated using different sizes of fluid
 351 elements in the impact zone, $l_f = \{2; 2.5; 3; 4; 5; 7.5\} \text{ mm}$. To study the influence of
 352 the size of the fluid elements on the numerical results, in this section only, the vertical
 353 position of the structure is defined by Eq (1). The cone velocity is equal to 0.6 m/s
 354 until it reaches the level $z = \delta_z$. The contact stiffness is set to $k_c = 0.1536 \text{ N/mm}$ for
 355 all the fluid element sizes considered here. The results in terms of hydrodynamic forces
 356 are presented in Fig. 12.

357 The computation times corresponding to each numerical simulation are provided in
 358 Table 5. The computational time is larger when the fluid elements are smaller. Indeed,
 359 the number of fluid elements is higher, and the stable time step is smaller (higher
 360 number of time steps to reach the same physical time).

361 From Fig. 12, it is observed that the duration of the stages where the force is positive
 362 and negative is independent of the size of the fluid elements. However, the maximum
 363 and minimum values of the hydrodynamic force obtained numerically decrease with l_f .
 364 Indeed, the contact height h_c is proportional to l_f . Therefore, fewer fluid nodes interact
 365 with the structure, at a given time, for a lower value of l_f . Note that, in terms of force
 366 amplitude, the exit phase is slightly more sensitive to the size of the fluid elements than
 367 the entry phase. Indeed, the maximum hydrodynamic force obtained for $l_f \leq 3$ mm are
 368 similar (difference lower or equal to 1.6 N). A more pronounced difference is observed
 369 for the minimum hydrodynamic force obtained for $l_f \leq 3$ mm (difference lower or equal
 370 to 3.1 N).

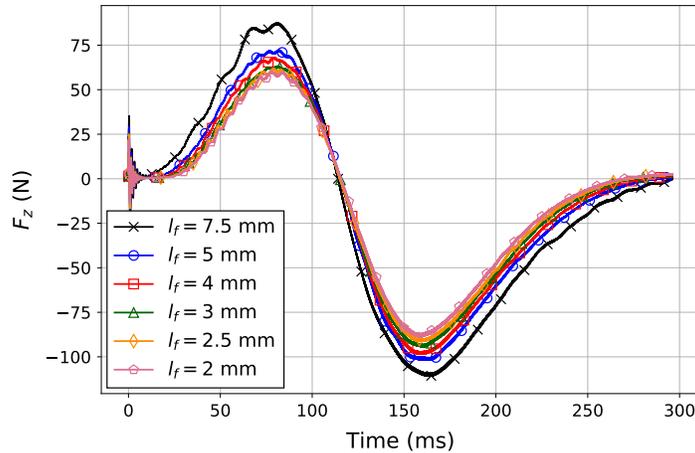


Figure 12: Time history of the hydrodynamic force during the water entry and subsequent exit of the cone, with $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s. The considered sizes of the fluid elements in the impact zone are $l_f = \{2; 2.5; 3; 4; 5; 7.5\}$ mm.

l_f (mm)	7.5	5	4	3	2.5	2
Total number of fluid finite elements	526 176	1 376 160	2 397 008	5 102 208	8 140 520	14 682 662
Computation time (DD-hh:mm:ss)	04:19:34	19:05:12	1-01:03:24	2-10:34:12	3-11:02:05	12-02:31:53

Table 5: Simulation of the water entry and subsequent exit of the cone, with $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s, for $l_f = \{2; 2.5; 3; 4; 5; 7.5\}$ mm. Comparison of the computation time and number of fluid finite elements associated with each model. 64 CPUs were used for all computations.

371 To further assess the convergence of the hydrodynamic force as a function of the
372 size of the fluid elements, the Grid Convergence method (based on the Richardson
373 extrapolation method [38]) has been applied to the present results. This method allows
374 estimating (i) the continuum value (at zero grid spacing) of a quantity of interest from
375 a series of lower-order discrete values and (ii) the error associated with the size of the
376 grid used to discretize the simulation domain. The reader is referred to [39] for details
377 and guidelines about the method. As explained in [40], this kind of error estimation
378 method is sensitive to noise. A polynomial fit has been used on the numerical results
379 to compensate for this limitation around the positive hydrodynamic force peak, where
380 the oscillations are the strongest. The data used for this convergence study and the
381 associated results are synthesized in Table 6. The maximum force values in Table 6 are
382 obtained with the data fit.

	$\Phi = F_z \text{ max}$	$\Phi = F_z \text{ min}$
l_f (mm)	2, 2.5, 3	2, 2.5, 3
Φ_1	58.75 N	-88.69 N
Φ_2	60.57 N	-91.27 N
Φ_3	62.41 N	-94.36 N
p	1.05	1.91
Φ_{21}^{extr}	51.85 N	-83.83 N
e_1^a	13.30%	5.8%
e_2^a	16.8%	8.88%
e_3^a	20.36%	12.57%
GCI_{21}	14.68%	6.85%

Table 6: Data used to perform the convergence study based on the Grid Convergence method and associated results.

383 The application of the Grid Convergence method yields an order of convergence
384 of $p = 1.05$ for $F_z \text{ max}$ and $p = 1.91$ for $F_z \text{ min}$. The estimated continuum values of
385 the maximum and minimum hydrodynamic forces, together with the results obtained
386 with different grid spacing, are presented in Fig. 13a and Fig. 13b, respectively. The
387 maximum hydrodynamic force is extrapolated to be $F_{z \text{ max } 21}^{extr} = 51.85$ N. The lower-
388 scripts refer to the grid number (1 for $l_f = 2$ mm, 2 for $l_f = 2.5$ mm and 3 for $l_f = 3$
389 mm). The numerical uncertainty associated with the model $l_f = 2$ mm, regarding
390 the extrapolated value, is $GCI_{21} = 14.68\%$ (Grid Convergence Index). The minimum

391 hydrodynamic force is extrapolated to be $F_{z\ min\ 21}^{extr} = -83.83\ \text{N}$, , with a GCI_{21} of
 392 6.85%. The extrapolated relative error is the error between a value obtained for a given
 393 grid and the extrapolated continuous solution. This error is given by $e_n^a = \left| \frac{\Phi_{21}^{extr} - \Phi_n}{\Phi_{21}^{extr}} \right|$,
 394 with n the grid number. The extrapolated relative error obtained for the different grid
 395 spacing is presented in Fig. 14. Note that for a normalized grid spacing $r \leq 1.25$, *i.e.*
 396 for a fluid element size $l_f \leq 2.5\ \text{mm}$, the extrapolated relative error remains under
 397 16.8% for $F_{z\ max}$ and under 8.88% for $F_{z\ min}$.

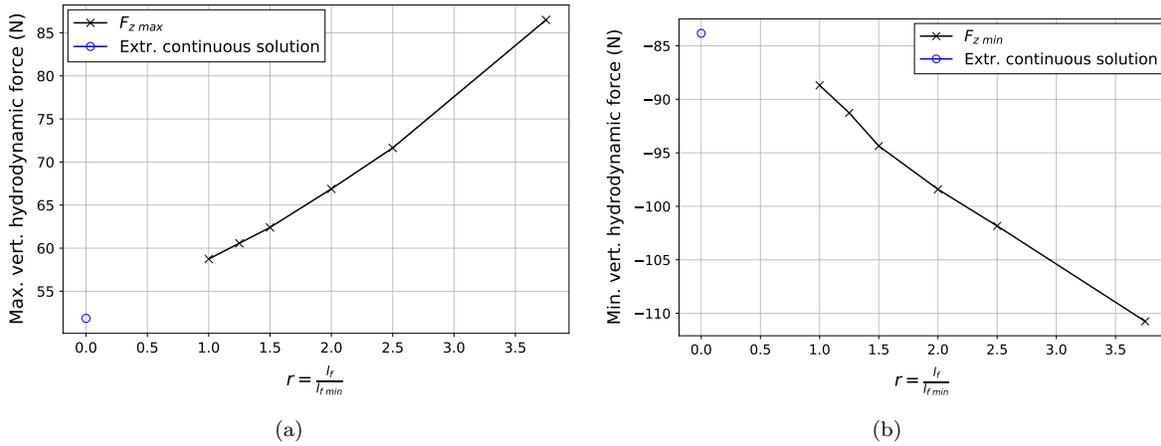


Figure 13: Extrapolation of the (a) maximum and (b) minimum hydrodynamic force using the Richardson extrapolation method.

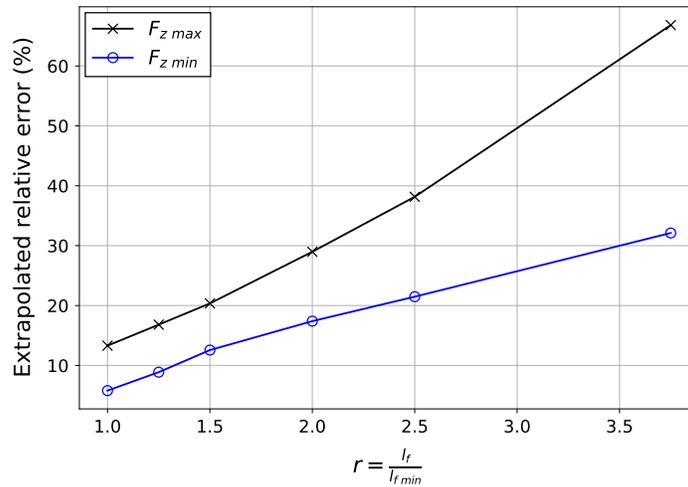


Figure 14: Estimation of the extrapolated relative error for the maximum and minimum hydrodynamic force associated for different normalized grid spacing.

398 In conclusion, the water entry and subsequent exit problem has been modelled us-
 399 ing different fluid element sizes. The convergence of the numerical results has been

400 estimated visually and with an error indicator based on an extrapolation method. It
401 has been observed that for the considered sizes, (i) smaller fluid elements lead to more
402 accurate numerical results and higher computation times, and (ii) the numerical results
403 are not entirely converged. However, to maintain a reasonable computation time (with
404 the model containing several millions of 3D fluid elements and the 64 CPUs used for the
405 computation), it has been decided to set $l_f = 2.5$ mm for the investigations presented
406 in the following sections. The loss of accuracy in terms of hydrodynamic force peak
407 predictions, in comparison with the case $l_f = 2$ mm and with the extrapolated con-
408 tinuous solution, has been considered reasonable compared to the gain in computation
409 time.

410 4. Comparisons with experimental results

411 In this section, the numerical results are compared to experimental data from [31]
412 for the water entry and subsequent exit of a cone (Section 4.1) and a hemisphere
413 (Section 4.2). The evolution of the non-dimensional hydrodynamic forces and the
414 wetted surface radius are considered for several maximum impact velocities ($U_{max} \in$
415 $[0.6; 0.4]$ m/s) and maximum wetted surface radius ($c_{max} \in [200; 250]$ mm). The evolu-
416 tion of the pressure coefficient at the different gauges locations (see Fig. 9) is given for
417 the cone case, with $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s, and for the hemisphere case,
418 with $c_{max} = 200$ mm and $U_{max} = 0.4$ m/s.

419 The conclusions drawn from Section 3 are considered to define the numerical models,
420 namely a water sound celerity $c_s = 500$ m/s and a mesh size $l_f = 2.5$ mm have been
421 adopted. For the cone, the contact stiffness is set to $k_c = 0.1536$ N/mm. For the
422 hemisphere, the contact stiffness is set to $k_c = 1.536$ N/mm.

423 4.1. Water entry and subsequent exit of a cone

424 In Fig. 15, the evolution of the hydrodynamic force is presented together with the
425 position of the lowest point of the cone (h) and the cone velocity (\dot{h}) for the water entry
426 and subsequent exit of a cone for $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s.

427 After the first contact between the cone (or influence zone) and the water, the
428 hydrodynamic force rapidly increases and reaches a maximum (positive force peak).
429 Then, the force decreases and becomes negative (suction force) as the cone decelerates
430 (see Fig. 15). As outlined in [31], when the force is null the cone velocity is still directed
431 downward (along $-\vec{z}$). The force reaches its minimum at the transition between the
432 entry and exit stages, *i.e.* when the cone velocity changes sign. Finally, the force
433 gradually tends to zero.

434 The evolution of the non-dimensional hydrodynamic force during the water entry
435 and subsequent exit of a cone is presented in Fig. 16a and 16b for $c_{max} = 250$ mm
436 and $c_{max} = 200$ mm, respectively. The non-dimensional force is expressed depending
437 on the non-dimensional time $t^* = \frac{U_{max}t}{R}$. Overall, the numerical model predicts with a
438 rather good accuracy the evolution of the hydrodynamic force, for two different initial
439 velocities and maximum penetration depths. In particular, the times at which the
440 force reaches its maximum, its minimum and changes sign are well reproduced by the
441 simulations. Notice that the stage during which the force is negative is longer than the
442 one during which it is positive. However, the numerical model slightly underestimates
443 the magnitude of the force peaks. These results may be related to the observations
444 made in Section 3.3. Larger force peak magnitudes will likely be achieved with a finer
445 mesh. Indeed, a finer mesh implies a smaller contact height (see Eq. (16)) and, thus, a
446 cone kinematics closer to the experiments (*i.e.* a greater maximum penetration depth
447 in the numerical model). The difference between experiments and simulations is more
448 pronounced for the case $c_{max} = 200$ mm and $U_{max} = 0.57$ m/s, with $\Delta f_{adim\ max} = 11\%$
449 and $\Delta f_{adim\ min} = 5\%$ (see Fig. 16b). It is less perceptible for the lower impact velocity
450 $U_{max} = 0.4$ m/s ($\Delta f_{adim\ max} = 3\%$ and $\Delta f_{adim\ min} = 9\%$) because of the oscillations of
451 the experimental data.

452 During the entry stage, the evolution of the hydrodynamic force is independent of
453 the velocity of the structure. The results diverge starting from the transition between
454 the entry and exit stages, around $t^* = 0.25$ and $t^* = 0.18$ in Fig. 16a and Fig. 16b

455 respectively. This divergence highlights the effect of the structure velocity, hence of
 456 gravity, on the evolution of the hydrodynamic force during the exit stage, as commented
 457 in [15]. For both experimental and numerical results, the effect of gravity is more
 458 pronounced for the lower values of U_{max} .

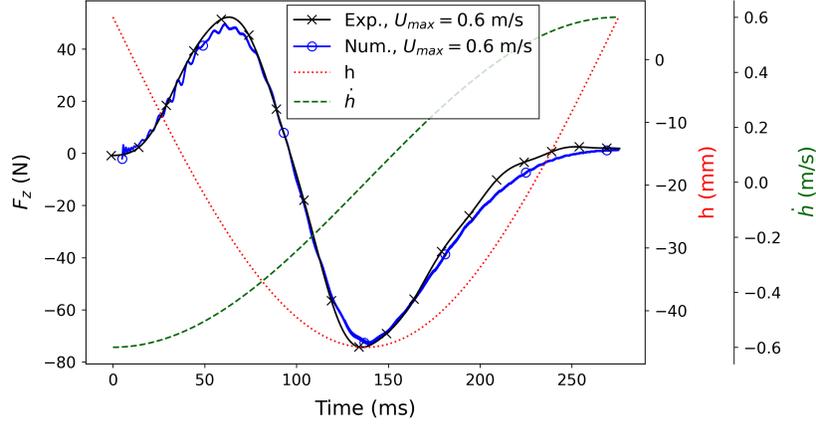


Figure 15: Time history of the hydrodynamic force, position of the lowest point of the cone (h) and the cone vertical velocity (\dot{h}) during the water entry and subsequent exit of a cone with $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s.

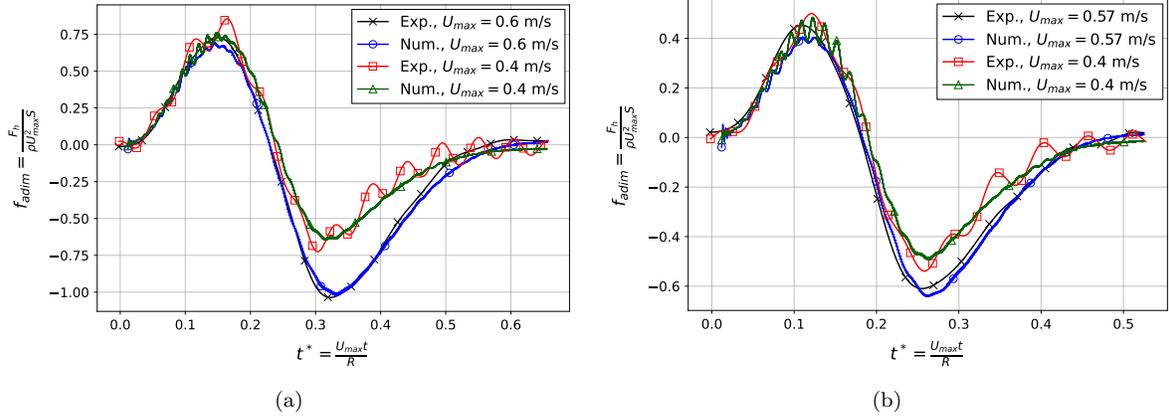


Figure 16: Evolution of the non-dimensional hydrodynamic force depending on the non-dimensional time during the water entry and subsequent exit of the cone, with (a) $c_{max} = 250$ mm and $U_{max} = \{0.4; 0.6\}$ m/s and (b) $c_{max} = 200$ mm and $U_{max} = \{0.4; 0.57\}$ m/s.

459 The evolution of the pressure coefficient measured at the gauges $p1-p5$ during the
 460 water entry-exit of the cone ($c_{max} = 250$ mm and $U_{max} = 0.6$ m/s) is presented in Fig 17.
 461 This configuration features the maximum suction force amplitude of the configurations
 462 considered in this study. Positive pressure peaks are observed for the gauges $p1$ to $p4$
 463 at the beginning of the entry stage ($t^* \leq 0.24$). The amplitude of the positive pressure

464 peaks decreases as the cone decelerates: $p_{adim\ max\ p2} > p_{adim\ max\ p3} > p_{adim\ max\ p4}$. The
465 relative pressure changes sign as the cone decelerates, around $t^* \simeq 0.24$, similarly to
466 the instant when the hydrodynamic force changes sign (see Fig. 16a). The minimum
467 relative pressures are measured around $t^* \simeq 0.33$. The minimum relative pressure are
468 comprised between $p_{adim} = -0.63$ for $p5$ and $p_{adim} = -1.47$ for $p3$. Finally, the pressure
469 returns toward $p_{adim} = 0$ at the end of the exit stage. One can note that the gauge $p5$
470 does not measure a positive pressure peak during the simulation. For this configuration,
471 the occurrence of the minimum relative pressures corresponds to the occurrence of the
472 minimum hydrodynamic force observed in Fig. 16a. It is however impossible to conclude
473 about the accuracy of these pressure evolutions, as no corresponding experimental data
474 is available.

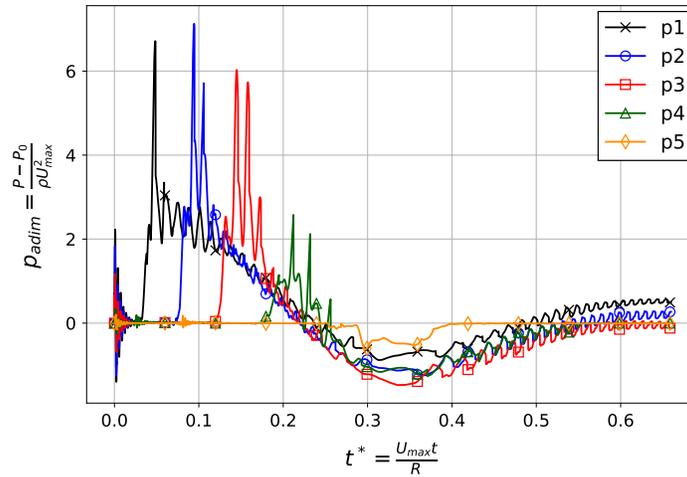


Figure 17: Evolution of the pressure coefficient at the gauges p1-p5 during the water entry and subsequent exit of the cone, with $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s.

475 The evolution of the wetted surface radius for the water entry and subsequent exit
476 of the cone is presented in Fig. 18a and 18b, for $c_{max} = 250$ mm and $c_{max} = 200$ mm,
477 respectively. The numerical results stop before the experimental ones because the
478 duration of the numerical simulations is inferior to the duration of the experiments,
479 but note that the hydrodynamic force is already almost equal to zero at this stage
480 (compare Fig. 16 and 18). For the considered body geometries and kinematics, the
481 maximum wetted surface radius is attained when the penetration depth of the structure

482 is maximum, i.e. when the cone velocity changes sign.

483 Overall, the numerical model accurately predicts the evolution of the wetted surface
 484 radius. However, it slightly underestimates the maximum value of the wetted surface
 485 radius. This is probably due to the absence of the water jets generated during the
 486 impact in the simulation. Indeed, the size of the fluid elements near the structure
 487 remains too large to model these thin jets. The numerical model also predicts a milder
 488 decrease of the wetted surface radius towards the end of the simulation.

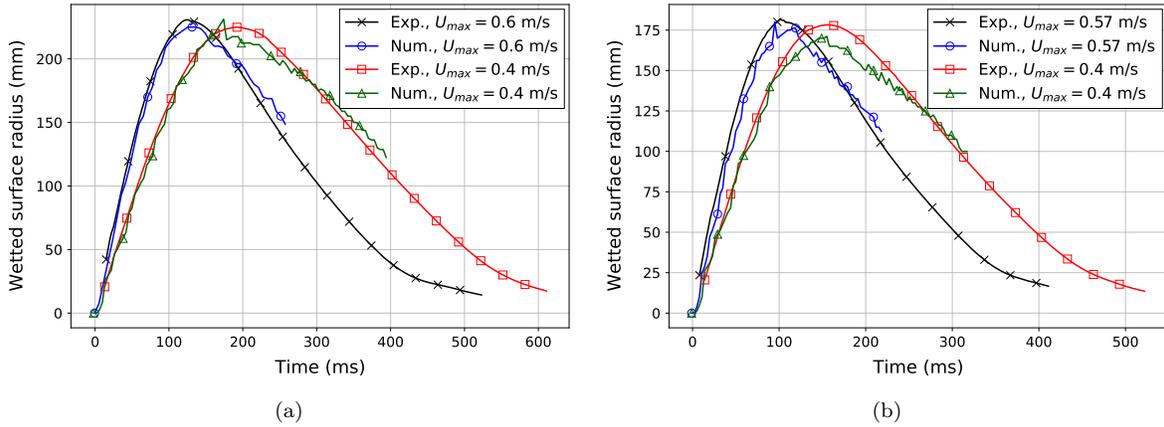


Figure 18: Time history of the wetted surface radius during the water entry and subsequent exit of the cone, with (a) $c_{max} = 250$ mm and $U_{max} = \{0.4; 0.6\}$ m/s and (b) $c_{max} = 200$ mm and $U_{max} = \{0.4; 0.57\}$ m/s.

489 4.2. Water entry and subsequent exit of a hemisphere

490 The evolution of the non-dimensional hydrodynamic force during the water entry
 491 and subsequent exit of a hemisphere is presented in Fig. 19a and 19b, for $c_{max} = 250$ mm
 492 and $c_{max} = 200$ mm, respectively. The non-dimensional force is expressed depending
 493 on the non-dimensional time $t^* = \frac{U_{max}t}{R}$.

494 Contrary to the cone case, the non-dimensional hydrodynamic force is higher during
 495 the entry stage than during the exit stage. Otherwise, the observations are similar
 496 to those made for the cone. The numerical model accurately predicts the evolution
 497 of the hydrodynamic force. The case $c_{max} = 200$ mm and $U_{max} = 0.4$ m/s aside,
 498 $\Delta f_{adim\ max/min} \leq 5\%$. For the case $c_{max} = 200$ mm and $U_{max} = 0.4$ m/s, the difference
 499 between the experimental and numerical minimum force amplitudes is particularly high

500 due to the oscillations of the experimental measures ($\Delta f_{adim\ min} \simeq 19\%$, see Fig. 19b).
501 The effect of gravity on the evolution of the hydrodynamic force is also similar for the
502 hemisphere. During the entry stage, the hydrodynamic force evolution is independent
503 of the velocity of the structure. The results start diverging at the transition between
504 the entry and exit stages, around $t^* = 0.15$ in Fig. 19b. Finally, the effect of gravity is
505 more pronounced for the lower values of U_{max} .

506 In addition, a couple of observations can be made for the case $c_{max} = 250$ mm
507 and $U_{max} = 0.6$ m/s. Firstly, a temporal discrepancy occurs at the end of the en-
508 try stage: the negative non-dimensional force peak occurs later in the simulation (see
509 Fig. 19a). This temporal discrepancy is also visible for the case $c_{max} = 200$ mm and
510 $U_{max} = 0.56$ m/s (see Fig. 19b). Secondly, the force measured experimentally at the
511 end of the exit stage becomes positive before decreasing toward zero. Note that this
512 phenomenon has also been observed numerically for the water entry and exit of an
513 expanding and contracting circular cylinder (see Section 4.3., Fig. 8.c and 8.d, in refer-
514 ence [9]). The present numerical model predicts a different tendency: a gradual increase
515 of the hydrodynamic force toward 0 N at the end of the exit stage. The reasons for this
516 difference are unknown.

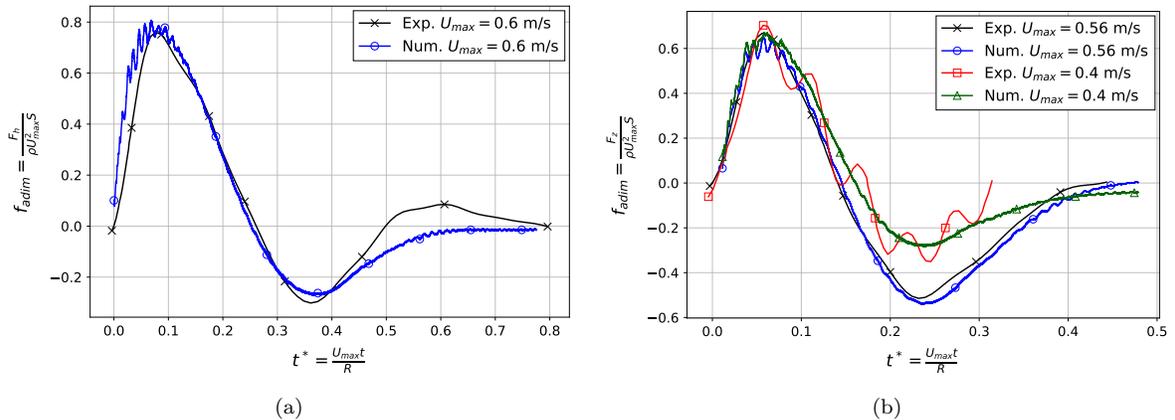


Figure 19: Evolution of the non-dimensional hydrodynamic force depending on the non-dimensional time during the water entry and subsequent exit of the hemisphere, with (a) $c_{max} = 250$ mm and $U_{max} = 0.6$ m/s and (b) $c_{max} = 250$ mm and $U_{max} = \{0.4; 0.56\}$ m/s.

517 The evolution of the pressure coefficient, measured at the gauges $p1$ to $p5$ during the
518 water entry-exit of the hemisphere ($c_{max} = 200$ mm and $U_{max} = 0.4$ m/s) is presented

519 in Fig 20. This configuration features the minimum suction force amplitude of the
520 configurations considered in this study. Positive pressure peaks are observed for the
521 gauges $p1$ to $p3$ at the beginning of the entry stage ($t^* \leq 0.2$). The amplitude of
522 the positive pressure peaks decreases as the hemisphere decelerates: $p_{adim\ max\ p2} >$
523 $p_{adim\ max\ p3}$. A slightly negative relative pressure is observed for the gauges $p2$ and $p3$
524 starting from $t^* \simeq 0.22$. After the positive pressure peak, aside from the gauges $p2$ and
525 $p3$, the relative pressure decreases toward $p_{adim} = 0$ at the end of the exit stage and
526 remains positive. These results seem consistent with the low value of the suction force
527 observed in Fig. 20. Again, it is impossible to conclude about the accuracy of these
528 pressure evolutions, as no corresponding experimental data is available.

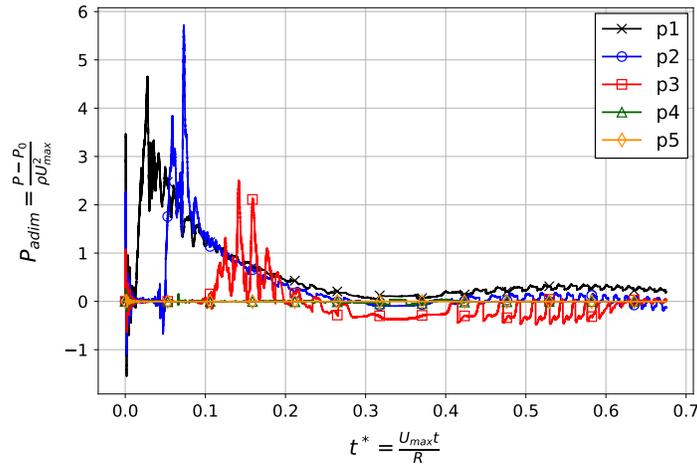


Figure 20: Evolution of the pressure coefficient at the gauges p1-p5 during the water entry and subsequent exit of the hemisphere, with $c_{max} = 200$ mm and $U_{max} = 0.4$ m/s.

529 The evolution of the wetted surface radius for the water entry and subsequent exit of
530 the hemisphere had not been measured experimentally, as explained in [31]. Therefore,
531 the corresponding numerical results are not presented.

532 5. Discussion and conclusion

533 In the present work, simulations of the water entry and subsequent exit of a cone
534 and hemisphere have been analysed. Particular attention has been dedicated to the
535 suction force prediction and the wetted surface evolution. As explained in the intro-
536 duction, suction forces play an important role during aircraft ditching. The objective

537 of the present study has been to assess the capacity of the presented numerical method
538 to model suction forces. The computations have been carried out using a CEL ap-
539 proach and the explicit solver *Radioss*, developed by Altair. This numerical method
540 is adapted to address both simple and more complex three-dimensional hydrodynamic
541 impact problems. Comparisons have been made with existing experimental results.
542 Overall, the numerical results are in good agreement with the experimental results. In
543 particular, the numerical model predicts quite well the evolution of the hydrodynamic
544 force and the transition from positive to negative (suction) force observed in the ex-
545 periments (ranging from 60 N to -75 N). The fluid-structure interaction method is not
546 able to model the thin jet generated during the impact, as observed in the experiments.
547 Indeed, the size of the fluid elements near the structure is too large compared to the
548 thickness of the jets. This may explain the slight underestimation of the wetted surface
549 radius by the numerical model. Nonetheless, the evolution of the wetted surface is
550 overall satisfactorily predicted. The sensitivity of the numerical results to different nu-
551 merical parameters has been studied, especially to the contact stiffness and the size of
552 the fluid elements in the impact zone (near the structure). [The uncertainty associated](#)
553 [with the results obtained for different sizes of fluid elements has been estimated using an](#)
554 [extrapolation method. It appeared that the convergence of the numerical results could](#)
555 [be attained with a finer spatial discretization of the fluid domain. However, a finer](#)
556 [spatial discretization would drastically increase the computational cost as the model is](#)
557 [3D and features several millions of fluid elements. Therefore, an intermediary size of](#)
558 [fluid elements has been selected as a compromise between precision and computational](#)
559 [cost.](#)

560 The evolution of the local pressure has been presented for two of the studied config-
561 urations: the cone case, with $c_{max} = 250$ mm and $U_{max} = 0.6$ mm; and the hemisphere
562 case, with $c_{max} = 200$ mm and $U_{max} = 0.4$ mm. The cone and hemisphere cases
563 respectively feature the maximum and minimum suction force amplitudes of the con-
564 figurations considered in this study. These results show that the present method is able

565 to provide insight into local pressure variations (near the structure) during water entry
566 and subsequent exit problems. A negative relative pressure is observed for the cone
567 case, and almost no negative relative pressure is observed for the hemisphere. Unfor-
568 tunately, the experimental results do not include pressure measurements. Therefore,
569 it is not possible to conclude about the accuracy of these pressure evolutions. How-
570 ever, as hydrodynamic impacts feature high spatial and temporal pressure gradients,
571 it is supposed that a finer spatial discretization of the fluid domain would improve the
572 precision of the computed pressure. The use of smaller fluid elements would obviously
573 induce higher computation times.

574 This numerical method, validated for water entry and subsequent exit experiments
575 led in a laboratory, could be applied to water impact simulations in realistic ditching
576 conditions. Notice that the vertical velocity order of magnitude in the present cases is
577 similar to the vertical velocity order of magnitude of an aircraft during ditching (less
578 than 5 m/s). For instance, in [17], the ditching of a generic rigid aircraft body has been
579 modelled with a structural vertical velocity of 1.5 m/s. Moreover, the same numerical
580 method has been proved efficient to model high-velocity hydrodynamic impacts [35].

581 Future work will be dedicated to performing simulations and new experiments for
582 more complex hydrodynamic impacts, more representative of realistic ditching condi-
583 tions, involving a horizontal velocity and a wavy free surface. Also, the cavitation phe-
584 nomenon could be considered in the present numerical method. Indeed, it is possible to
585 take into account a cut-off pressure corresponding to the physical water vapour pressure
586 (at a given temperature) in the water equation of state. Cavitation acts like a natural
587 limit for the suction forces as it limits the magnitude of negative (relative) pressure
588 in the water. Therefore, considering cavitation would be interesting when modelling
589 hydrodynamic impacts with higher impact velocities, such as aircraft ditching [41].

590 **Acknowledgement**

591 The authors would like to thank ONERA (French aerospace research centre) and
592 IFREMER (*Institut Français de Recherche pour l'Exploitation de la Mer*) for co-funding

593 this project. The authors would like to thank Mathis LOVERINI (Altair) and Thierry
594 SCHWOERTZIG (Altair) for the discussions that improved the quality of the results
595 presented in this paper.

596 Declaration of interests

597 The authors report no conflict of interest.

598 References

- 599 [1] S. Abrate, Hull Slamming, *Applied Mechanics Reviews* 64 (2011) 1003. doi:
600 10.1115/1.4023571.
- 601 [2] C. M. Seddon, M. Moatamedi, Review of water entry with applications to
602 aerospace structures, *International Journal of Impact Engineering* 32 (7) (2006)
603 1045–1067. doi:10.1016/j.ijimpeng.2004.09.002.
604 URL [https://www.sciencedirect.com/science/article/pii/
605 S0734743X04001976](https://www.sciencedirect.com/science/article/pii/S0734743X04001976)
- 606 [3] T. von Karman, The Impact on Seaplane Floats During Landing, Report 321,
607 NACA, Aerodynamical Institute of the Technical High School, Aachen, number:
608 NACA-TN-321 (Oct. 1929).
609 URL <https://digital.library.unt.edu/ark:/67531/metadc54062/>
- 610 [4] H. Wagner, Phenomena Associated with Impacts and Sliding on Liquid Surfaces,
611 Tech. rep., Translation of Über Stoß- und Gleitvorgänge an der Oberfläche von
612 Flüssigkeiten. *Z. Angew. Math. Mech.* 12, 193–215 (in German) (1932).
613 URL http://archive.org/details/nasa_techdoc_20010003513
- 614 [5] H. Ribet, P. Laborde, M. Mahé, Numerical modeling of the impact on water of a
615 flexible structure by explicit finite element method — Comparisons with Radioss
616 numerical results and experiments, *Aerospace Science and Technology* 3 (2) (1999)
617 83–91. doi:10.1016/S1270-9638(99)80032-7.

- 618 URL [https://www.sciencedirect.com/science/article/pii/](https://www.sciencedirect.com/science/article/pii/S1270963899800327)
619 S1270963899800327
- 620 [6] V. Faucher, M. Bulik, P. Galon, Updated VOFIRE algorithm for fast
621 fluid–structure transient dynamics with multi-component stiffened gas flows
622 implementing anti-dissipation on unstructured grids, *Journal of Fluids and*
623 *Structures* 74 (2017) 64–89. doi:10.1016/j.jfluidstructs.2017.07.001.
- 624 URL [https://www.sciencedirect.com/science/article/pii/](https://www.sciencedirect.com/science/article/pii/S0889974616304960)
625 S0889974616304960
- 626 [7] E. E. McBride, L. J. Fisher, Experimental investigation of the effect of rear-fuselage
627 shape on ditching behavior, Report 2929, NACA, Langley Aeronautical Labora-
628 tory, Langley Field, Virginia, USA, number: NACA-TN-2929 (Apr. 1953).
- 629 URL <https://digital.library.unt.edu/ark:/67531/metadc56694/m1/1/>
- 630 [8] D. J. Piro, K. J. Maki, Hydroelastic analysis of bodies that enter
631 and exit water, *Journal of Fluids and Structures* 37 (2013) 134–150.
632 doi:10.1016/j.jfluidstructs.2012.09.006.
- 633 URL [http://www.sciencedirect.com/science/article/pii/](http://www.sciencedirect.com/science/article/pii/S0889974612001806)
634 S0889974612001806
- 635 [9] A. Tassin, D. J. Piro, A. A. Korobkin, K. J. Maki, M. J. Cooker, Two-dimensional
636 water entry and exit of a body whose shape varies in time, *Journal of Fluids and*
637 *Structures* 40 (2013) 317–336. doi:10.1016/j.jfluidstructs.2013.05.002.
- 638 URL [http://www.sciencedirect.com/science/article/pii/](http://www.sciencedirect.com/science/article/pii/S0889974613001187)
639 S0889974613001187
- 640 [10] K. Hughes, R. Vignjevic, J. Campbell, T. De Vuyst, N. Djordjevic, L. Papagiannis,
641 From aerospace to offshore: Bridging the numerical simulation gaps–Simulation
642 advancements for fluid structure interaction problems, *International Journal of*
643 *Impact Engineering* 61 (2013) 48–63. doi:10.1016/j.ijimpeng.2013.05.001.

644 URL [https://www.sciencedirect.com/science/article/pii/
645 S0734743X13001073](https://www.sciencedirect.com/science/article/pii/S0734743X13001073)

646 [11] C. E. Brennen, Cavitation and bubble dynamics, no. 44 in Oxford engineering
647 science series, Oxford University Press, New York, 1995.

648 [12] C. Judge, A. Troesch, M. Perlin, Initial water impact of a wedge at vertical and
649 oblique angles, Journal of Engineering Mathematics 48 (3) (2004) 279–303. doi:
650 10.1023/B:engi.0000018187.33001.e1.
651 URL <https://doi.org/10.1023/B:engi.0000018187.33001.e1>

652 [13] D. H. Peregrine, L. Thais, The effect of entrained air in violent wa-
653 ter wave impacts, Journal of Fluid Mechanics 325 (1996) 377–397.
654 doi:10.1017/S0022112096008166.
655 URL [https://www.cambridge.org/core/product/identifier/
656 S0022112096008166/type/journal_article](https://www.cambridge.org/core/product/identifier/S0022112096008166/type/journal_article)

657 [14] G. N. Bullock, A. R. Crawford, P. J. Hewson, M. J. A. Walkden, P. A. D. Bird,
658 The influence of air and scale on wave impact pressures, Coastal Engineering
659 42 (4) (2001) 291–312. doi:10.1016/S0378-3839(00)00065-X.
660 URL [https://www.sciencedirect.com/science/article/pii/
661 S037838390000065X](https://www.sciencedirect.com/science/article/pii/S037838390000065X)

662 [15] A. Del Buono, G. Bernardini, A. Tassin, A. Iafrati, Water entry and exit of 2D
663 and axisymmetric bodies, Journal of Fluids and Structures 103 (2021) 103269.
664 doi:10.1016/j.jfluidstructs.2021.103269.
665 URL [https://www.sciencedirect.com/science/article/pii/
666 S0889974621000529](https://www.sciencedirect.com/science/article/pii/S0889974621000529)

667 [16] N. R. S. Toso, Contribution to the modelling and simulation of aircraft struc-
668 tures impacting on water, Ph.D. thesis, Institute of Aircraft Design, Universität

669 Stuttgart, accepted: 2009-12-18 (2009).

670 URL <http://elib.uni-stuttgart.de/handle/11682/3840>

671 [17] B. Langrand, M. H. Siemann, Full aircraft ditching simulation: a comparative
672 analysis of advanced coupled fluid-structure computational methods, International
673 Conference on Impact Loading of Structures and Materials (2018) 4.

674 [18] J. Donea, A. Huerta, J.-P. Ponthot, A. Rodríguez-Ferran, Encyclopedia of com-
675 putational mechanics - Chapter 14, John Wiley, Chichester, West Sussex, 2004,
676 oCLC: ocm55800932.

677 [19] L. B. Lucy, A numerical approach to the testing of the fission hypothesis, The
678 Astronomical Journal 82 (1977) 1013–1024. doi:10.1086/112164.

679 URL <http://adsabs.harvard.edu/abs/1977AJ.....82.1013L>

680 [20] J. J. Monaghan, Simulating Free Surface Flows with SPH, Journal of Computa-
681 tional Physics 110 (2) (1994) 399–406. doi:10.1006/jcph.1994.1034.

682 URL [https://www.sciencedirect.com/science/article/pii/](https://www.sciencedirect.com/science/article/pii/S0021999184710345)
683 [S0021999184710345](https://www.sciencedirect.com/science/article/pii/S0021999184710345)

684 [21] M. Souli, J.-F. Sigrist, Interaction fluide-structure: Modélisation et simulation
685 numérique, Vol. 19, Hermès - Lavoisier, 2010.

686 URL [https://www.tandfonline.com/doi/full/10.1080/17797179.2010.](https://www.tandfonline.com/doi/full/10.1080/17797179.2010.9737468)
687 [9737468](https://www.tandfonline.com/doi/full/10.1080/17797179.2010.9737468)

688 [22] F. Casadei, N. Leconte, M. Larcher, Strong and weak forms of a fully non-
689 conforming FSI algorithm in fast transient dynamics for blast loading of
690 structures, in: EU Science Hub - European Commission, Corfu, Greece, 2011,
691 p. 20.

692 URL [https://ec.europa.eu/jrc/en/publication/](https://ec.europa.eu/jrc/en/publication/contributions-conferences/strong-and-weak-forms-fully-non-conforming-fsi-algorith)
693 [contributions-conferences/strong-and-weak-forms-fully-non-conforming-fsi-algorith](https://ec.europa.eu/jrc/en/publication/contributions-conferences/strong-and-weak-forms-fully-non-conforming-fsi-algorith)

- 694 [23] I. Stenius, A. Rosén, J. Kutteneuler, Explicit FE-modelling of fluid–structure
695 interaction in hull–water impacts, *International Shipbuilding Progress* 53 (2)
696 (2006) 103–121, publisher: IOS Press.
- 697 URL [https://content.iospress.com/articles/
698 international-shipbuilding-progress/isp006](https://content.iospress.com/articles/international-shipbuilding-progress/isp006)
- 699 [24] I. Stenius, A. Rosén, J. Kutteneuler, Explicit FE-modelling of hydroelasticity in
700 panel-water impacts, *International Shipbuilding Progress* 54 (2-3) (2007) 111–127,
701 publisher: IOS Press.
- 702 URL [https://content.iospress.com/articles/
703 international-shipbuilding-progress/isp022](https://content.iospress.com/articles/international-shipbuilding-progress/isp022)
- 704 [25] I. Stenius, A. Rosén, J. Kutteneuler, Hydroelastic interaction in panel-
705 water impacts of high-speed craft, *Ocean Engineering* 38 (2) (2011) 371–381.
706 doi:10.1016/j.oceaneng.2010.11.010.
- 707 URL [https://www.sciencedirect.com/science/article/pii/
708 S0029801810002556](https://www.sciencedirect.com/science/article/pii/S0029801810002556)
- 709 [26] M. H. Siemann, B. Langrand, Coupled fluid-structure computational
710 methods for aircraft ditching simulations: Comparison of ALE-FE
711 and SPH-FE approaches, *Computers & Structures* 188 (2017) 95–108.
712 doi:10.1016/j.compstruc.2017.04.004.
- 713 URL [http://www.sciencedirect.com/science/article/pii/
714 S0045794916312998](http://www.sciencedirect.com/science/article/pii/S0045794916312998)
- 715 [27] T. Xiao, N. Qin, Z. Lu, X. Sun, M. Tong, Z. Wang, Development of a
716 smoothed particle hydrodynamics method and its application to aircraft
717 ditching simulations, *Aerospace Science and Technology* 66 (2017) 28–43.
718 doi:10.1016/j.ast.2017.02.022.
- 719 URL [https://www.sciencedirect.com/science/article/pii/
720 S1270963817303413](https://www.sciencedirect.com/science/article/pii/S1270963817303413)

721 [28] C. Chen, A.-M. Zhang, J.-Q. Chen, Y.-M. Shen, SPH simulations of water entry
722 problems using an improved boundary treatment, *Ocean Engineering* 238 (2021)
723 109679. doi:10.1016/j.oceaneng.2021.109679.

724 URL [https://www.sciencedirect.com/science/article/pii/
725 S0029801821010532](https://www.sciencedirect.com/science/article/pii/S0029801821010532)

726 [29] M. Übüerrück, all WP2 contributing partners, D2.2: Progress Report, Tech. rep.,
727 Increased Safety & Robust Certification for ditching of Aircrafts & Helicopters
728 (Sep. 2018).

729 [30] TUHH, , all WP2 contributing partners, D2.4: Final Report - Benchmarking,
730 Tech. rep., Increased Safety & Robust Certification for ditching of Aircrafts &
731 Helicopters (Feb. 2020).

732 [31] T. Breton, A. Tassin, N. Jacques, Experimental investigation of the water entry
733 and/or exit of axisymmetric bodies, *Journal of Fluid Mechanics* 901, publisher:
734 Cambridge University Press (Oct. 2020). doi:10.1017/jfm.2020.559.

735 URL [https://www.cambridge.org/core/journals/
736 journal-of-fluid-mechanics/article/experimental-investigation-of-the-water-entry-
737 B9E3CA11EA4C3493F92ABBB9B3B04D01](https://www.cambridge.org/core/journals/journal-of-fluid-mechanics/article/experimental-investigation-of-the-water-entry-B9E3CA11EA4C3493F92ABBB9B3B04D01)

738 [32] J. Gobeltz, P. Gythiel, Essai d'amerrissage forcé sur bassin d'une maquette au
739 1/16 ème de l'avion Mercure, Tech. Rep. 74-05, Institut de Mécanique des FLuides
740 de Lille (IMFL), Lille, France (Feb. 1974).

741 [33] A. Tassin, T. Breton, B. Forest, J. Ohana, S. Chalony, D. Le Roux, A. Tancray,
742 Visualization of the contact line during the water exit of flat plates, *Experiments
743 in Fluids* 58 (8) (2017) 104. doi:10.1007/s00348-017-2383-1.

744 URL <https://doi.org/10.1007/s00348-017-2383-1>

745 [34] Richard Saurel, F. Petitpas, R. A. Berry, Simple and efficient relaxation methods
746 for interfaces separating compressible fluids, cavitating flows and shocks in

- 747 multiphase mixtures, *Journal of Computational Physics* 228 (5) (2009) 1678–1712.
748 doi:10.1016/j.jcp.2008.11.002.
749 URL [https://www.sciencedirect.com/science/article/pii/
750 S0021999108005895](https://www.sciencedirect.com/science/article/pii/S0021999108005895)
- 751 [35] M. Goron, B. Langrand, T. Fourest, N. Jacques, A. Tassin, Assesment of coupled
752 lagrangian eulerian finite element simulations to model suction forces during hy-
753 drodynamic impacts, *International Conference on High Performance and Optimum
754 Structures and Materials Encompassing Shock and Impact Loading* (2022) 12.
- 755 [36] D. Delsart, B. Langrand, A. Vagnot, Evaluation of a Euler/Lagrange coupling
756 method for the ditching simulation of helicopter structures, in: *WIT Transactions
757 on the Built Environment*, Vol. 105, Royal Mare Village, Crete, Greece, 2009.
758 doi:10.2495/fsi090241.
- 759 [37] A. Bayliss, E. Turkel, Outflow Boundary Conditions for Fluid Dynamics, *SIAM
760 Journal on Scientific and Statistical Computing* 3 (Jul. 1982). doi:10.1137/
761 0903016.
- 762 [38] L. F. Richardson, J. A. Gaunt, VIII. The deferred approach to the limit,
763 *Philosophical Transactions of the Royal Society of London. Series A, Containing
764 Papers of a Mathematical or Physical Character* 226 (636-646) (1927) 299–361,
765 publisher: Royal Society. doi:10.1098/rsta.1927.0008.
766 URL [https://royalsocietypublishing.org/doi/abs/10.1098/rsta.1927.
767 0008](https://royalsocietypublishing.org/doi/abs/10.1098/rsta.1927.0008)
- 768 [39] I. B. Celik, U. Ghia, P. J. Roache, C. J. Freitas, H. Coleman, P. E. Raad,
769 Procedure for Estimation and Reporting of Uncertainty Due to Discretization
770 in CFD Applications, *Journal of Fluids Engineering* 130 (7) (Jul. 2008). doi:
771 10.1115/1.2960953.
772 URL <https://doi.org/10.1115/1.2960953>

- 773 [40] L. Eça, M. Hoekstra, A procedure for the estimation of the numerical uncertainty
774 of CFD calculations based on grid refinement studies, Journal of Computational
775 Physics 262 (2014) 104–130. doi:10.1016/j.jcp.2014.01.006.
776 URL [https://www.sciencedirect.com/science/article/pii/
777 S0021999114000278](https://www.sciencedirect.com/science/article/pii/S0021999114000278)
- 778 [41] A. Iafrati, S. Grizzi, Cavitation and ventilation modalities during ditching, Physics
779 of Fluids 31 (5) (2019) 052101, publisher: American Institute of Physics. doi:
780 10.1063/1.5092559.
781 URL <https://aip.scitation.org/doi/full/10.1063/1.5092559>