# FLOW SEPARATION MODEL FOR THE WATER IMPACT PROBLEM

**A. DEL BUONO**[*†], **A. IAFRATI**[*] **AND G. BERNARDINI**[†*]

[*]CNR-INM INstitute of Marine engineering
Via di Vallerano 139, 00128 Rome, Italy

[†] Roma Tre University,
Via della Vasca Navale 79, 00146 Rome, Italy

**Key words:** Water entry, Flow separation

**Abstract.** The present paper proposes a new flow separation model for the water impact problem which is here solved through by a potential flow model with fully non-linear boundary conditions at the free-surface. The unsteady boundary value problem is numerically solved through a hybrid BEM-FEM approach where a boundary element method is coupled to a simplified finite element method for describing the thinnest part of the jet. In the water entry of bodies with smoothly curved surface, the point where the flow detaches from the body contour is unknown and has to be computed as a part of the numerical solution. For this reason, the hybrid BEM-FEM approach is here extended to include a flow separation model based on a kinematic criterion.

The proposed model is applied to the water entry of different smoothly curved body with a constant vertical velocity. Numerical results show the capability of the formulation to accurately describe the free-surface evolution and pressure distribution as well as to provide a consistent prediction of the flow separation.

## 1 Introduction

During the water impact of a body, the free-surface rises along the body contour. However, owing to the geometrical properties of the body contour or to the variation of the entry velocity, the flow, can detach from the body surface [1]. An accurate modelling of the flow separation phenomena is necessary for providing a correct prediction of both free-surface dynamics and hydrodynamic loads. Generally, the flow separation is modelled by applying a Kutta condition at the separation point, implying that the free-surface leaves the body tangentially [2, 3]. For bodies with hard chines, such as a wedge, impacting the water with either constant or increasing impact velocity, the separation point can be located *a priori* at the sharp corner, i.e. where there is a geometry discontinuity. Conversely, if the impacting bodies have smoothly curved contours, such as a circular cylinder, or in the case of bodies undergoing large deceleration, the flow separation point is unknown and has to be derived as a part of the numerical solution.

An approach based on the negative pressure is often adopted in the literature, e.g. [4]. In this approach, the flow is assumed to detach from the body surface when the pressure drops below the atmospheric value at least on a reasonably large portion of the wetted area. However, such a criterion seems too strong as there is evidence that negative pressure can occur without flow separation. For example, this is shown in [5] for a rigid plate. In [6] during the decelerating water entry problem of a wedge and a cone, the pressure becomes negative in a large part of the body contour, but the flow is still attached. Also high speed ditching tests presented in [7] display negative pressures although the flow is still attached.

In this paper, a new flow separation model, based on a kinematic criterion, is presented. The latter is based on the displacement of the separated panels with respect to the body contour. This criterion is expected more general than the approach based on the negative pressure. The aim is to further extend the capabilities of the 2D fully non-linear potential flow model, proposed and validated in [2, 8] for the water entry of a wedge and more recently in [6] for the water entry/exit of a wedge and a cone, to deal with the water impact of bodies with smoothly curved contours. The problem is formulated via a boundary-element representation of the velocity potential (BEM model) and the time evolution is described by a mixed Eulerian-Lagrangian approach, originally proposed in [9]. The thinnest part of the jet, that rises along the body contour, is modeled with control volumes where the velocity potential is represented by a harmonic polynomial expression. In this way, the thin jet is accounted for and an improved prediction of the flow separation is possible with a reduced computational effort.

The investigation in the paper is mainly focused on the description of the flow about a 2D circular cylinder impacting the water surface with a constant entry velocity, which is largely studied in the literature. For example, experimental data of this type are found in [10]. In [11] the problem was studied by using a boundary element method based on Cauchy's theorem, whereas the CIP (Constrained Interpolation Profile) method is adopted in [12], where a multiphase flow problem is studied. The last two papers also investigate the water exit stage of a circular cylinder. To assess the capablity of the separation model to handle different body shapes, the kinematic crierion is also tested in the water entry of a 2D elliptical cylinder with constant vertical velocity. Here, the aim is twofold: (i) to verify the capability of the hybrid BEM-FEM approach in describing the free-surface evolution and the pressure distribution in the water entry problem of smoothly curved surface; (ii) to test the use of the kinematic approach for describing the flow separation phenomena.

## 2 Theoretical formulation

In this section the 2D fully non-linear potential flow model is briefly outlined. The model is based on a hybrid BEM-FEM approach proposed and validated in [2, 8]. Then, the kinematic criterion, used to describe the flow separation phenomena, is introduced and discussed.

### 2.1 Hybrid BEM-FEM approach

The water impact problem is studied under the hypotheses of incompressible fluid and irrotational flow. It is formulated in terms of the velocity potential, $\varphi$, which satisfies the Laplace equation in the fluid domain, the Neumann boundary condition on the body surface, coming from the impermeability of the body, and a Dirichelet boundary condition on the free-surface.

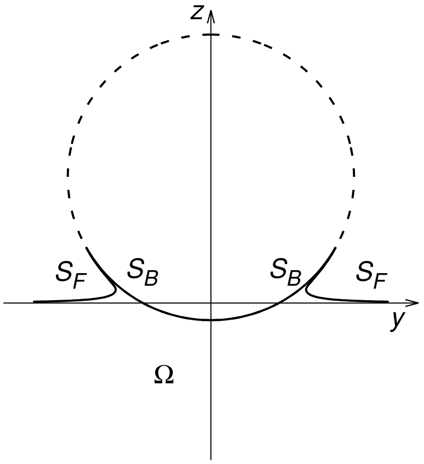


Figure 1: Sketch of the problem

The latter is represented by a dynamic boundary condition imposing constant pressure over the free-surface. Moreover, it is assumed that, according to the kinematic condition, particles lying on the free-surface remain there. So, the governing equations in an earth-fixed frame of reference, $y$ being the horizontal axis coinciding with the still water level and $z$ being the vertical axis oriented upwards, read

$$\begin{aligned} \nabla^2\varphi &= 0 && \Omega \\ \frac{\partial\varphi}{\partial n} &= \mathbf{V}\cdot\mathbf{n} && S_B \\ \frac{D\varphi}{Dt} &= \frac{|\nabla\varphi|^2}{2} && S_F \\ \frac{D\mathbf{x}}{Dt} &= \mathbf{u} && S_F \end{aligned} \tag{1}$$

where $\Omega$ is the fluid domain which is assumed infinitely deep and is bounded by the free-surface, $S_F$, and by the body surface, $S_B$ (see figure 1). $\mathbf{V}$ is the body vertical velocity, $\mathbf{n}$ is the unit vector normal to the boundary of the fluid domain oriented inwards, $\frac{D}{Dt}$ is the total derivative with respect to time $t$, $\mathbf{x}$ denotes the position of a particle lying at the free-surface and $\mathbf{u}$ the corresponding velocity. Although the model allows the inclusion of the gravity effects [6], which may have an influence on the flow separation, they are neglected here. Also the surface-tension effects are neglected. The problem is solved through a mixed Eulerian-Lagrangian approach [9] and a Boundary Integral Representation of the velocity potential is used at each time step. Enforcing it at the boundary of the fluid domain, a boundary integral equation of mixed first and second kind for the velocity potential and its normal derivative on the free-surface is obtained

$$\frac{1}{2}\varphi(P) = \int_{S_B \bigcup S_F} \left( \frac{\partial\varphi(Q)}{\partial n} G(P,Q) - \varphi(Q) \frac{\partial G(P,Q)}{\partial n} \right) dS(Q) \qquad P \in S_B \cup S_F \tag{2}$$

where $G$ is the free-space Green's function of the Laplace operator, which in two dimensions is

$$G(P,Q) = \frac{1}{2\pi} log\left(|P-Q|\right). \tag{3}$$

Equation (2) is numerically solved through a Boundary Element Method, by discretizing the fluid contour with straight line panels where a piecewise constant distribution for the velocity potential and its normal derivative is assumed. The numerical solution of the problem provides the velocity potential on the wetted surface and its normal derivative on the free-surface. The velocity field on the free-surface is completely known as the tangential velocity is obtained through differentiation of the velocity potential along the boundary. The free-surface is followed in a Lagrangian way by integrating in time the kinematic and dynamic boundary conditions, thus obtaining the new free-surface shape and the velocity potential on it. The time integration is performed through a second order Runge-Kutta scheme.

In order to reduce the high computational effort required by a detailed description of the thin jet developing along the body contour and to provide an accurate prediction of the flow separation, a simplified jet model was developed in [2, 8]. The simplified model is based on a hybrid Finite Element-Boundary Element model, which divides the thin jet in control volumes and uses a harmonic polynomial series, up to second order, about the corresponding centroid $(y^*, z^*)$, to represent the velocity potential inside each control volume

$$\varphi_i^j(y,z) = A_i + B_i(y-y^*) + C_i(z-z^*) + \frac{1}{2}D_i\left[(y-y^*)^2 - (z-z^*)^2\right] + E_i(y-y^*)(z-z^*) \tag{4}$$

It is worth nothing that the vertices of each control volume $i$ correspond to the panel centroids on the wetted area and on the free-surface. The coefficients $A_i, .., E_i$ of the equation 4 are unknown and are derived by enforcing the boundary conditions at the body and free-surface sides and by ensuring the continuity of the normal derivative of the velocity potential at adjacent elements [2, 8].

The pressure distribution along the wetted surface is computed by using the unsteady Bernoulli's equation

$$p - p_{inf} = -\rho\left(\frac{\partial\varphi}{\partial t} + \frac{|\nabla\varphi|^2}{2}\right), \tag{5}$$

where $\rho$ is the fluid density and $p_{inf}$ is the reference pressure. In the equation (5), the time derivative of the velocity potential along the body, $\frac{\partial\varphi}{\partial t}$, is unknown and is evaluated by formulating a boundary value problem similar to (1) which is solved numerically by the hybrid BEM-FEM approach [6].

### 2.2 Kinematic criterion

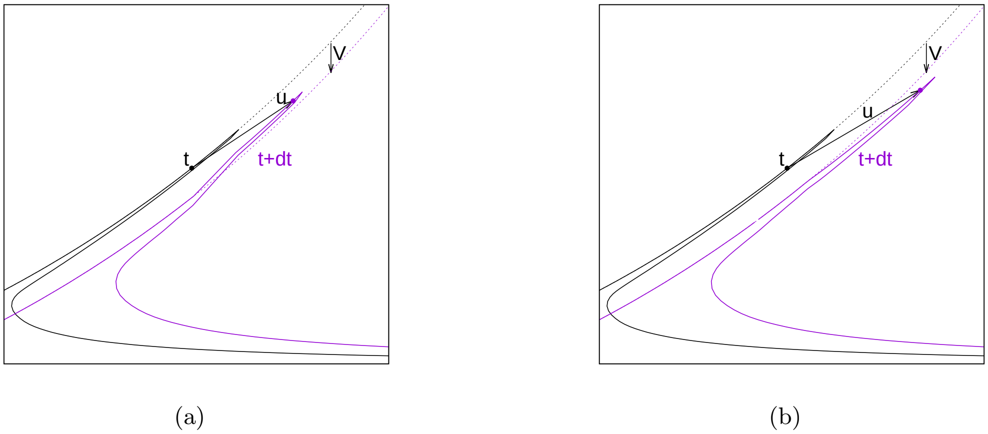
As the flow rises along the body contour, it detaches at the separation point. Contrary to what happens for bodies with hard chines, where the separation point is known *a priori*, in the water impact of a convex body, the separation point has to be determined as a part of the numerical solution. As aforesaid, the existing approach based on the appearance of negative pressure zones does not seem to be always appropriate. In this perspective, a new flow separation model is included in the fully non-linear potential flow solver. The model is based on a kinematic criterion which can be more general than the one based on negative pressure. The adjective kinematic is used because the model is based on the relative vertical motion between the latest panels still attached to the body and the body itself. The approach works as follows. Before the onset of the flow separation, the last few panels lying on the wetted part of the body surface close to the jet tip are considered as *check* panels and an attempt solution is computed by assuming the *check* panels to behave as free-surface panels, i.e. by assigning the dynamic and kinematic boundary conditions. The velocity field is evaluated by solving numerically the boundary value problem (1), as explained above. So, the *check* panels are moved with the flow velocity $\mathbf{u}$. Correspondingly, the impacting body is moved with the vertical velocity $\mathbf{V}$ (see figure 2). At the end of the time step there exist two possibilities:

- The panels are above the body contour (Figure 2a): it means that the fluid particles are still able to follow the body contour and, although developing a slightly negative pressure for centripetal acceleration, the flow remains attached. Hence, the solution is restarted from the beginning of the time step applying the standard body boundary condition on the panels under consideration and the correct free-surface shape is evaluated.
- The panels are below the body contour (Figure 2b): in this case, the inertia of the fluid particles prevent them to follow the body contour and the panels leaves the body surface. Therefore the *check* panels are actually separated and the flow separation has started.

This procedure is repeated as it is until the flow separation occurs. Once the separation starts, the same procedure is applied to the last panel lying on the body contour and attached to the separated portion. The reason why multiple panels are chosen as check panels is for having a larger separated area when the separation starts. This choice provides a better development of the separated area at the beginning of the separation. The kinematic criterion is activated at certain step, expressed in terms of the ratio between the current depth $d$ and the characteristics length of the impacting body, e.g. the radius of the cylinder. Although the instant when the separation starts is arbitrary and could occur earlier, at this stage of the development the model is activated artificially in order to achieve a better control of the separation process and to guarantee that the jet is fully developed when the model is activated.

## 3 Results

The vertical water entry problem of a 2D circular cylinder moving with a constant vertical velocity is here investigated. This case was already studied with a pure Boundary Element Method in [13] where the thinnest part of the jet was truncated. Otherwise, the hybrid BEM-FEM approach is adopted here and the thin jet is retained. Figure 3 shows, for different time instants, expressed in terms of the ratio between the current depth $d$ and the radius of the circular cylinder $R$, the results in terms of free-surface evolution (left figures) and pressure coefficient distribution (right figures). Only the right side of the domain is shown because of the symmetry of the problem. These results refer to the phase of motion in which the flow is still attached to the body. As the jet model is used here, when the flow rises along the body contour a thin jet is formed. The thickness at the jet root increases with the local deadrise angle of the impacting body and the wetted area is significantly larger than the penetration depth. The pressure distribution along the wetted surface is characterized by a peak occurring just behind



(a) (b)

Figure 2: Sketch of the kinematic model procedure. The dashed lines represent the body contour; the continue lines represent the fluid boundary. a) The panels penetrate the body: they are still attached. b) The panels don't penetrates the body: they are separated and the flow separation starts.

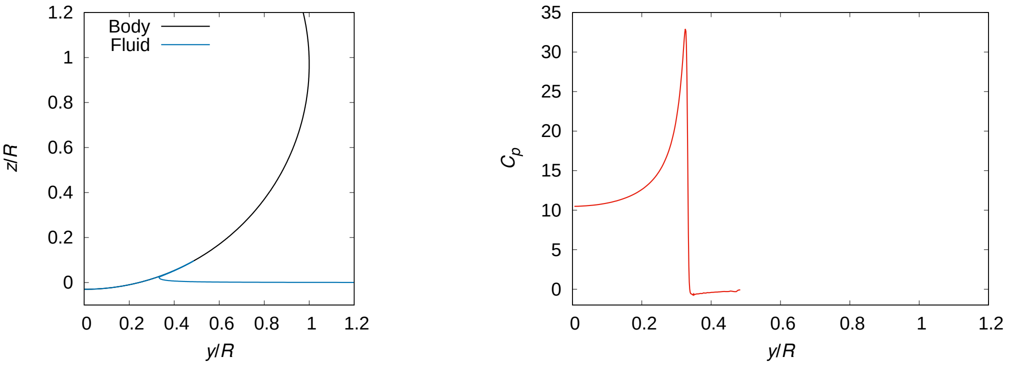
the jet root which progressively diminishes in time as the local deadrise angle increases. The pressure inside the thin jet is essentially negligble even if it takes on small negative values. It is not clear if the negative pressure is physical, due to the body curvature, or is an artifact of the numerical procedure. The possibility that the computed pressure is affected by the numerical model is another motivation for the develpoment of a new model based on a different criterion.

Hence the same simulation is repeated by activating the kinematic model at the time step corresponding to $d/R = 0.075$, where $R$ is the radius of the cylinder. Figure 4 shows the results in terms of free-surface evolution (left figures) and pressure coefficient distribution (right figures), obtained for two different time steps. The jet tip starts to detach from the body contour and the pressure decreases again (see figure 4a). As the body continues to move down, the length of the separated part increases and propagates regardless of the one still attached to the body and the pressure peak progressively disappears (see figure 4b). Figure 5 shows the time history of the slamming coefficient which is defined as

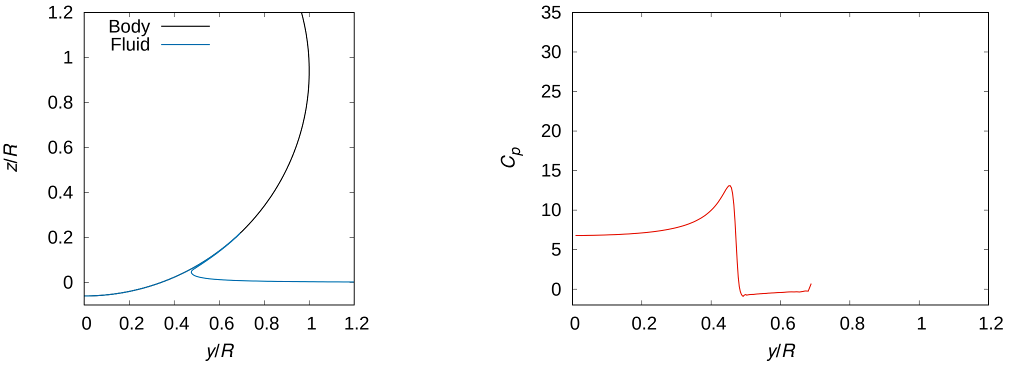
$$C_s = \frac{F}{\frac{1}{2}\rho V^2} \tag{6}$$

where $F$ is the vertical hydrodynamic load and $V$ is the body vertical velocity. The initial step on the curves is because the numerical simulations starts with the body already sligthly submerged. The dark curve is obtained by imposing no-separation, i.e. the flow continues to follow the body curvature, whereas the blue curve is obtained when the separation model is activated. After the separation start (at $d/R = 0.075$) the two curves diverge. Even though a deeper verification and validation is necessary, the result highlights the possible influence of the separation on the total loading.

In order to further test the capabilities of the kinematic criterion, the model is used in the water entry of a 2D elliptical cylinder with constant vertical velocity. The eccentricity of the



(a) $d/R = 0.030$



(b) $d/R = 0.060$

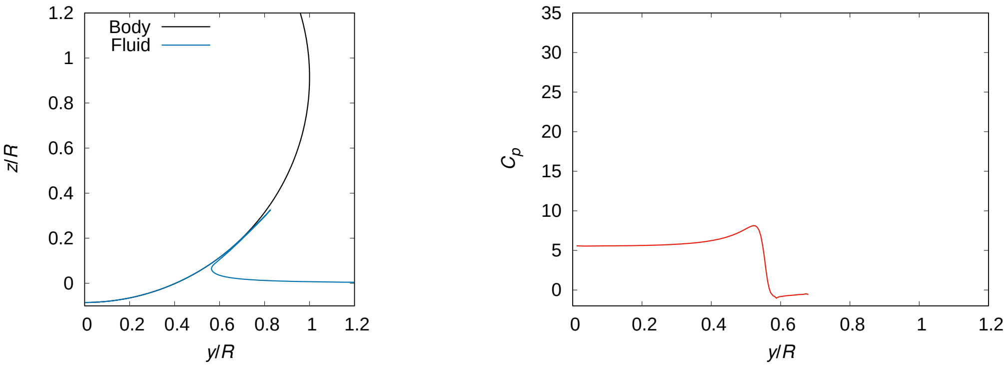
Figure 3: Circular cylinder. Free-surface evolution and pressure coefficient distribution at different time steps before the flow separation.

body is 3/5 and the separation model is activated at the time step corresponding to $d/a = 0.075$, where $a$ is the horizontal semi-axis of the body. The results in terms of free-surface evolution (left figures) and pressure coefficient distribution (right figures) are shown in the figure 6. Also in this case, as soon as the jet tip starts to detach from the body contour, the length of the separated part increases in time and the pressure peak progressively disappears.

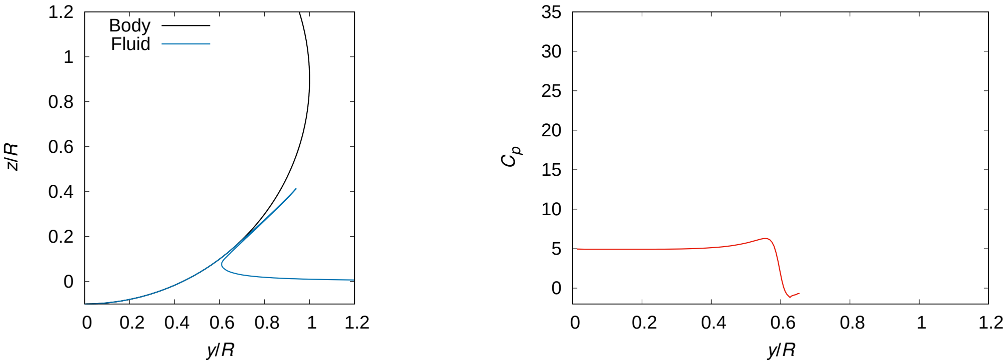
The performed tests have shown that the proposed procedure works well and seems to be consistent with the physics of the problem. However, further investigations are needed to fix some instability issues which born in the transition region between BEM and FEM solution, which causes the simulation stops shortly after the separation start.

## 4 Conclusion

A fully non-linear potential flow model based on the hybrid BEM-FEM approach has been proposed for the description of the flow about the water impact of 2D bodies with smoothly curved surface. In particular, a new flow separation model, based on a kinematic criterion has been proposed, in order to determine the flow separation point as a part of the numerical solution. The water entry with constant vertical velocity of a 2D circular cylinder has been investigated.



(a) $d/R = 0.085$



(b) $d/R = 0.100$

Figure 4: Circular cylinder. Free-surface evolution and pressure coefficient distribution at different time steps after the flow separation.

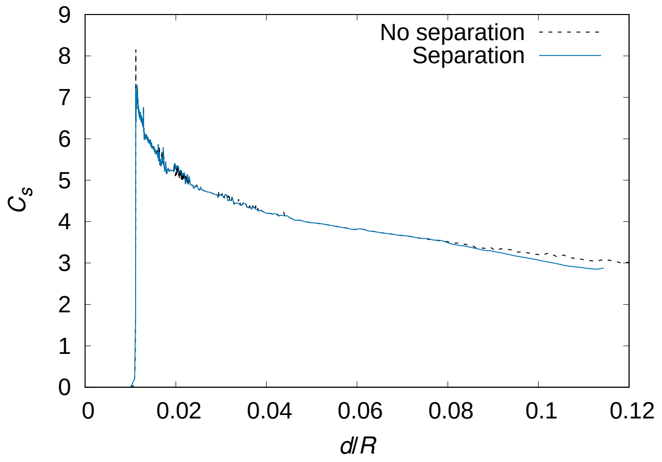
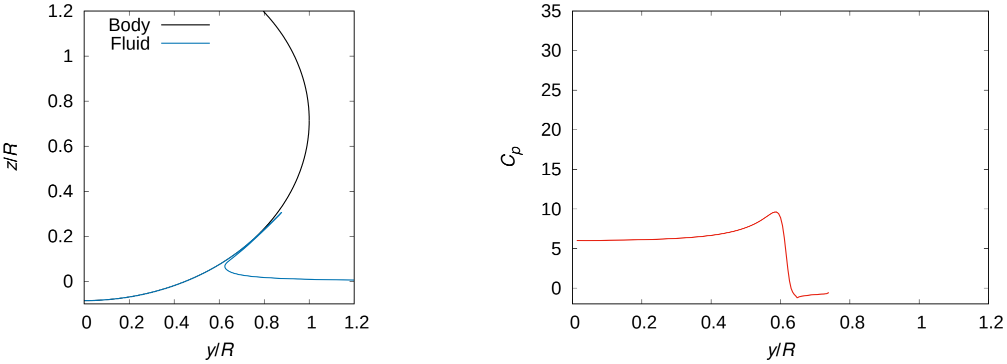
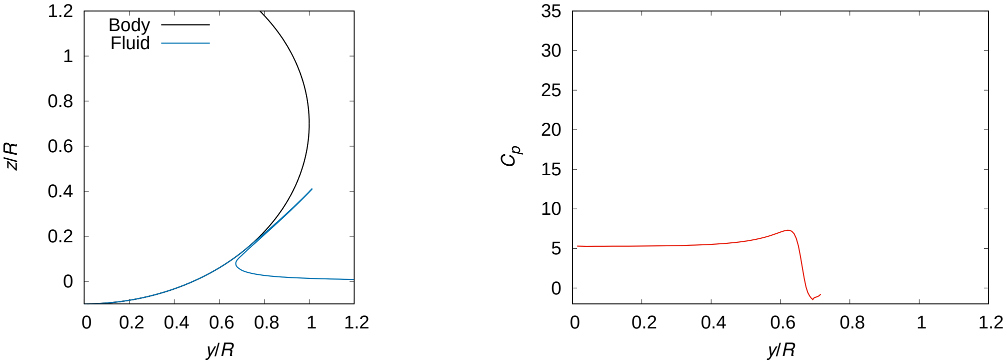


Figure 5: Circular cylinder. Slamming coefficient time history: comparison between the no-separated case and the separated case.



(a) $d/R = 0.085$



(b) $d/R = 0.100$

Figure 6: Elliptical cylinder. Free-surface evolution and pressure coefficient distribution at different time steps after the flow separation.

The results show that the hybrid BEM-FEM approach is able to provide a very good description in terms of free-surface evolution and pressure distribution. The kinematic criterion adopted for the flow separation seems to work reasonably well and the flow separation phenomena is initially captured. The capability of the model to predict the flow separation has been also tested on different body shapes. Despite the first conforting results, further development are needed for managing the numerical instabilities which appear after the flow separation start.

The approach can be also applied it the case of decelerating water entry of a wedge or a cone, where, due to the reduction of the entry velocity, the separation may be anticipated and the flow might detach from the body contour well before the chine [6].

The present work presents a further extension of the numerical model that goes towards the development of a 2D+t model to be applied to either the aircraft ditching or to the hydrodynamics of high-speed planing hulls.

## REFERENCES

[1] Greenhow M, Lin W. M., 1983. Nonlinear free surface effects: Experiments and theory. Report no. 83-19. Department of Ocean Engineering, MIT.

[2] Iafrati, A., Battistin, D., 2003. Hydrodynamics of water entry in presence of flow separation from chines. Proceedings of the 8th International Conference on Numerical Ship Hydrodynamics, Busan, Korea.

[3] Zhao, R., Faltinsen, O. M., Aarsnes, J. V., 1996. Water entry of arbitrary two-dimensional sections with and without flow separation. In: Proceedings of twenty-first symposium on naval hydrodynamics, Trondheim, Norway.

[4] Sun, H., Faltinsen, O. M., 2006. Water impact of horizontal circular cylinders and cylindrical shells. Applied Ocean Research, 28, 299-311.

[5] Korobkin, A. A., Khabakhpasheva, T. Maki, K. J., 2017. Hydrodynamic forces in water exit problems. Journal of Fluids and Structures 69, 16-33.

[6] Del Buono, A., Bernardini, G., Tassin, A., Iafrati, A., 2021. Water entry and exit of 2D and axisymmetric bodies. Journal of Fluid and Structures 103, 103269.

[7] Iafrati, A., Grizzi, S., 2019. Cavitation and ventilation modalities during ditching. Physics of Fluids, 31, 052101.

[8] Battistin, D., Iafrati, A., 2004. A numerical model for the jet flow generated by water impact. Journal of Engineering Mathematics, 48, 353-374.

[9] Longuet-Higgins, M. S., Cokelet, E. D., 1976. The deformation of steep surface waves on water. I. A numerical method. Proc R Soc London, A350, 1-26.

[10] Campbell, I.M.C., Weynberg, P.A., 1980. Measurement of parameters affecting slamming. Report No 440, Wolfson Unit of Marine Technology, Tech. Rep. Centre No. OT-R-8042, Southampton, UK.

[11] Greenhow, M., 1988. Water-entry and -exit of a circular cylinder. Applied Ocean Research 10, 191–198.

[12] Zhu, X., Faltinsen, O. M., and Hu, C. 2007. "Water Entry and Exit of a Horizontal Circular Cylinder." ASME. J. Offshore Mech. Arct. Eng. 129(4): 253–264.

[13] Battistin, D., Iafrati, A., 2003. Hydrodynamic loads during water entry of two-dimensional and axisymmetric bodies. Journal of Fluids and Structures, 17, 643-664.