Flow separation model for the water entry of smoothly curved bodies

A. Del Buono*[1,2], A. Iafrati[1], G. Bernardini[2,1]
* alessandro.delbuono@inm.cnr.it

[1]CNR-INM, INstitute of Marine engineering, 00128 Rome, Italy
[2]University Roma Tre, 00146 Rome, Italy

1 Introduction

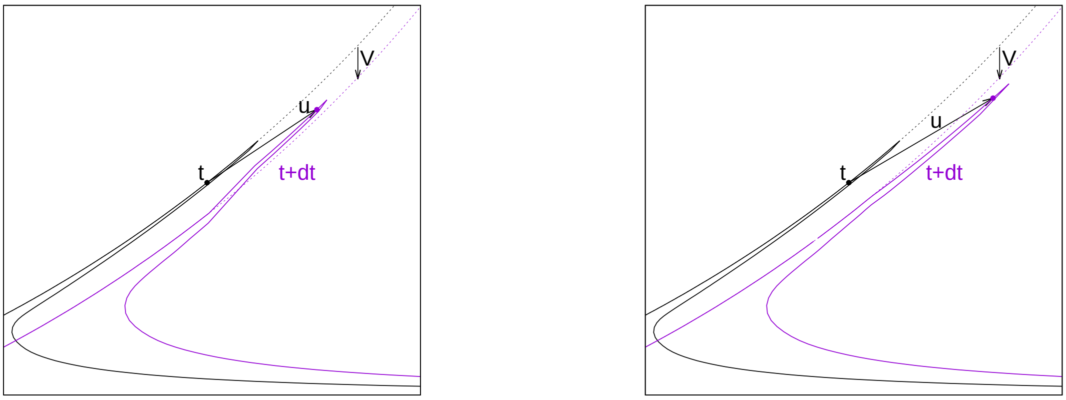
During the water impact of a body, owing to the geometrical properties of the body contour or to the variation of the entry velocity, the flow can detach from the body surface [1]. For a correct prediction of the free-surface dynamics and of the hydrodynamic loads, an accurate modelling of the flow separation phenomena is necessary. 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. Conversely, in the case of bodies with 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. Often, it is assumed that the flow detaches from the body contour at the point where the pressure drops below the atmospheric value at least on a reasonably large portion of the wetted area [4]. However, such a criterion seems too strong as there is evidence that negative pressure can occur without flow separation [5, 6]. In this paper, a new flow separation model, based on a kinematic criterion, is presented. The aim is to further extend the capabilities of the 2D fully non-linear potential flow model, proposed and validated in [7, 8] and more recently in [9], to deal with the water impact of bodies with smoothly curved contours. After a discussion of the theory lying behind the model, an application to the water entry of a 2D circular cylinder impacting at constant entry velocity is presented.

2 Numerical method

The water impact problem is modelled under the hypotheses of irrotational incompressible inviscid flows and is formulated via a boundary-element representation of the velocity potential. The resulting boundary value problem is numerically solved through a Boundary Element Method, discretizing the fluid contour with straight line panels where a piecewise constant distribution for the velocity potential and its normal derivative is assumed. The solution of the problem provides the velocity field on the free-surface which is followed in a Lagrangian way by integrating in time the kinematic boundary condition. A second order Runge-Kutta scheme is used for time integration. 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 [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 to represent the velocity potential inside each control volume.

In the impact of a convex body, the separation point has to be determined as a part of the solution. In this perspective, a new kinematic model is under development. The adjective kinematic is used because the model is based on the relative vertical motion between the latest fluid panels still attached to the body and the body itself. The model 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 applying a Dirichlet boundary condition. The velocity field is evaluated by solving the boundary value problem and 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 1). At the end of the time step there exist two possibilities:

- The fluid panels are above the body contour (Figure 1a): it means that the fluid particles are still able to follow the body contour and, although developing a slightly negative pressure for centripetal acceleration,



(a)

(b)

Figure 1: Sketch of the kinematic model procedure. The dashed lines represent the body contour; the continue lines represent the fluid boundary. a) The fluid panels penetrate the body: they are still attached. b) The fluid panels don't penetrates the body: they are separated and the flow separation starts.

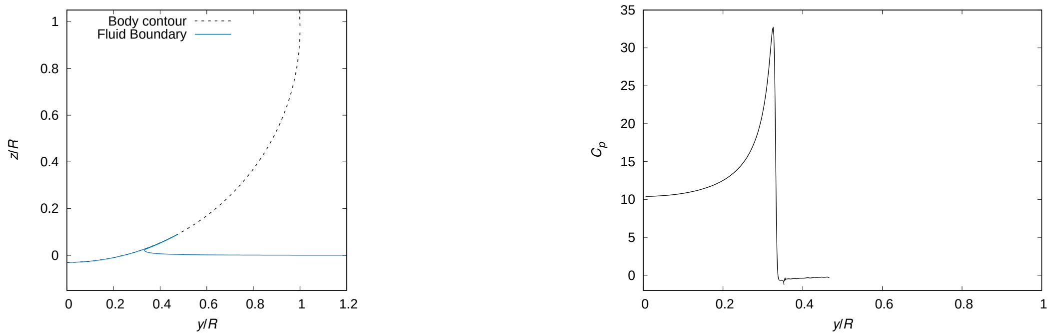
the flow remains attached. Hence, the solution is restarted from the beginning of the time step applying the standard body boundary condition on the panels under consideration and is advanced in time again.

- The fluid panels are below the body contour (Figure 1b): 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.

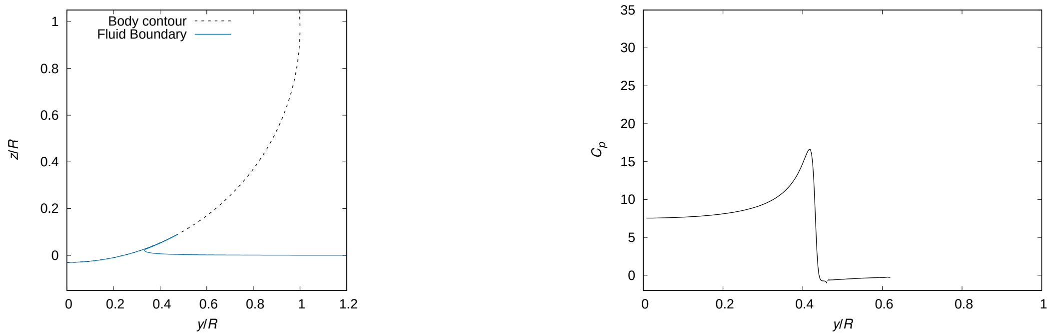
At the next time steps, the above procedure is repeated as it is until the flow separation occurs. Once the separation starts, the same procedure is applied to the last fluid panel lying on the body contour and attached to the separated portion.

3 Results

The proposed model has been tested in the vertical water entry with constant velocity of a 2D circular cylinder. This case was already studied with a pure Boundary Element Method in [7] where the thinnest part of the jet is truncated. Figure 2 shows, for different time steps, expressed in terms of the ratio between the current depth d and the radius of the circular cylinder R, the results, obtained before flow separation, in terms of fluid boundary evolution (left figures) and pressure coefficient distribution (right figures). The latter is defined as $C_p = p/(\frac{1}{2}\rho V^2)$, where the pressure p is computed trhough the unsteady Bernoulli's equation, ρ is the fluid density and V is the body velocity. 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 the jet root which progressively diminishes in time as the local deadrise angle increases. The pressure inside the thin jet is essentially negligible 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: this is another reason why the separation model based on negative pressure has been discarded here. The kinematic model is activated at the time step corresponding to $d/R = 0.075$ and seems working reasonably well, as shown by the results in the figure 3. The flow starts to detach from the body and the lenght of the separated part increases and updates regardless of the one still attached to the body. The pressure continues to decrease and its peak progressively disappears. Although the procedure works well and seems to be consistent with the physics of the problem, further investigations are needed because some instabilities issue born in the transition region between BEM and FEM solution, and the simulation stops shortly after the separation start.

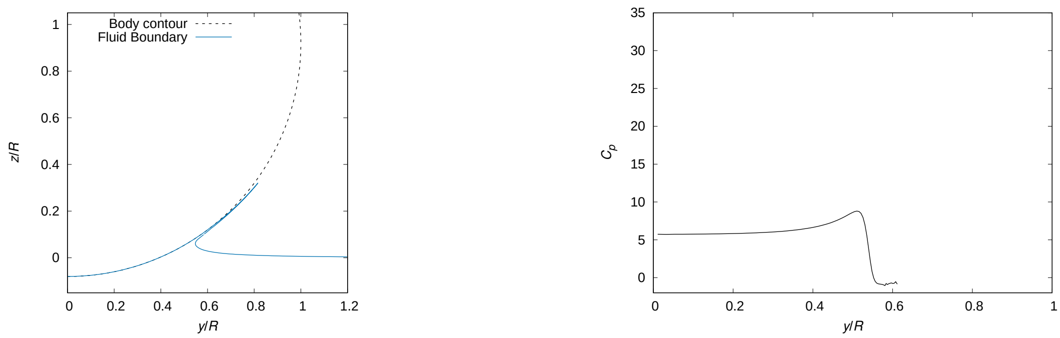


(a) $d/R = 0.030$

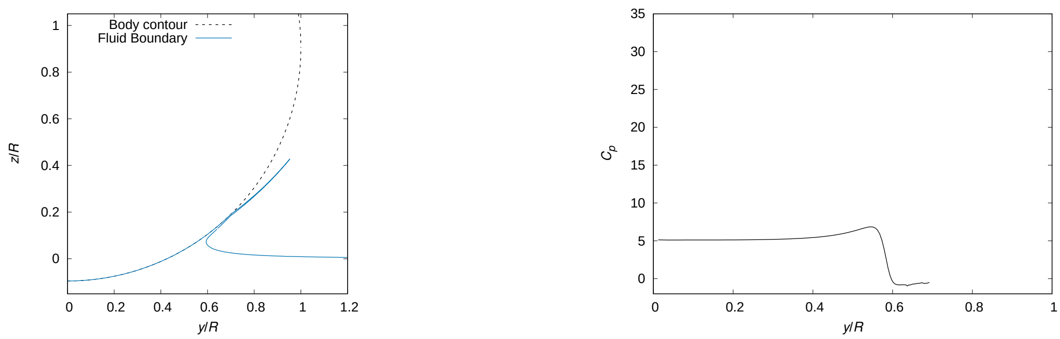


(b) $d/R = 0.050$

Figure 2: Flow evolution and pressure coefficient distribution at different time steps before the flow separation.



(a) $d/R = 0.080$



(b) $d/R = 0.095$

Figure 3: Flow evolution and pressure coefficient distribution at different time steps after the flow separation.

4 Conclusion

A fully non-linear potential flow model based on the hybrid BEM-FEM approach was proposed for the water impact description of 2D bodies with smoothly curved surface. The approach was developed with a flow separation model, based on a kinematic criterion, in order to retrieve the unknown flow separation point as a part of the numerical solution. The proposed model has been tested in the water entry with constant velocity of a 2D circular cylinder and seems to work reasonably well, but further development are needed for managing the numerical instability which appear after the flow separation. Moreover, as the instant when the kinematic model is activated is chosen in terms of a limiting depth, the aim could be to reduce the dependance of the solution to the input data. Finally, the approach could be also used in the decelerating water entry of a wedge or a cone, where, due to the deceleration, the separation may be anticipated and the flow detaches from the body contour before the point of geometric singularity [9].
This work presents an 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.

Acknowledgment

This project has been partly funded from the European Union's Horizon 2020 Research and Innovation Programme under Grant Agreement No. 724139 (H2020-SARAH: increased SAfety & Robust certification for ditching of Aircrafts & Helicopters).

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

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

[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] Del Buono, A., Iafrati, A., Tassin, A., Ianniello, S., 2020. A fully-nonlinear potential flow model for water entry/exit in aircraft ditching applications, 35th IWWWFB, Seoul, South-Korea.