A fully-nonlinear potential flow model for water entry/exit in aircraft ditching applications

A. Del Buono*[1,2], A. Iafrati[1], A. Tassin[3], S. Ianniello[1]
* alessandro.delbuono@inm.cnr.it

[1]CNR-INM, INstitute of Marine engineering, 00128 Rome, Italy
[2]University Roma Tre, 00146 Rome, Italy
[3]IFREMER, Marine Structures Laboratory, Z.I. Pointe du Diable, Plouzan'e, 29280, France

1 Introduction

The aircraft ditching, despite being a rare event, has to be considered in the design phase to guarantee safety and to respect certification. As a way to avoid the expensive full scale experimental tests, computational approaches able to provide a reasonably accurate description of hydrodynamics and fluid-structure interaction taking place during the aircraft ditching are of primary interest. Besides high fidelity, fully coupled, fluid and structural solvers, fast and efficient solvers, albeit approximate, are strongly needed by aircraft manufacturers for the design and certification stage when many different configurations have to be analyzed. Simplified methods based on Modified Logvinovich Model [1] or Generalized Wagner [2] have been found to be very efficient and able to provide accurate predictions of the sectional forces. In this paper, a fully nonlinear model is developed with the aim of providing a more accurate prediction of the pressure distribution and of the fluid-structure coupling. The model is based on the mixed Eulerian-Lagrangian BEM formulation. As a preliminary step towards the development of a 2D+t procedure, in this paper the model is tested in the vertical water entry and exit of a wedge and a cone and validated against other numerical or experimental data.

2 Numerical method

The model is based on the incompressible and irrotational flow assumptions and makes use of a boundary integral representation of the velocity potential. As a classical mixed Eulerian-Lagrangian approach [3], the velocity potential is assigned on the free surface by integrating the unsteady Bernoulli equation, whereas its normal derivative is assigned on the body contour. The free surface evolution is followed using a Lagrangian approach by integrating in time the kinematic boundary condition.

The problem is solved numerically by discretizing the fluid boundary by straight line segments and assuming a piecewise constant distribution of the velocity potential and of its normal derivative. A second order Runge-Kutta scheme is employed for time integration to update the free surface shape and the velocity potential on it. In order to reduce the high computational effort required by a detailed description of the thin jet developing along the body, the simplified hybrid BEM-FEM model proposed in [4] is used for the thinnest part of the jet. The model has been validated in the case of 2D and axisymmetric body impact with constant velocity [4, 5], and it is found to be able to correctly describe the free surface dynamics together with an accurate evaluation of the pressure distribution and loads on the body surface.

For the application to the ditching problem, the method cannot be limited to the water entry with constant velocity. By looking at the ditching problem in a 2D+t manner, in the rear part of the fuselage the problem is more a water exit than a water entry problem. For this reason, the hybrid BEM-FEM approach has been tested on the water impact problem with imposed entry/exit motion. With respect to the entry phase, where the model performs satisfactorily, the description of the exit phase seems more critical and some instabilities issues arise. In order to improve the stability of the numerical solution, two different strategies have been tested. The first one is to cut the jet at the onset of the exit phase. In fact, in this phase, the jet is very thin and its length continues to increase and it could be easily neglected. In the cut procedure, a new panel, orthogonal to the local body surface, is introduced (Fig.1): the corresponding unknown potential is found by BEM solution and a new equation is added for the corresponding unknown potential normal derivative

$$\phi_n(N_{B+1}) = -\phi_s(N_B)$$

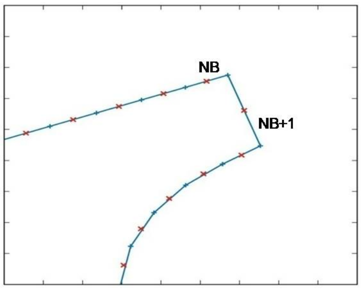


Figure 1: Cut procedure

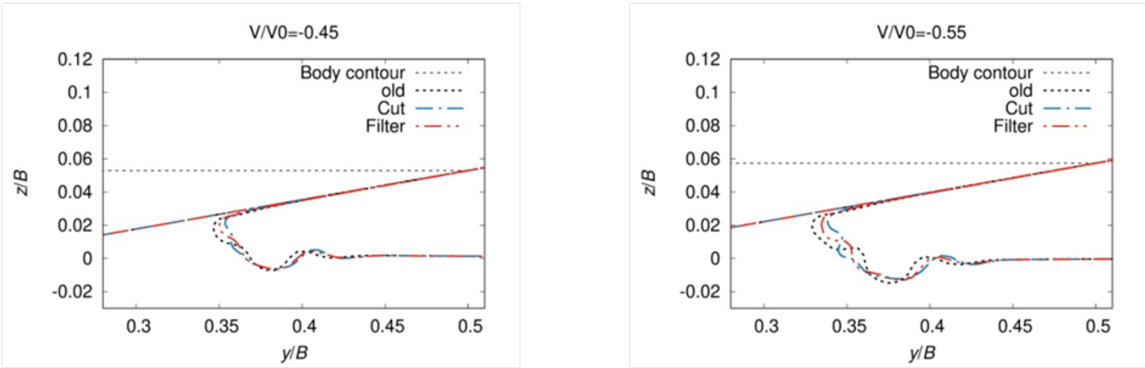


Figure 2: Free surface evolution

which is imposed equal to the negative potential tangential derivative at the last panel on the body side. The second strategy is to increase, during the exit phase, the action of the numerical filter used to overcome the saw-tooth instability of the BEM solution.

3 Results

Two different cases have been tested: a wedge with linearly reducing entry velocity and a cone with sinusoidal motion.

In the first case a 10 degrees rigid wedge is considered. The body impacts the water with a given initial (downward) velocity, $V_0 = 4m/s$, and then the velocity is linearly reduced in time $V(t) = V_0 - V_1 t$, where V_1 depending on the deadrise angle and the width of the wedge: at non dimensional time ($t^* = tV_1/V_0$) equal 1 there is the transition from entry to exit conditions (Fig.5a). The figure 2 shows the free surface evolution obtained without strategies, called *old*, cutting the jet in the exit phase, called *cut*, and increasing the filter order in the exit phase with respect to *old*, called *filter*. Both strategies enable an improved stability of the solution and allow to simulate almost the entire exit phase. Figure 3 shows a comparison between the free surface configurations at two different times, one in the entry and one in the exit phase; as shown in CFD results [6], the wetted length increases faster than it decreases in the exit phase. Figure 4 shows the pressure coefficient distribution acting on the body at different time instants for both phases: for simplicity, only the results of the *filter* strategy are reported. It can be seen that in the entry phase, i.e. when the body is decelerating, the pressure decreases, as does its peak, until it becomes negative. In the exit phase, i.e. when the velocity is negative and increases in amplitude, the pressure remains negative but diminishes in amplitude. This behavior affects the non-dimensional force which is derived by pressure integration along the wetted surface. The force is positive in the first part entry stage and turns negative afterwards, approaching zero during the exit phase (Fig.5b); moreover the negative peak occurring in the exit phase is larger than the positive one occurring during the entry phase. A quite satisfactory agreement with other numerical results available in literature [1, 6] is achieved, especially for the entry phase.

The second case is based on experiments made in the current-wave flume of IFREMER located in Boulogne-sur-Mer (France), for a 15 degrees rigid cone [7]. The use of a transparent mockup and a LED edge-lighting system with a high speed video camera placed above the mock-up allows to follow the evolution of the wetted

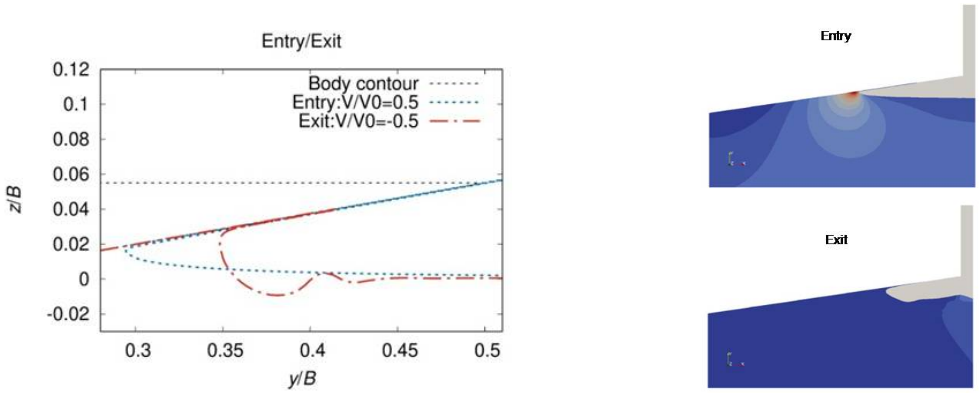


Figure 3: Wetted length during entry and exit phase. On the right there are CFD results [6]

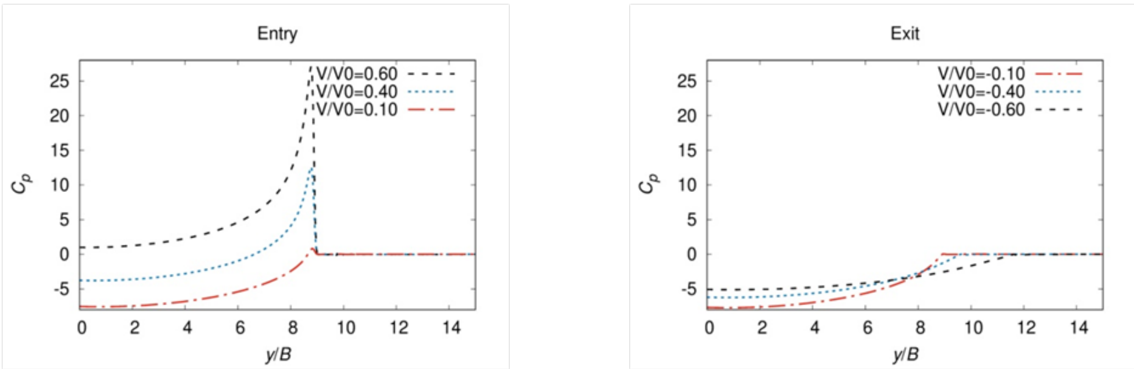


Figure 4: Pressure coefficient distribution

surface during the combined entry and exit phase, where the cone moves with a sinusoidal imposed motion

$$\begin{cases} z(t) = -H_{max}\sin(\omega t), & t \leq T/2 \\ \dot{z}(t) = -U_{max}, & t > T/2 \end{cases}$$

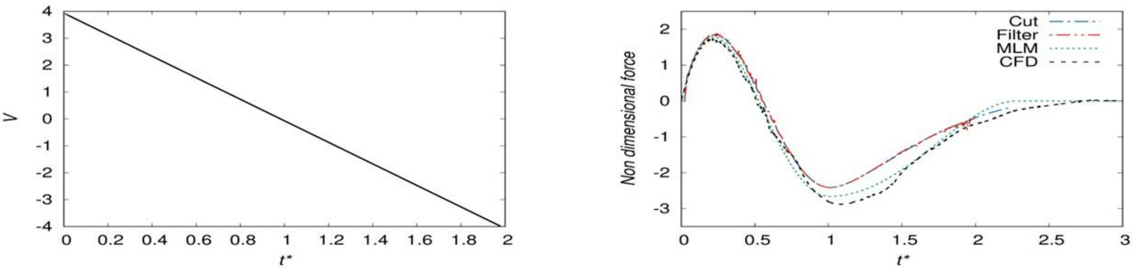
where H_{max}, is the maximum penetration depth, $\omega = U_{max}/H_{max}$ with U_{max} maximum initial (downward) velocity and $T = 2\pi H_{max}/U_{max}$.

Even though the strategies described above increase the stability of the solution, in this case the simulation still stops before the exit phase is completed. However, the comparisons in terms of time histories of the force and of the contact line (evaluated as the jet root position) shown in Fig.6b and Fig6c display a good agreement with the experimental data for the case with $U_{max} = 0.57m/s$. Results are drawn starting from the time at which the cone touches the water in the experiments ($t \simeq 3s$).

4 Conclusion

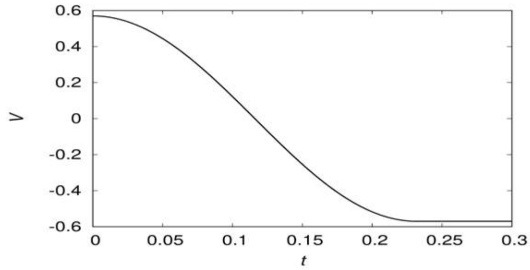
A fully non linear potential model has been used for the description of the water entry and exit problems of 2D and axisymmetric bodies. Two different strategies aimed at improving the stability of the free surface shape, particularly in the exit phase, have been proposed and tested separately. Both of them have been found able to enhance the stability of the solution and therefore they could be deeper investigated and, possibly, used in combination to improve the stability even further. Additional results in this respect will be presented at the workshop.

Acknowledgment
This project has received funding from the European Union's Horizon 2020 Research and Innovation Programme under grant agreement No 724139 (H2020-SARAH)

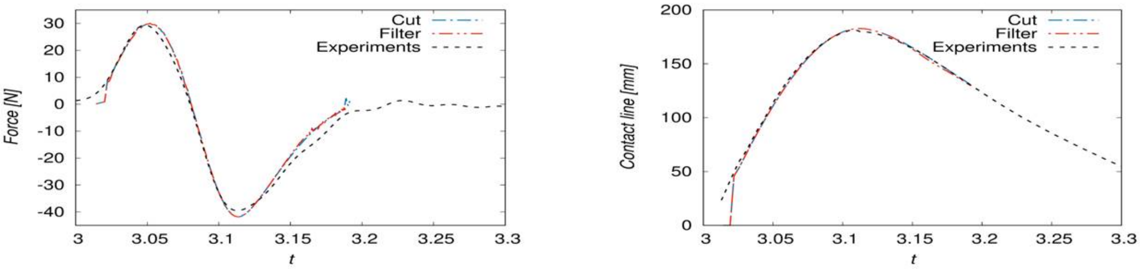


(a) (b)

Figure 5: Velocity (positive downward) (a) and non dimensional force (b) distribution for the wedge case



(a)



(b) (c)

Figure 6: Velocity (positive downward) (a), non dimensional force (b) and contact line (c) distribution for the cone case

References

[1] Tassin, Piro, Korobkin, Maki, Cooker, *Two-dimensional water entry and exit of a body whose shape varies in time*, J Fluids Struct, Vol. **40**, pp. 317-336, (2013).

[2] Gropengießer, Rung, *Computational Modelling of Aircraft Ditching with two-way Fluid-Structure-Interaction*, ECCOMAS Congress (2016)

[3] Longuet-Higgins, Cokelet, *The deformation of steep surface waves on water. I. A numerical method*, Proc R Soc London, Vol A350, 126, (1976).

[4] Battistin, Iafrati, *A numerical model for the jet flow generated by water impact*, J Engng Math, Vol. **48**, pp. 353-374, (2004).

[5] Battistin, Iafrati, *Hydrodynamic loads during water entry of two-dimensional and axisymmetric bodies*, J Fluids Struct, Vol. **17**, pp. 643-664, (2003).

[6] Piro, Maki, *Hydroelastic wedge entry and exit*, 11th International Conference on Fast Sea Transportation, 26–29 September 2011, Honolulu, HI, USA.

[7] Breton, Tassin, Jacques *Experimental investigation of the water entry and/or exit of axisymmetric bodies*, submitted to Journal of Fluid Mechanics (2020).