VERTICAL WATER ENTRY OF A 3D FUSELAGE: COMPARISON BETWEEN A POTENTIAL-FLOW MULTISECTION APPROACH AND A FULLY 3D CFD SOLUTION

A. Del Buono*[1] and A. Iafrati[1]
[1]CNR-INM, INstitute of Marine engineering
Via di Vallerano 129 00128 Rome, Italy
*alessandro.delbuono@cnr.it, alessandro.iafrati@cnr.it

ABSTRACT: This paper proposes the analysis of the hydrodynamic behaviour of the vertical water entry of a scaled fuselage. The investigation is carried out with a low-fidelity multisection approach which exploits a 2D fully non-linear potential flow solver based on a hybrid BEM-FEM approach. The problem is also investigated through a high-fidelity CFD simulation by using the open-source CFD library OpenFOAM. The aim is to compare the prediction obtained by the multisection approach with those obtained by the 3D CFD solution, in order to highlight the advantages and limitations of the former, which represents an approximated solution of the fully 3D problem.

1. INTRODUCTION

Aircraft ditching is a very strong event which includes four phases such as approach, impact, landing and floatation [1]. Although it happens quite rarely, it needs to be considered in the design phase to guarantee safety and ensure the certifications' respect. The investigation and prediction of the ditching phenomenon is a quite challenging problem which can be investigated through both experimental tests and computational approaches. Examples of first attempts to investigate experimentally the ditching phenomenon can be found in [2], where ditching tests on rectangular plates with different pitch angles and vertical/horizontal velocity ratio are reported, and in [3] which analyzed the free flight ditching of several fuselage shapes highlighting the effect of the rear fuselage curvatures on the aircraft dynamics. More recently scaled model tests on full aircraft configurations, scaled with the Froude similarity, are presented in [4, 5]. However, the use of the scaled model represents a limit as cavitation and ventilation phenomena cannot be correctly predicted [4] as the similitude in terms of the ratios between stagnation pressure and both atmospheric and vapor pressure are not respected. It represents a criticality also in presence of significant structural deformations that affect the hydrodynamic loading [6]. For these reasons, several quasi-full scale tests were recently conducted in High Speed Ditching Facility designed and built at National Research Council–Institute of Marine Engineering (CNR-INM) and presented in [7]. Regarding the use of computational approaches, in [8, 9] there are examples of ditching simulations focused on the description of fluid structure interaction. The hydrodynamics loads are computed with a Smoothed Particle Hydrodynamics (SPH) and the Finite Element Method are adopted as structural model. SPH method has been also used for ditching applications in [10, 11]. Furthermore in [12, 13] ditching simulations have been performed with CFD approach which solves the unsteady Reynolds averaged Navier–Stokes (URANS) equations in conjunction with the Volume of Fluid method (VoF), which is used to capture the water–air interface. Both SPH and CFD represent high fidelity approaches that require a high computational effort. For this reason, also the use of fast and efficient solvers, albeit of lower fidelity, is of primary interest to aircraft manufacturers to have a quick but at the same time accurate estimate of the hydrodynamic loads acting on the aircraft, to be effectively exploited in its conceptual and preliminary design phases. This is particularly true during the aircraft optimization process where many different configurations have to be analyzed. In this perspective, developing a 2D+t procedure which exploits a 2D fully non-linear potential flow model [14, 15] to describe the hydrodynamics of the aircraft ditching phenomenon may be a reliable low time-cost alternative. The 2D+t approach is based on a slender body approximation and reduces the 3D problem into a series of 2D cross section problems, with the shape of the section changing in time, in an earth-fixed frame of reference. As a first step through the implementation of a 2D+t approach, a fully non-linear potential flow multisection procedure has been developed in [16] for computing the vertical water impact of a fuselage. Differently from the 2D+t method, in the multisection procedure the forward velocity in the longitudinal direction is not considered and the shape of the 2D sections don't change in time during the immersion.

In this paper the potential-flow multisection procedure is used to analyze the hydrodynamic behaviour of the vertical water entry of a scaled fuselage. A fully 3D solution is also provided by computing a CFD simulation. In particular, the vertical water entry of the 3D scaled fuselage is also solved through the open-source CFD library OpenFOAM, a widely used software in the literature [17, 18]. A comparison in terms of free surface shape and pressure distribution along some cross section planes and 3D hydrodynamic force time history is proposed. The objective of the comparison is highlighting the advantages and limitations of the multisection approach, which represents an approximated solution of the fully 3D problem. The present paper outlines as follows. First, the potential-flow multisection approach and the CFD method are briefly described; then, results on the chosen test case are presented and discussed; finally, conclusions are drawn.

2. NUMERICAL METHODS

The vertical water entry of a scaled fuselage is here faced with a potential flow multisection approach proposed in [16]. This method exploits a slender body approximation to reduce the 3D problem into a series of 2D water entry problems of different fuselage cross sections in an earth-fixed frame of reference (see Fig. 1). This is possible when the longitudinal derivatives are of lower order with respect to those in the transversal plane and can be neglected. So the multisection approach provides an approximated solution in which 3D effects along the longitudinal direction are ignored. In the method here used, each 2D water entry problem is solved independently using a 2D fully non linear potential flow model based on a Hybrid BEM-FEM approach proposed and validated in [19] and more recently in [20, 21]. The water entry problem is studied under the hypotheses of incompressible fluid and irrotational flow and is formulated in terms of boundary-element representation of the velocity potential. The time evolution is described by a mixed Eulerian-Lagrangian approach. The corresponding unsteady boundary value problem is numerically computed by discretizing the fluid contour with straight line panels where a piecewise constant distribution for the velocity potential and its normal derivative is assumed. To solve the discretized problem, a hybrid BEM-FEM approach is adopted, where the Boundary Element Method is coupled with a simplified Finite Element Method, which is used for describing the thinnest part of the jet and reduce the computational effort [19]. The solution of the problem provides the velocity field on the free surface which is followed in a Lagrangian way by integrating in time, using a second order Runge-Kutta scheme, the kinematic boundary condition. The time step for the integration in time is governed by the Courant number. The pressure exerted on the wetted area is determined using the unsteady Bernoulli equation, while the force is computed by integrating the pressure along the wetted area. When used in the multisection procedure, the Hybrid BEM-FEM solver requires a careful definition of

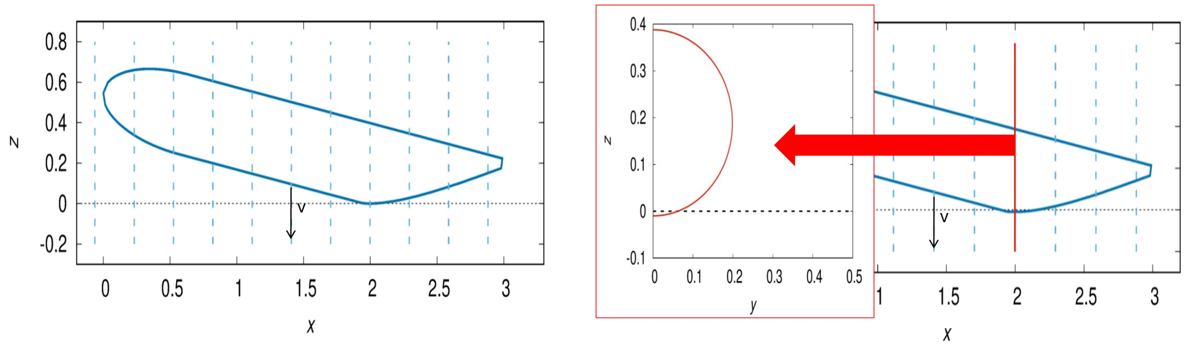

Figure 1 - Multisection approach: the vertical dashed lines represent the equally spaced vertical space-fixed cross planes; the corresponding 2D section on the y-z plane is derived by linear interpolation and represents the 2D body geometry that the hybrid BEM-FEM solver uses to solve the 2D water entry problem on that plane.

the initial discretization of each fuselage section to ensure the numerical solution stability [16]. In this perspective, an algorithm which exploits the relationship on the velocity singularity at the intersection point between the body contour and the free surface and the Courant number definition is applied [16]. In particular, the following relationship is proposed for setting the size Δl of the first free surface panel of the new activated section:

$$\Delta l = \left(\frac{\sigma 2^{\sigma-1} \Delta t}{C} \right)^{1/(2-\sigma)} , \quad \sigma = \frac{\pi}{2(\pi-\beta)} \tag{1}$$

where C is the Courant number and β is the local deadrise angle. The use of Eq. 1 yields an initial discretization that should guarantee a regular evolution of the 2D numerical simulation on each activated section [16], where the free surface shape and the pressure distribution are provided by the 2D Hybrid BEM-FEM solver. Furthermore, it is possible to obtain a sectional force distribution on the whole wetted fuselage, which can be integrated along the fuselage longitudinal axis to estimate the 3D vertical hydrodynamic force, F^{3D}, acting on the entering fuselage. The integration is done numerically with the trapezoidal rule, thus obtaining

$$F^{3D} \simeq \sum_{k=1}^{N-1} \frac{F_{k+1}^{2D} + F_k^{2D}}{2} \Delta x, \tag{2}$$

where N is the number of the activated section and Δx is the distance between the fixed cross planes.

To assess the capability of the potential-flow multisection approach, the same vertical water entry of a scaled fuselage is faced with a CFD simulation as well. This yields a fully 3D solution that can be compared with the approximated solution obtained by using the multisection approach. The CFD simulation is conducted using the open-source finite-volume CFD library OpenFOAM, employing the incompressible multiphase pressure-based solver *interFoam*. The latter solves numerically, with the finite-volume technique, the Navier-Stokes governing equation of the fluid, which is here consider incompressible and laminar. Moreover, the applied solver adopts the Volume-of-Fluid (VOF) interface-capturing method to define the fully non-linear free surface evolution. A dynamic mesh technique is employed to simulate the displacement of the body into the water. This method involves moving both the body and the entire mesh. The grid is built with the *blockMesh* and *snappyHexMesh* dictionaries available in the OpenFOAM library. The former generates a background mesh, whereas the second removes

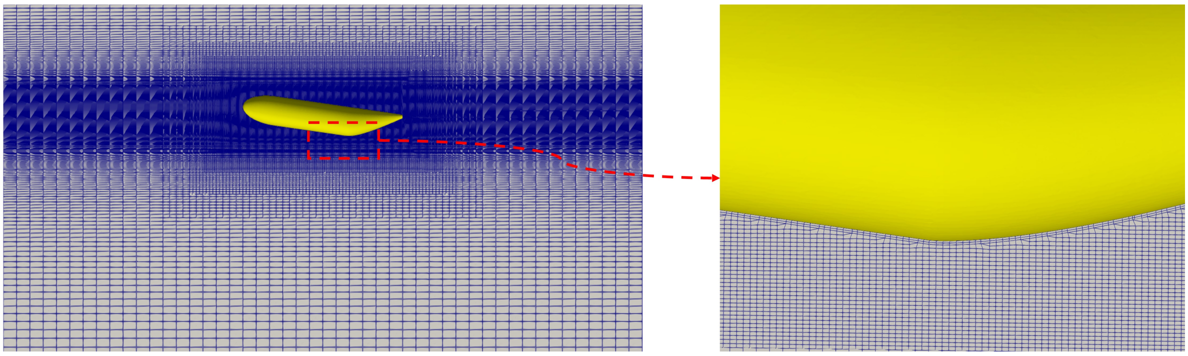

Figure 2 - Global and close-up view in the plane x-z of the mesh used for the CFD simulation.

the cells containing the body and reconfigures the grid around it. As shown in Fig. 2, several refinements close to the body are also adopted. Given the unsteady nature of the application, the time step is adjustable and controlled by the maximum local Courant Number.

3. RESULTS AND DISCUSSION

A vertical water entry of a scaled fuselage has been chosen as test case and is investigated in the following. The body is characterized by circular cross sections and impacts the water with a constant vertical velocity of $1m/s$ and a fixed pitch angle of 6^o. Firstly, the hydrodynamic behaviour along the cross section corresponding to the first contact point is analyzed. The free surface shape and the pressure distribution acting along this section are displayed in Fig. 3 for three different time steps. The behaviour is the same observed in the vertical water entry with constant vertical velocity of a 2D circular cylinder [22]. At the beginning the jet flow rises and remains attached to the body surface and the pressure is characterized by a peak just behind the jet root. Then, flow separation occurs and the separated portion of the flow evolves in time. In terms of pressure, the peak at the jet root gradually diminishes and eventually disappears as a result of the increasing local deadrise angle and flow separation. Moreover, if the effect of gravity is negligible during the water entry, the pressure distribution should became monotonic [22]. Conversely, in this test case, the effect of gravity is evident (and well predict by the numerical methods) and, as a consequence, the pressure peak moves on the bottom of the body section and the overall pressure distribution tends to increase. The comparison with the CFD solution shows how the multisection approach overestimates the wetted surface and the jet root position is further forward. Also the pressure distribution is overestimated, in particular the pressure peak. Such an overestimate is expected as the multisection approach neglects the flow in the fuselage longitudinal direction due to the pressure differences between different sections. In some sense, such differences, due to 3D effects, are the same observed between the 2D and the axisymmetric water impact cases in [23].

A similar behaviour in terms of free surface evolution and pressure distribution can be observed along other two cross section placed in the front and in the rear part of the fuselage respectively (see Figs. 4 and 5). It is worth nothing that the front section has the same shape of the section analyzed before, whereas the rear section is smaller. For both sections the flow rises along the body contour and separates at certain time. Moreover the pressure peak at the jet root progressively disappears, the peak moves on the bottom of the body section and, under the effect of gravity, the pressure globally increases in time. Also for these sections, it is interesting observing the comparison between multisection and CFD results. Specifically, in the case of

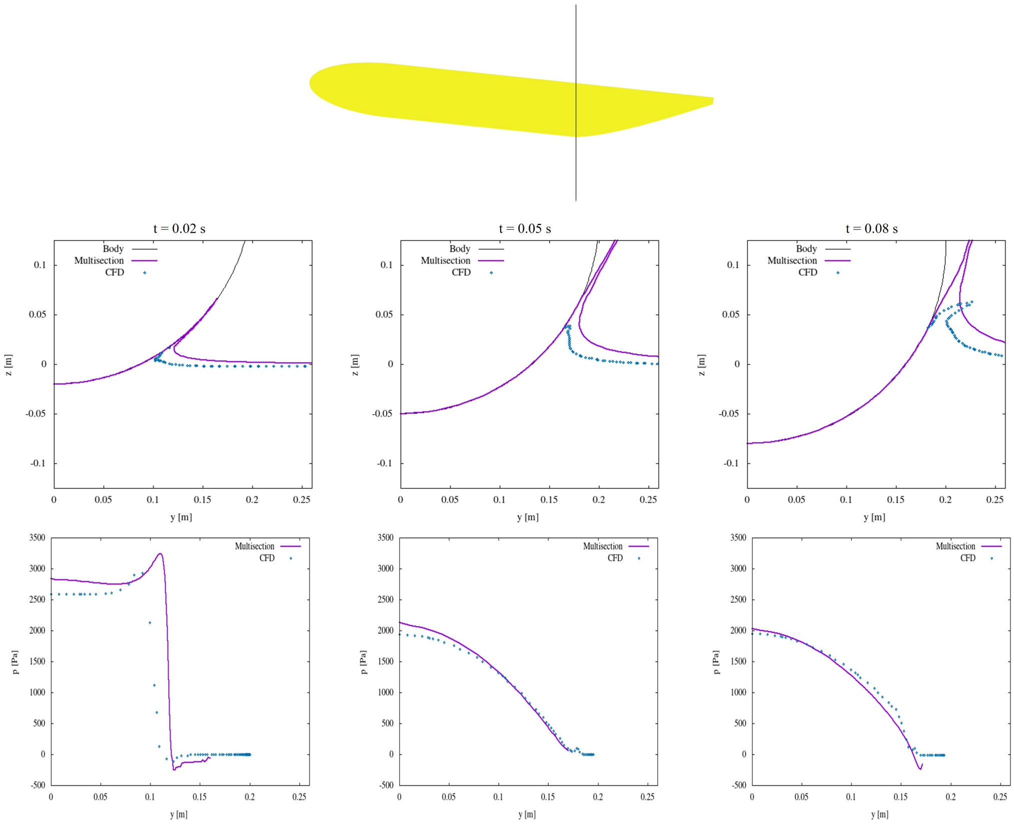

Figure 3 - Free surface shape (first row) and pressure distribution (second row) at three different time step along the first cross section.

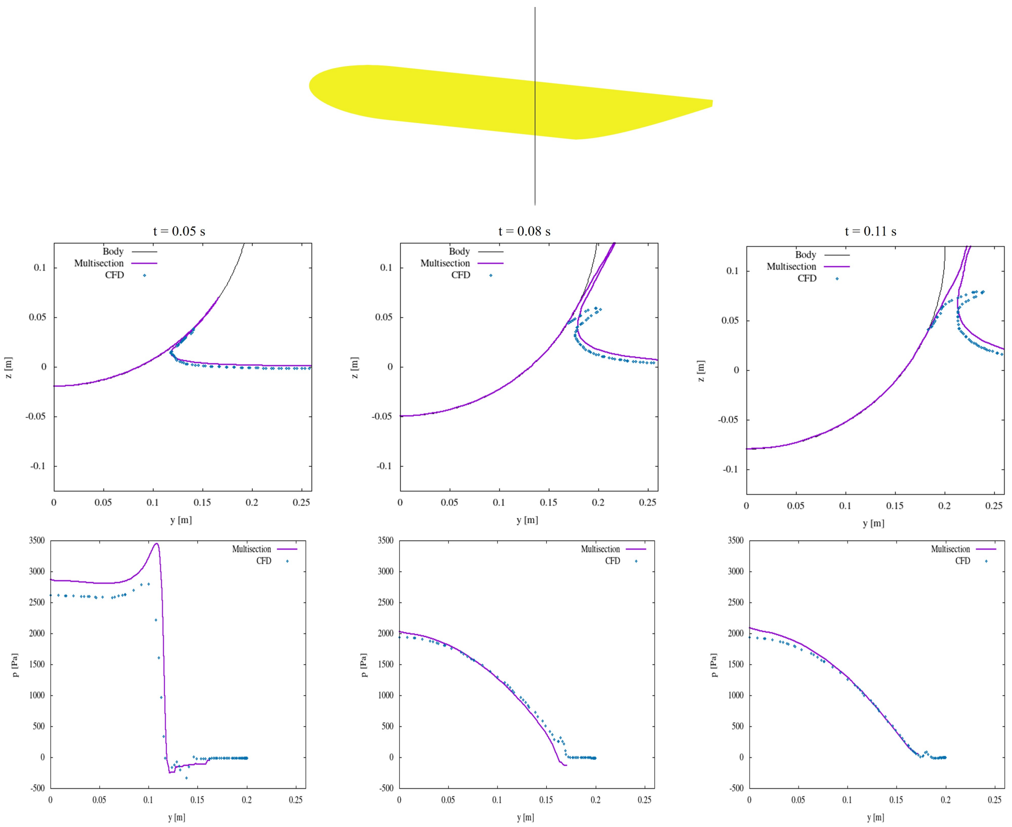

Figure 4 - Free surface shape (first row) and pressure distribution (second row) at three different time step along a cross section in the front part of the fuselage.

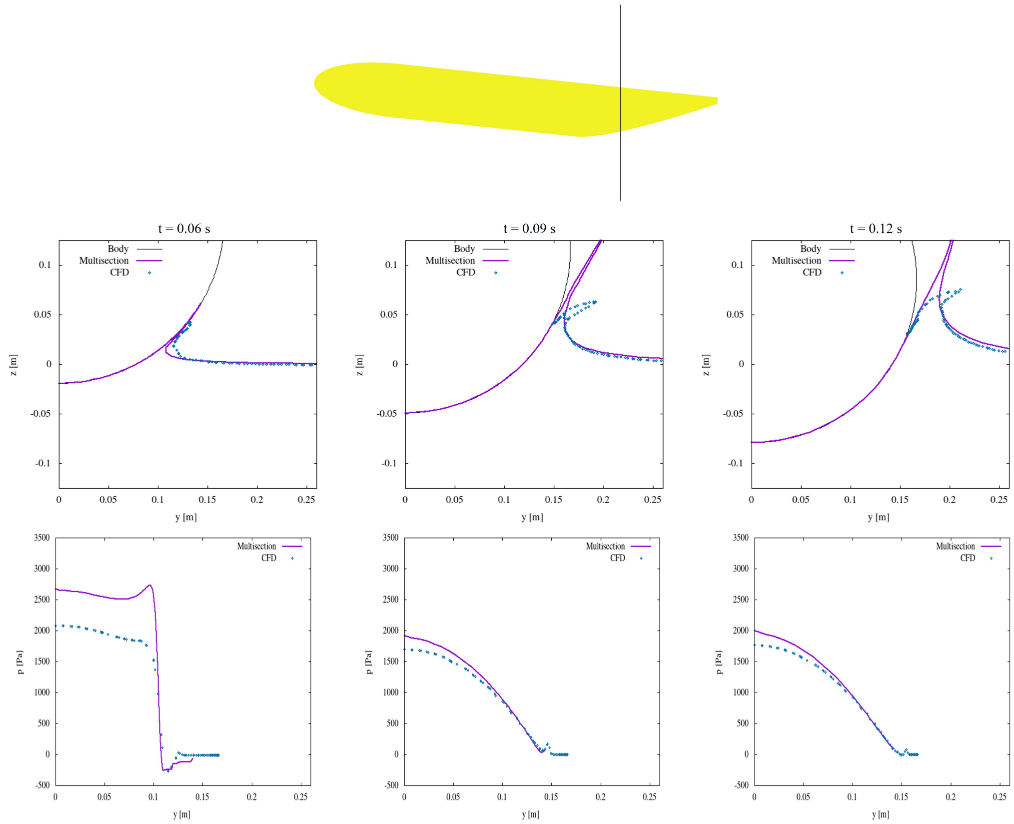

Figure 5 - Free surface shape (first row) and pressure distribution (second row) at three different time step along a cross section in the rear part of the fuselage.

the rear section, at the beginning of the own impact, the multisection approach underestimates the evolution of the free surface. Thus, there is an opposite behaviour compared to what has been observed for the first impacting section. This is due to a longitudinal rise-up effect which is neglected in the multisection approach. As seen in Fig. 6 in the fully 3D case, despite the section being above the still water level (celestial line in the figure), it is already immersed in water due to the longitudinal rise-up. Therefore, in the CFD case, it appears that the sectional water entry problem arises earlier compared to the multisection case (which starts when the section is just below the still water level). As a consequence the jet is further ahead in the former case. This rise-up effect also occurs in the front part of the fuselage, albeit to a lesser extent, which explains why at the beginning of the impact of the front section, multisection and CFD solutions in terms of free surface are very similar. However, the rise-up effect does not affect the pressure, which is always overestimated by the multisection procedure, due to the 3D effects described before for the first impacting section. These effects become pronounced even in terms of the free surface, as the advancement of the free surface in the multisection case is more significant compared to that in the CFD case as simulations evolve over time (see Fig. 5).

The time history of the 3D vertical force acting on the impacting fuselage is now analyzed. As shown in Fig. 7, after certain time, the force tends to increase linearly. This is due to gravity effects which are, as aforesaid, quite important in the test case here investigated. This statement is confirmed by analyzing the time history of the sectional vertical force on the three sections investigated above. This is displayed in Fig. 8, where "Sec 1" is the section at the first contact point, "Sec 2" is the front section and "Sec 3" is the rear section. In absence of gravity effects, the 2D force should rapidly decreases and tends to a monotonic behaviour [22]. Conversely, in this case one can observed that, at each section, the 2D force after an initial rapid decreas-

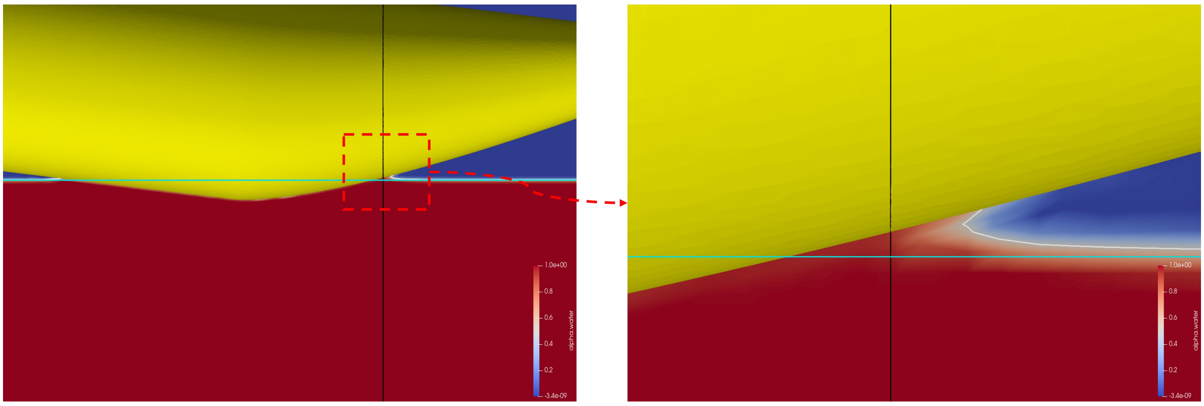

Figure 6 - Rise-up effect along the longitudinal direction. The black vertical line represents the selected rear section. The celestial horizontal line represents the still water level.

ing, starts to increase as gravity contribution becomes dominant. The latter is consistent with was observed in [24] where gravity influence on the water entry of 2D circular ellipses with different eccentricity is presented. By comparing multisection and fully 3D results, one can see how the multisection approach exhibits an overestimation with respect the CFD solution. This is consistent with the previous results as the force strictly depends on the pressure distribution. Please note that the initial step in the multisection curve occurs because the numerical simulation begins with the body already slightly submerged [20]. However, although the multisection approach provides solutions that neglect some 3D effects (as mentioned above), the agreement with the fully 3D solution is satisfactory. It means that it can be consider a valid method to obtain an accurate and quite fast prediction of loads acting on the entering fuselage.

4. CONCLUSION

The present paper proposed the analysis of the vertical water entry problem of a scaled fuselage. The investigation has been conducted with an approximated 3D solution through a potential-flow multisection approach and with a fully 3D CFD simulation. Results in terms of free surface and pressure distribution along three different fuselage sections and in terms of 3D vertical force time history have been provided and compared. The comparison has outlined how neglecting 3D effects in the axial direction leads the multisection method to:

- Globally overestimate the wetted surface along the body sections. As a consequence, the jet root results to be further ahead. However, due to the rise-up effect along the longitudinal direction, which is ignored in the multisection method, an underestimation of the wetted surface by the multisection method has been also observed. The latter is more pronounced in the rear part of the fuselage due to the body curvature and the section narrowing along the longitudinal direction.

- Overestimate the pressure distribution along the body sections. As a consequence, the global vertical force acting on the entering fuselage is also overestimated.

However, considering the multisection solution represents an approximated solution of the fully 3D problem, the agreement with the CFD results is satisfactorily. So the multisection procedure based on a fully non-linear potential flow model proved to be a time-efficient and a reliable approach for providing a reasonable and accurate description of complex 3D water entry problems. This kind of approach can be helpful in an optimization process where many different configurations have to be analyzed or to provide solutions to be used as initial guesses for 3D high-fidelity solvers, thus improving their computational efficiency. As a future perspective, the development of a 3D correction model could make the multisection method even more reliable and applicable, even in problems where 3D effects in the longitudinal direction are particularly pronounced.

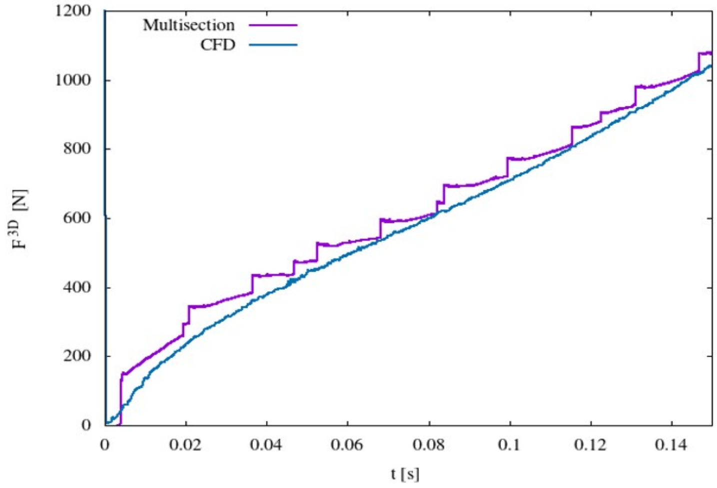

Figure 7 - Time history of the vertical 3D force acting on the impacting fuselage.

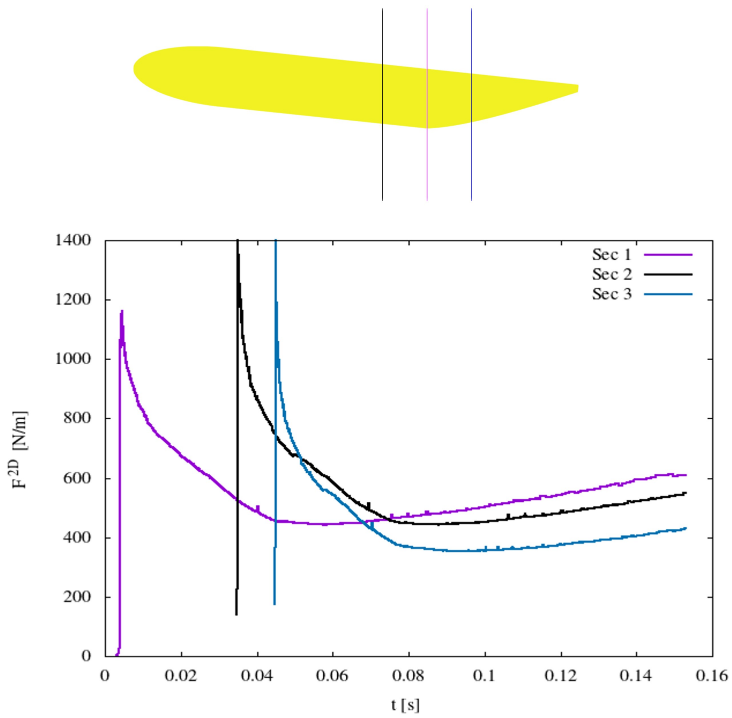

Figure 8 - Time history of the 2D sectional vertical force acting on three different cross sections.

REFERENCES

1. Hughes, K., Vignjevic, R., Campbell, J., Vuyst, T. D., Djordjevic, N. and Papagiannis, L. (2013) From aerospace to offshore: bridging the numerical simulation gaps-simulation advancements for fluid structure interaction problems. *Int. J. Impact Eng.* 61, 48-63

2. Smiley, R. (1951) An experimental study of water-pressure distributions during landings and planing of a heavily loaded rectangular flat-plate model. *Tech. rep., NACA TN 2453*

3. McBride, E. E. and Fisher, L. J. (1953) Experimental investigation of the effect of rear-fuselage shape on ditching behavior. *Tech. rep., NACA TN 2929*

4. Climent, H., Benitez, L., Rosich, F., Rueda, F. and Pentecote, N. (2006) Aircraft ditching numerical simulation. *Proc. of 25th International Congress of the Aeronautical Sciences - ICAS*, 3-8 September, Hamburg, Germany

5. Zhang, T., Li, S. and Dai, H. (2012) The suction force effect analysis of large civil aircraft ditching. *Science China–Technological Sciences* 55

6. Iafrati, A. (2016) Fluid-structure interaction during the high speed water entry of a plate. *Proc. of 18th International Conference on Ships and Shipping Research, The European Marine Energy Centre Ltd.*, pp. 197–206

7. Iafrati, A., Grizzi, S. and Olivieri, F. (2021) Experimental investigation of fluid-structure interaction phenomena during aircraft ditching. *AIAA J.* 59(5), 1561-1574.

8. Siemann, M. H., Schwinn, D. B., Scherer, J. and Kohlgruber, D. (2017) Advances in numerical ditching simulation of flexible aircraft models. *Int. J. Crashworthiness* 23(2), 236-251

9. Siemann, M. N. and Langrand, B. (2017) Coupled fluid-structure computational methods for aircraft ditching simulation: comparison of ALE-FE and SPH-FE approaches. *Comput. Struct.* 188, 95-108

10. Xiao, T., Qin, N., Lu, Z., Sun, X., Tong, M. and Wang, Z. (2017) Development of a smoothed particle hydrodynamics method and its application to aircraft ditching simulations. *Aerosp. Sci. Technol.* 166, 28-43

11. Xiao, T., Lu, Z. and Deng, S. (2021) Effect of initial pitching angle on helicopter ditching characteristics using SPH method. *J. Aircraft* 58(1), 167-181

12. Streckwall, H., Lindenau, O. and Bensch, L. (2007) Aircraft ditching:a free surface/free motion problem. *Archives of Civil and Mechanical Engineering* 7, 177–190

13. Qu, Q., Liu, C., Liu, P., Guo, B. and Agarwal, R. K. (2016) Numerical simulation of water-landing performance of a regional aircraft. *J. Aircraft* 53(6), 1680-1689

14. Iafrati, A. and Broglia, R. (2008) Hydrodynamics of planing hulls: a comparison between RANS and 2D + t potential flow models. *Proc. of 27th Symposium on Naval Hydrodynamics*, 5-10 October, Seoul, Korea, pp. 795–813

15. Sun, H. and Faltinsen, O. M. (2011) Dynamic motions of planing vessels in head seas. *Journal of Marine Science and Technology*, 16 (2011) 168–180

16. Del Buono, A., Bernardini, G. and Iafrati, A. (2021) Multisection approach for the vertical water impact of a fuselage. *Proc. XXVI International AIDAA Congress*

17. Xiang, G., Wang, S. and Guedes Soares, C. (2020) Study on the motion of a freely falling horizontal cylinder into water using OpenFOAM. *Ocean Eng.* 196, 106811

18. Wang, K., Ma, X., Bai, W., Lin, Z. and Li, Y. (2021) Numerical simulation of water entry of a symmetric/asymmetric wedge into waves using OpenFOAM. *Ocean Eng.* 227, 108923

19. Battistin, D. and Iafrati, A. (2004) A numerical model for the jet flow generated by water impact. *Journal of Engineering Mathematics* 48, 353–374.

20. Del Buono, A., Bernardini, G., Tassin, A. and Iafrati, A. (2021) Water entry and exit of 2D and axisymmetric bodies. *J. of Fluid and Structures* 103, 103269

21. Hulin, F., Del Buono, A., Tassin, A., Bernardini, G. and Iafrati, A. (2022) Gravity effects in two-dimensional and axisymmetric water impact models. *J. of Fluid Mech.* 944, A9

22. Del Buono, A., Iafrati, A. and Bernardini, G. (2021). Flow separation model for the water impact problem. *IX International Conference on Computational Methods in Marine Engineering*

23. Iafrati, A. (2007) Free surface flow generated by the water impact of a flat plate. *Proc. 9th International Conference on Numerical Ship Hydrodynamics*

24. Zhang, X., Liu, P., Qu, Q.,Wang, R. and Agarwal, R. K. (2018). Effects of froude number and geometry on water entry of a 2-d ellipse. *Journal of Hydrodynamics*, 30, 738–749