Modeling and validation of guided ditching tests using a coupled SPH-FE approach

Martin H. Siemann
German Aerospace Center
Department of Structural Integrity
70569 Stuttgart, Germany
martin.siemann@dlr.de

Paul H. L. Groenenboom
ESI Group Netherlands
2629 HD Delft, The Netherlands
pgr@esi-group.com

***Abstract*—Oblique water impact of structures with high horizontal speed is investigated using a coupled Smoothed Particle Hydrodynamics – Finite Elements (SPH-FE) approach. The paper introduces the topic of aircraft ditching analysis, comprising its necessity together with current challenges. A brief overview of an extensive experimental campaign of guided ditching tests carried out in the FP7-SMAES project is given. Numerical simulations of these tests have been conducted by hybrid SPH-FE models. Key code improvements relevant for state of the art ditching simulations such as translating periodic boundary conditions and spatially non-uniform initial particle distributions using weighted Voronoi tessellation will be presented. Comparison with experimental data in terms of force and strain results is established with the aim to validate the numerical model. Good correlation could be proven for test cases with purely elastic as well as elastic-plastic aluminum panels.**

I. Introduction

Emergency landing on water (ditching) needs to be studied during aircraft design and certification processes for aircraft certified for extended overwater operations. Hence, it applies for the majority of, if not all, passenger and military aircraft.

The analysis of ditching in research and industry tends towards the (supportive) use of numerical simulation tools [2, 3, 5, 26] due to a number of limitations and drawbacks related to experimental testing which — besides comparison to aircraft with similar design having previously passed ditching certification — is the state of the art means to proof ditching capabilities [5, 22, 24, 31]:

- extensive financial and temporal efforts of test campaigns,
- testing of full-scale aircraft is economically unfeasible,
- inherent but undesirable scale effects in hydrodynamics of sub-scale experiments,
- reduction to rigid sub-scale models, which do not accurately account for deformation and hydroelasticity,
- limitations in terms of design changes and amount of scenarios to be tested,
- limited amount of measurable results (virtually it is possible to display results in manifold ways), etc.

There is a variety of semi-analytical methods utilized to simulate water impact, e.g. [25, 30]. However, when regarding highly non-linear structural behavior with potential failure of complex and realistic structures, semi-analytical approaches are no longer suitable to solve this fluid-structure interaction problem and advanced numerical methods are needed.

The violent nature of a ditching event, which includes large fluid deformations and complex free surface shapes, identifies the Smoothed Particle Hydrodynamics (SPH) method as well suited for the solution. In order to analyze the structural response including deformation and potential damage, a coupled approach of SPH and the Finite Element (FE) method is chosen.

Challenges for the numerical simulation arise from sharp gradients with extremely small time and spatial scales, which require a relatively fine spatial resolution. Additionally, the fluid domain has to extend far enough beyond the projected area of the impacting structure and also to a sufficient depth to avoid the influence of reflected waves from the numerical boundaries on the interaction with the structure. The resulting number of fine particles required to model a ditching event would become prohibitively large for practical applications to study multiple ditching scenarios or for design purposes. Therefore, the development and enhancement of appropriate numerical simulation tools is required to overcome current limitations. Moreover, reliable experimental data are needed to allow for validation and to enhance the understanding of the involved physical phenomena.

II. Experimental Campaign

Water impact experiments of simple geometrical shapes at high horizontal velocity were widely conducted by researchers in the 1940s and 1950s, e.g. [7, 8, 28]. Available results, however, lack a reasonable time resolution and appropriate detail to support code development and essential validation. Therefore, a new experimental campaign of guided ditching tests was conducted, aiming to provide detailed experimental data, i. e. information about the structural loads and deformation as well as pressure distributions at suitable time resolution. Moreover, due to the number and complexity of the physical processes relevant for ditching (which cannot be scaled accurately in sub-scale experiments) full-scale structures at representative impact velocities are used.

A. *Test facility and setup*

Until recently, no existing water impact facility was capable of performing guided ditching tests at velocities representative of actual aircraft velocities during ditching. Therefore, within the FP7 research project SMAES [4], a new experimental facility has been designed, built, and installed over the towing tank #1 ($L{\times}B{\times}H = 470 \times 13.5 \times 6.5\,m$) at the CNR-INSEAN site in Rome, Italy.

The facility mainly consists of a guide track, a trolley holding the test specimen, and a catapult-type acceleration system (Fig. 1). The guide track is reinforced in the impact zone to minimize its structural deformations during the impact. For this test campaign, the structural panel design was derived from typical aeronautical skin panels. Tested panels measure $1000 \times 500\,mm$ and are riveted to an L-shaped frame in order to provide realistic boundary conditions. Table I provides an overview of test case dependent instrumentation.

B. *Test program and results*

Studied parameters of the test program comprising over 65 tests include impact velocity ($v_{x,0} = 30 - 46\,m/s$), pitch angle ($\alpha = 4 - 10°$), transverse shape of the structural panel (flat, convex, concave), panel thickness ($t = 0.8 - 15\,mm$), and material (aluminum, composite). So-called purely elastic test cases ($t = 15\,mm$) focus on the analysis of flow related results, such as pressure distributions, whereas elastic-plastic

TABLE I
OVERVIEW OF MEASUREMENTS BY ON-BOARD DATA ACQUISITION SYSTEM IN GUIDED DITCHING EXPERIMENTS

Measured value	**Sampling rate** [kHz]	**Amount of Signals** *Purely elastic*	*Elastic-plastic*
velocity	20	1	1
acceleration		6	6
force		4	4
strain		12	16
pressure	200	18	-
synchronization		2	2
time		1	1

test cases ($t = 0.8 - 3\,mm$) are conducted with the focus on the structural behavior under water pressure loading.

In order to minimize the amount of multiple repeats of each test condition, an adequate repeatability had to be proven, which was a major concern prior to testing. Therefore, two series of test-to-test dispersion experiments ($v_{x,0} = 40\,m/s$,$v_{z,0} = -1.5\,m/s, \alpha = 4$ and $10°$) with ten repeats each were performed. A high level of repeatability could be proven based on Pearson correlation coefficients while the highest repeatability was observed at $10°$ pitch angle. A followup analysis of the underwater high-speed recordings allowed to attribute this to entrapped air in the $4°$ tests which is responsible for larger oscillations in pressure results.

In this paper, experimental results are purely used to verify and validate the numerical modeling approach (see section V).

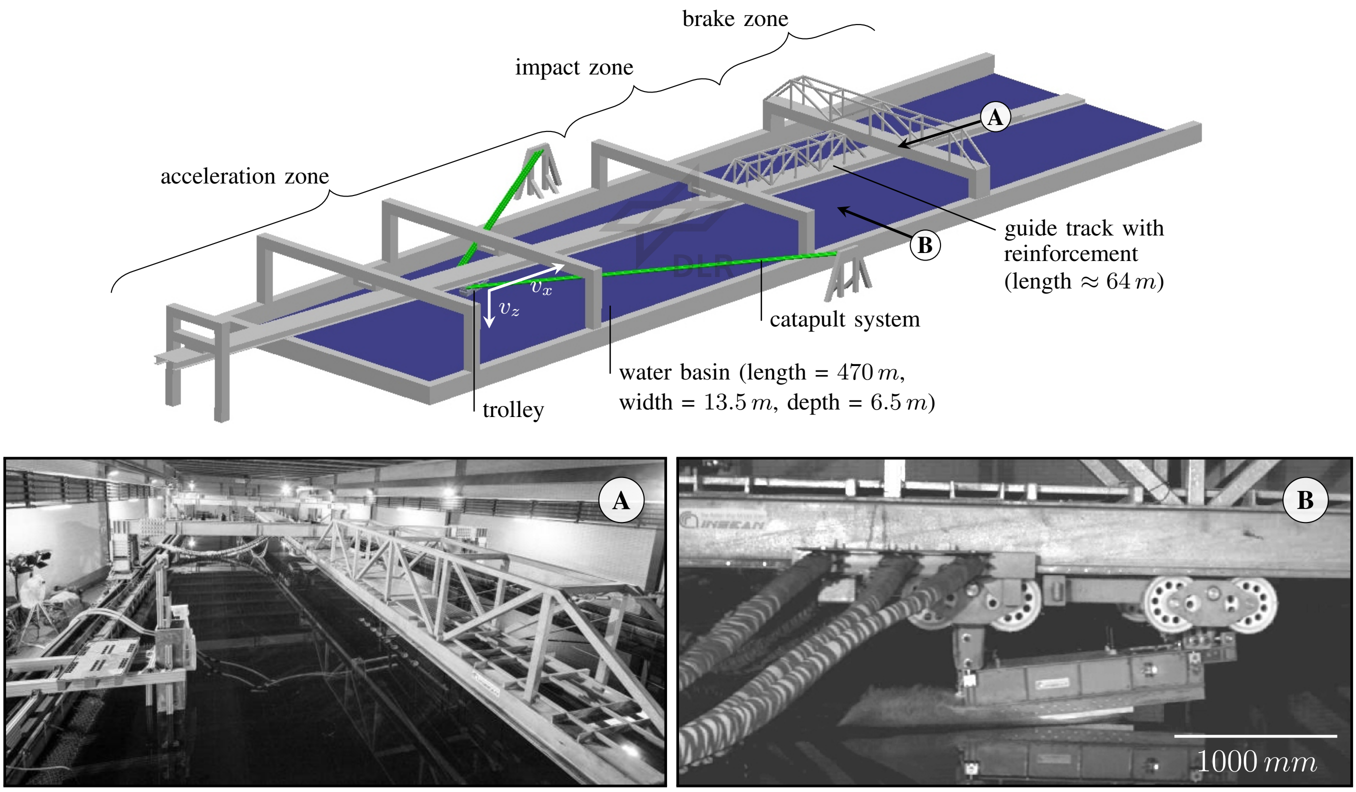


Fig. 1. Schematic overview of guided ditching experimental facility (top), photograph of impact zone seen from A (left) and observation during impact stage using an external high-speed camera seen from B (right).

For more detailed analysis of initial test results as well as further information on the experimental facility readers are referred to [16, 17, 18]. The evaluation of all experimental results is ongoing and findings are to be published in related journals as well as international conferences.

III. Numerical modeling approach

The violent nature of the considered water impact problem, i.e. oblique water impact or aircraft ditching, comes along with large deformations and complex free surface geometries. These characteristics cause mesh-based Lagrangian methods, e.g. FE method , to fail in accurately describing the fluid flow because they cannot handle such amounts of mesh distortion. Eulerian methods, on the other hand, may lead to numerical diffusion in case interfaces are transported across the faces of the finite volumes. Mesh-free particle methods such as Smoothed Particle Hydrodynamics (SPH) provide a powerful tool to overcome this handicap. Therefore, the fluid is modeled using SPH. The fluid equations to be solved are mass , momentum and energy conservation (eq. 1).

$$\frac{d\rho}{dt} = -\rho\nabla\cdot\mathbf{v} \qquad \frac{d\mathbf{v}}{dt} = -\frac{1}{\rho}\nabla p \qquad \frac{du}{dt} = -\frac{p}{\rho}\nabla\cdot\mathbf{v} \tag{1}$$

Note that the air phase is not explicitly modeled because it would significantly increase the computational effort while it is expected to have limited influence on results for the majority of regarded cases in this work. Nevertheless, when the angle between impacting structure and water surface is small, an air cushioning effect may affect results [11, 31].

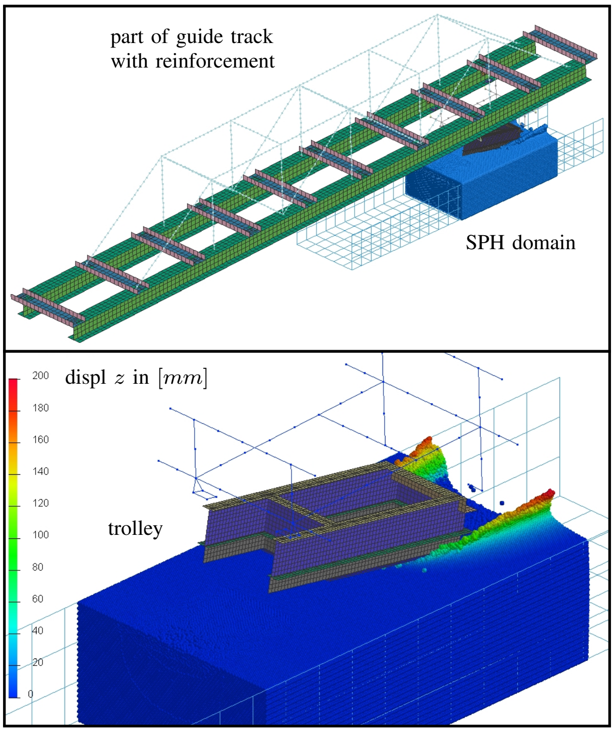


Fig. 2. Guided ditching simulation model (top) and zoom on trolley with SPH domain (bottom).

Numerical simulation of deformable structures, including potential failure, can appropriately be modeled by the Finite Element Method (FEM). The most relevant parts of the structure that get into contact with the water, both for the guided ditching tests and the actual ditching of an aircraft, are thin-walled, and shell elements provide a suitable solution. Advanced material models allowing for permanent deformation and failure are applied. The interaction between the finite elements and the particles for the fluid is modeled by penalty contact algorithms [12]. Note that in most cases involving flexible structures, the computational time step will be determined by the finite elements.

The chosen numerical approach couples SPH and FEM within one model; the fluid domain is represented by SPH and the structures are discretized using FEM. The developed guided ditching simulation model is shown in Fig. 2. Presented numerical work is based on using the commercial explicit finite element software package Virtual Performance Solution (VPS) with embedded SPH solver (ESI Group, www.esi-group.com).

IV. Smoothed Particle Hydrodynamics

A. General

Unless mentioned otherwise, the SPH discussed here is basically similar to that of Monaghan [10, 20, 21], including the anti-crossing option (XSPH), variable smoothing length and Monaghan-Gingold (M-G) artificial viscosity. To improve the pressure field the correction method proposed in [14] has been adopted because it showed superior pressure results in a study of two-dimensional oblique water impact [26]. Viscosity of the water could be included but it is not relevant for the considered application; yet, small amounts of M-G viscosity have been included to handle numerical stability. The Tait equation [29] will be used as equation of state for water:

$$p(\rho) \quad = \quad p_0 + \frac{c_0^2\rho_0}{\gamma}\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right] \tag{2}$$

with reference pressure p_0, speed of sound c_0 in the fluid at the state $\rho = \rho_0$, ratio of current over initial mass density ρ/ρ_0 and adiabatic exponent γ of the fluid ($\gamma = 7$ for water).

A quintic Wendland kernel function as in (3) with a range of $2h$ is applied; Macià et al. [19] found it to give superior results over the frequently used (renormalized) Gaussian kernel in free surface flow simulations.

$$W(q) = \begin{cases} \beta\cdot(2-q)^4\cdot(1+2q) & \text{for } 0 \le q \le 2 \\ 0 & \text{for } q > 2 \end{cases} \tag{3}$$

Symbols refer to $q = r/h$, r being the variable of the kernel function (radius), h the smoothing length and the dimension-dependent constant β controlling the normalization criterion (i.e. $\int W(q)dq = 1$).

B. Translating periodic boundary conditions

Modeling particles only where they are needed is general modeling practice to reduce the size of the fluid domain which is responsible for the majority of the computational cost. In the regarded cases, however, this becomes complicated by the presence of a high forward velocity which requires the fluid domain to translate with the impacting structure.

Translating periodic boundary conditions (TPBC's) as introduced in [13, 14] extend the well-known periodic boundary concept with the possibility to allow the boundaries to translate according to the motion of a given reference node. In that case, particles leaving the domain at the rear surface will be re-introduced at the front end without introducing any additional velocities into the system. For aircraft ditching the reference node should conveniently be defined as the center of gravity node of the aircraft.

In case the rear end of the domain is relatively close to the impacting structure, particles have not returned to their hydrostatic equilibrium position, as is obvious from Fig. 2. Since it is undesirable to re-enter the particles with the distribution and velocities in the wake of the impact, an option has been developed to reset all particle variables as well as the displacement in all directions, except for the translational direction, to the initial values upon re-entry.

Since for periodic boundaries the particles close to the rear end are to be considered as neighbors for the particles close enough to the front-end, the water in front of the structure may still be disturbed. In order to counteract this artifact, a new damping option has been implemented to linearly decrease the non-equilibrium displacements and velocities to their initial values (in most cases equal to zero) while the particles traverse over a part of the fluid domain having a length of a few times the smoothing length. A schematic illustration is given in Fig. 3. This domain is attached to the same reference node as used for the TPBC's. It has been verified that this feature allows to damp any remaining disturbances arising from the TPBC's to be used as quiescent water conditions for the impact during the entire ditching event.

To conclude, particles are only present where they are needed at any given moment, leading to a significant reduction in computational effort. In other words, the total number of particles necessary for a domain of sufficient size does no longer increase with longer simulation time. Therefore, possible savings increase with longer simulation time as this previously was the driver for the required size of the SPH domain. Savings are, for instance, in the range of a factor of 4 for the amount of particles (proportional to run time) for a guided ditching simulation over $200\,ms$ with an initial velocity of $40\,m/s$. In addition, storing and handling the results during post-processing is facilitated.

C. Spatially non-uniform initial particle distributions

An appropriate manner to further reduce the number of particles would be to use a fine enough resolution in the region immediately affected by the impact and a coarser distribution elsewhere. The possibility to adapt the spatial resolution to the phenomena of interest is an accepted approach for FEM and to finite volume solution methods, but has not become standard for SPH. Despite some approaches to generate spatially non-uniform initial particle distributions based on lattice structures [14, 23], analytical prescripts [32] as well as static and dynamic refinement techniques [1, 9], there did not yet exist a generally accepted method to distribute particles of non-uniform size in $3D$ domains with arbitrary geometry.

Due to its nature, the SPH method does not provide accurate results when neighboring particles vary too much in size, i.e. smoothing length. Therefore, spatially non-uniform initial distributions created based on geometrical principles may become unstable under simple gravity load [15]. An efficient methodology to generate three-dimensional, initial non-uniform particle distributions is the weighted Voronoi tessellation (WVT) technique proposed by Diehl et al. who conducted a comprehensive review of different initial particle distributions with the aim to find optimal conditions for SPH simulations [6]. The iteration takes only a few hundred cycles and is fast because it is not necessary to solve the fluid equations (1). Groenenboom extended the original iteration technique allowing for filling $3D$ domains with boundaries of arbitrary shape [15]. As discussed in that paper, a smooth variation of the smoothing length and particle size does not

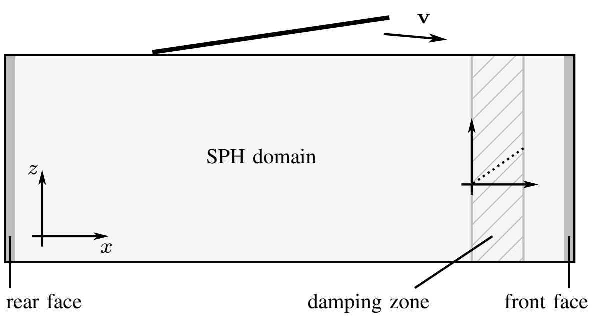


Fig. 3. Schematic view on translating periodic boundary condition with special damping zone.

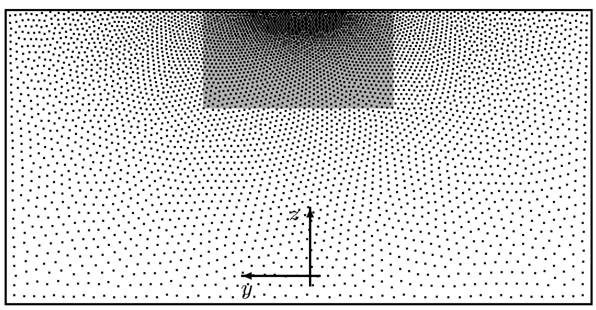


Fig. 4. Spatially non-uniform initial particle distribution generated using extended WVT in $2D$ starting from a simple rectangular patch of uniformly spaced particles occupying only $1/9$ (shaded) of the final volume. The ratio of maximum to minimum smoothing length is $h_{max}/h_{min} = 5$.

provide a sufficient condition to guarantee that the resulting particle distribution is stable for SPH simulation when gravity is applied. The cause of such non-equilibrium configurations has been found to be the anisotropic distribution of neighbor particles that may occur in such distributions [15]. One means to assess the amount of local anisotropy is to regard the ratio of minimum and maximum eigenvalues of the SPH inertia tensor at any particle at the interior of the domain. An ideal isotropic setup shall give a value of unity whereas lower values imply higher anisotropy. It has been demonstrated that the WVT procedure converges towards a state in which the inertia tensor is a multiple of the unit tensor representative for an isotropic distribution [15]. The resulting particle distribution is then stable when loaded by gravity.

The advantage of the WVT is that it is a fast and simple method to fill three-dimensional domains starting from a simple patch of uniformly spaced particles which reduces excessively the pre-processing effort. The quality of the distribution can be judged by inspection of the smoothing length and, more importantly, the ratio of minimum and maximum eigenvalues of the SPH inertia tensor. An exemplary non-uniform distribution in $2D$ generated using the presented approach is shown in Fig. 4.

V. Numerical Model Verification and Validation

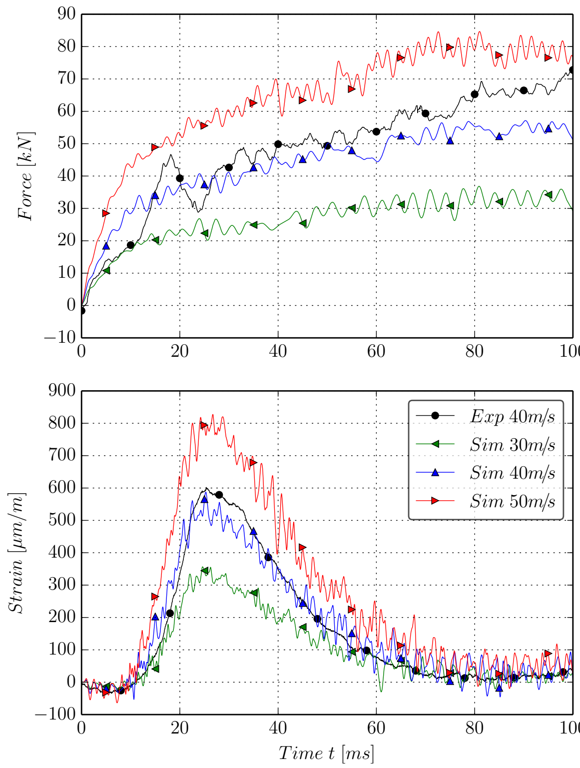
This section provides insight into the validation progress of the simulation model of guided ditching tests shown in Fig. 2 based upon results of the experimental campaign (section II).

We regard initially undeformed aluminum panels which are inclined at an angle α in regard to the water surface. All panels measure $1000 \times 500\,mm$ in plane and their thickness is $15\,mm$ for purely elastic cases and 3.0 or $0.8\,mm$ for elastic-plastic cases.

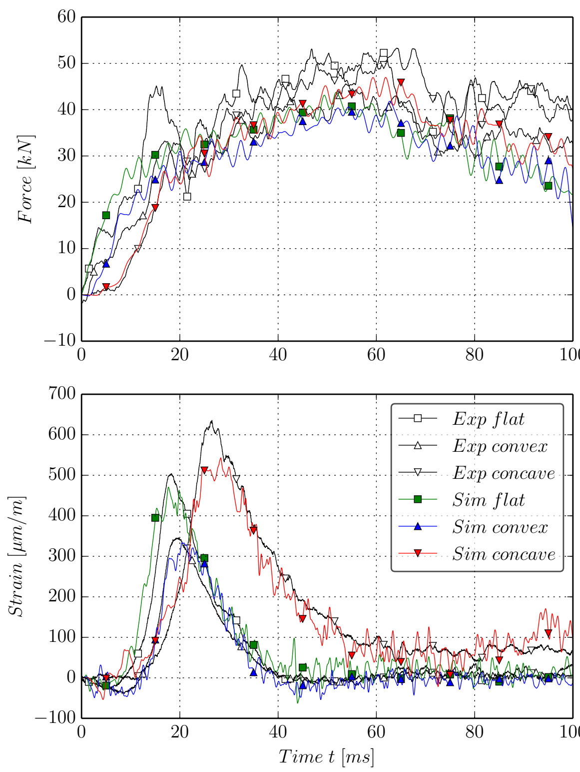
A. *Purely elastic panels*

Qualitative comparison of the flow kinematics (bow wave formation and shape) show satisfactory correlation between experimental underwater images and corresponding contour plots of the numerical simulation. This is exemplary shown in [27] for test cases using different panel shapes (flat, concave, convex). The bow wave is curved positive or negative depending on the panel shape as it may be anticipated. This three-dimensional nature of the flow is stronger for larger pitch angles because the fluid is less constrained underneath the impacting structure. In the experiment a stripe of bubbles was observed in the center line of the concave panel which points at air being entrained due to the shape of the panel. This, however, cannot be covered by the numerical model at this time as it does not account for air.

Quantitative comparison in Fig. 5 shows very good correlation both for force and especially for strain results. Forces in the computational model are extracted using numerical load cells at the same positions as in the experiment. Summed maximum force values in z-direction of the panel increase considerably with the regarded impact velocities. Strain results show a similar increase with the impact velocity but they



(a) Variation of impact velocity of flat panel at 10° pitch angle.



(b) Variation of panel shape at 6° pitch angle and $40\,m/s$ impact velocity.

Fig. 5. Summed force-time history of load cells in z-direction of the panel and strain-time history for gauge in x-direction on the center line of the panel. Purely elastic test cases using $15\,mm$ thick aluminum panel with variation of impact velocities (a) and panel shape (b).

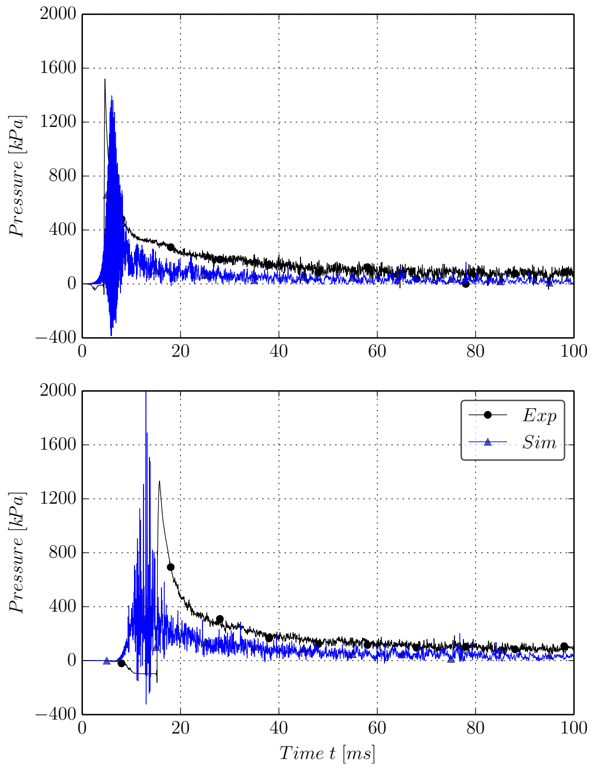


Fig. 6. Comparison of experimental and numerical pressure-time histories at two positions along the center line: $x = 125\,mm$ (top) and $x = 250\,mm$ (bottom). Test case with flat $15\,mm$ thick aluminum panel at $6°$ pitch angle and $40\,m/s$ impact velocity.

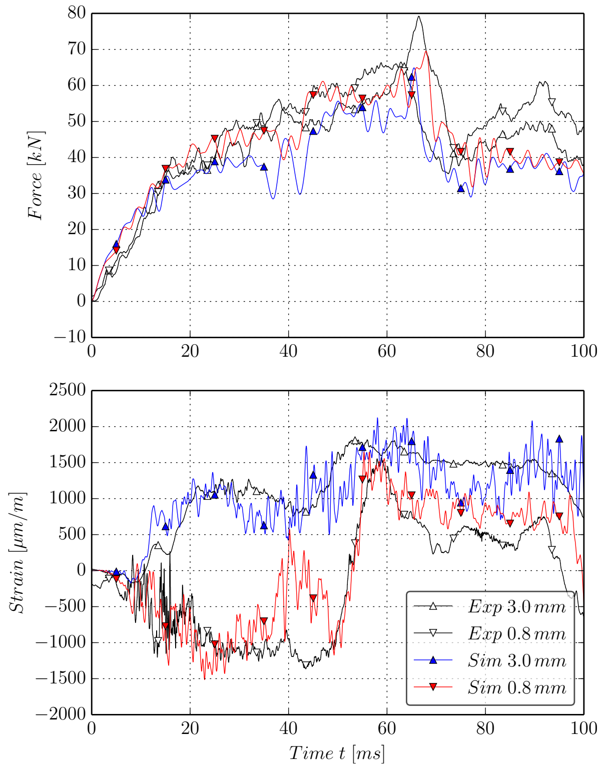


Fig. 7. Experimental and numerical results of elastic-plastic test cases using thin flat aluminum panels at $40\,m/s$ impact velocity and $6°$ pitch angle. Summed force-time history of load cells in z-direction of the panel (top) and strain-time history for gauge in y-direction on the panel center line (bottom).

remain still well within the purely elastic range. As anticipated, Fig. 5 (bottom) shows that loads and strains are slightly higher for concave cases compared to convex cases due to the additional constraint the fluid is facing.

Despite very good agreement of force and strain results, numerical pressure-time histories are still suffering from considerable oscillations (Fig. 6). These high-frequency vibrations, however, are not relevant for the structure which in the numerical simulation is loaded just as in the experiment (shown in Fig. 5). Nevertheless, in a simplified two-dimensional simulation model the sharp pressure gradient could be captured by further refinement of the particle distribution (not shown here). It is intended to improve pressure results of the guided ditching simulation by application of appropriately refined non-uniform particle distributions in the proximity of the panel as discussed in section IV.

B. Elastic-plastic panels

After numerical results using purely elastic panels showed good correlation with experimental ones, the next step was to correctly simulate the behavior of elastic-plastic aluminum panels. New challenges arise as, for instance, correctly reproducing deformations requires a finer discretization of the structure which reduces the critical time step of the simulation. This emphasizes once more the necessity of efficient numerical modeling.

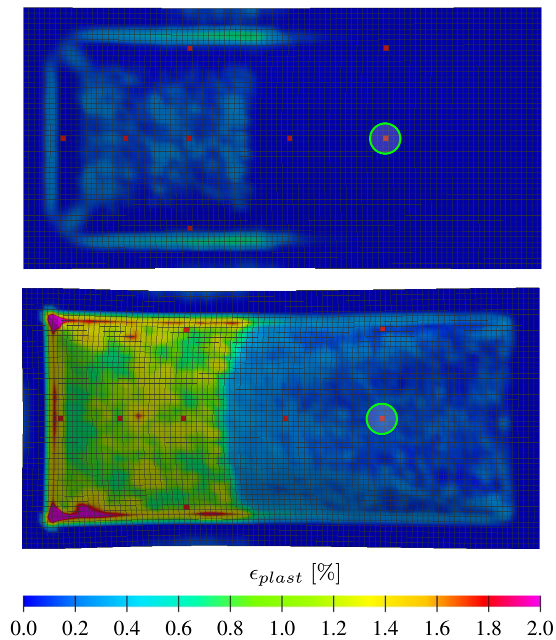


Fig. 8. Contour plots of maximum plastic strain over thickness at $t \approx 30\,ms$ for elastic-plastic test cases using $3.0\,mm$ (top) and $0.8\,mm$ (bottom) thin aluminum panels at $40\,m/s$ horizontal impact velocity and $6°$ pitch angle. Strain gauge positions used for above time history plot are encircled in green color; others are highlighted in red color.

Fig. 7 illustrates experimental as well as numerical force- and strain-time histories of elastic-plastic test cases using panel thicknesses of 0.8 and $3.0\,mm$. In comparison to purely elastic cases, loads increase with reduced panel thickness because of the inherent increasing deformation into a concave shape (regarding the studied thickness range and recalling that rupture occurred neither in experiments nor in simulations). Measured strain values are in the elastic regime; however, experiments as well as simulations show permanent deformation. The apparent discrepancy is due to the fact that highest strains are found in the proximity of the inner edge of the frame which holds the panels. This could be justified by analyzing strain contour plots of the simulations as shown in Fig. 8. The high strain zones are very local and do not coincide with the strain gauge positions in the experiments.

Overall, force and strain results compare well between experiments and simulations which allows validating the chosen simulation approach for purely elastic as well as elastic-plastic metallic panels.

VI. Conclusion

This work emphasizes the necessity as well as the requirements for advanced numerical tools to simulate aircraft ditching especially when highly non-linear structural behavior accompanied by potential panel rupture shall be captured.

A brief introduction to the experimental campaign of guided ditching tests with references to more detailed literature was given. Achieved full-scale experimental results are of high quality and, therefore, provide insight into the complex involved physical phenomena as it was not previously possible. Available data offer a novel test case to be used for verification of numerical codes.

Next, a hybrid modeling approach utilizing SPH to represent the fluid and FE to model structures was presented. Most recent enhancements of the SPH method applied to the guided ditching simulation model were discussed. Guided ditching experiments could successfully be modeled using this coupled SPH-FE approach wherein the enhancements of the SPH modeling are key to efficiency. Comparison of strain and force results for purely elastic as well as elastic-plastic test cases of aluminum panels showed good agreement. However, numerical pressure results in the proximity of the impacting structure are still challenging to capture due to the extremely small temporal and spacial scales of present pressure peaks. Yet, it is reassuring that even when the pressure results do not match well due to the oscillations, the force and strain transmitted to the structure may very well be correct and this is what is relevant for the design and certification process.

Future work will address spatially non-uniform initial particle distributions with the aim of (1) further reduction of computational effort and (2) improvement of pressure results. Therein, the ease of use is an important objective. Moreover, experimental and numerical results of composite panels are to be compared in order to verify the simulation model. In a next step, the inclusion of a model for the suction effect and other hydrodynamic phenomena will be pursued. The long term objective is a full-scale deformable aircraft ditching simulation.

Acknowledgment

Most of the work leading to the results presented in this paper has received funding from the European Commission's 7th Framework Programme under grant agreement n° FP7-266172 and was performed within the project SMAES - SMart Aircraft in Emergency Situations.

The authors would like to thank especially Dr. Alessandro Iafrati and his team at CNR-INSEAN (Rome, Italy) for their excellent work with the experimental campaign of guided ditching tests.

References

[1] D.A. Barcarolo, *Improvement of the Precision and the Efficiency of the SPH Method: Theoretical and Numerical Study*, PhD thesis, École Centrale de Nantes, 2013.

[2] L. Benítez Montañés, H. Climent Máñez, M. Siemann and D. Kohlgrüber, *Ditching Numerical Simulations: Recent Steps in Industrial Applications*, in Proc. of Aerospace Structural Impact Dynamics International Conference, Wichita, USA, 2012.

[3] J. Campbell, *Prediction of aircraft structural response during ditching: An overview of the SMAES project*, in Proc. of Aerospace Structural Impact Dynamics International Conference, Wichita, USA, 2012.

[4] J.C. Campbell and R. Vignjevic, *Simulating Structural Response to Water Impact*, International Journal of Impact Engineering 49(2012), pp.1–10.

[5] H. Climent Máñez, L. Benítez Montañés, F. Roisch, F. Rueda and N. Pentecôte, *Aircraft Ditching Numerical Simulation*, in Proc. of 25th ICAS, Hamburg, Germany, 2006.

[6] S. Diehl, G. Rockefeller, C. L. Fryer, D. Riethmiller and T. S. Statler, *Generating Optimal Initial Conditions for Smooth Particle Hydrodynamics Simulations*, Preprint submitted to Monthly Notices of the Royal Astronomical Society, 2012.

[7] P.M. Edge Jr., *Impact-Loads Investigation of Chine-Immersed Model having a Circular-Arc Transverse Shape*, NACA Technical Note 4103, Langley Field, Virginia, USA, 1957.

[8] P.M. Edge Jr., *Hydrodynamic Impact Loads of a -20 Dead-Rise Inverted-V Model and Comparisons with Loads of a Flat-Bottom Model*, NACA Technical Note 4339, Langley Field, Virginia, USA, 1958.

[9] J. Feldman, *Dynamic refinement and boundary contact forces in Smoothed Particle Hydrodynamics with applications in fluid flow problems*, PhD thesis, University of Wales Swansea, 2006.

[10] R. A. Gingold and J. J. Monaghan, *Smoothed particle hydrodynamics - Theory and application to non-spherical stars*, Monthly Notices of the Royal Astronomical Society 181(1977), pp.375-389.

[11] P. Groenenboom, *Lifeboat water entry simulation by the hybrid SPH-FE method*, in Proc. of 3rd ERCOFTAC SPHERIC workshop on SPH applications, Lausanne, Switzerland, 2008.

[12] P.H.L. Groenenboom and B.K. Cartwright, *Interaction between structures and waves using the coupled FE-SPH method* in Proc. of International Conference on Computational Methods in Marine Engineering, MARINE 2007, Barcelona, Spain, 2007.

[13] P. Groenenboom and B. Cartwright, *SPH simulations of free surface waves and the interaction with objects*, in Proc. of 5th European Conference on Computational Fluid Dynamics, Lisbon, Portugal, 2010.

[14] P. Groenenboom and B. Cartwright, *Pressure-corrected SPH with innovative particle regularization algorithms and non-uniform, initial particle distributions*, in Proc. of 8th International SPHERIC Workshop, Trondheim, Norway, 2013.

[15] P.H.L. Groenenboom, *Particle Filling and the Importance of the SPH Inertia Tensor*, in Proc. of 9th International SPHERIC Workshop, Paris, France, 2014.

[16] A. Iafrati and D. Calcagni, *Numerical and experimental studies of plate ditching*, Abstract in Proc. of 28th IWWWFB, L'Isle sur la Sorgue, France, 2013.

[17] A. Iafrati, M.H. Siemann and L. Benítez Montañés, *Experimental study of high speed plate ditching*, Abstract in Proc. of 29th

IWWWFB, Osaka, Japan, 2014. http://iwwwfb2014.naoe.eng.osaka-u.ac.jp/accept/iwwwfb29_22.pdf.

[18] A. Iafrati, M.H. Siemann and L. Benítez Montañés, *Experimental analysis of the water entry of a plate at high horizontal speed*, submitted to 30th Symposium on Naval Hydrodynamics, Hobart, Australia, 2014.

[19] F. Macià, A. Souto-Iglesias, M. Antuono and A. Colagrossi, *Benefits of using a Wendland kernel for free-surface flows*, in Proc. of 6th International SPHERIC Workshop, Hamburg, Germany, 2011.

[20] J.J. Monaghan, *Smoothed Particle Hydrodynamics*, Annu. Rev. Astron. Astrophys. 30(1992), pp.543–574.

[21] J.J. Monaghan, *Simulating Free Surface Flows with SPH*, Journal of Comp. Phys. 110(1994), pp.399–406.

[22] NTSB, *Loss of Thrust in Both Engines after Encountering a Flock of Birds and Subsequent Ditching on the Hudson River: US Airways Flight 1549*, NTSB Technical Report, Weehawken, New Jersey, USA, 2010.

[23] G. Oger, M. Doring, B. Alessandrini and P. Ferrant, *Two-dimensional SPH simulations of wedge water entries*, Journal of Computational Physics 213 (2006), pp.803–822.

[24] M. Pilorget, *Development of a dynamic calculation tool for simulation of ditching*, MSc thesis, SUPAERO - École nationale supérieure de l'aéronautique et de l'espace, 2010.

[25] M.V. Reinhard, *Free Elastic Plate Impact into Water*, PhD thesis, University of East Anglia, 2013.

[26] M. Siemann and P. Groenenboom, *Pressure Evaluation at Arbitrary Locations in SPH Water Impact Simulations*, in Proc. of 3rd International Conference on Particle-based Methods, Stuttgart, Germany, 2013.

[27] M.H. Siemann,D. Kohlgrüber,L. Benítez Montañés and A. Iafrati, *Numerical Simulation and Experimental Validation of Guided Ditching Tests*, in Proc. of 11th WCCM/5th ECCM, Barcelona, Spain, 2014.

[28] R.F. Smiley, *An experimental study of water-pressure distributions during landings and planing of a heavily loaded rectangular flat-plate model*, NACA Technical Note 2453, Langley Field, Virginia, USA, 1951.

[29] P.G. Tait, *Report on some of the physical properties of fresh water and sea water*, Physics and Chemistry 2 (1888), pp.1–76.

[30] A. Tassin, *Modelisation tridimensionnelle d'impacts hydrodynamiques pour l'etude du tossage des bulbes d'etrave*, PhD thesis, École Doctorale SICMA, 2010.

[31] N. Toso, *Contribution to the Modelling and Simulation of Aircraft Structures Impacting on Water*, PhD thesis, Institute of Aircraft Design, University of Stuttgart, 2009.

[32] C. Ulrich and T. Rung, *SPH Modelling of Propeller Induced Harbour-Bed Erosion by a Container Vessel*, in Proc. of 7th International SPHERIC Workshop, Prato, Italy, 2012.