

INNOVATIVE SPH METHODS FOR AIRCRAFT DITCHING

PAUL H.L. GROENENBOOM¹, JAMES CAMPBELL², LUIS BENÍTEZ
MONTAÑÉS³ AND MARTIN SIEMANN⁴

¹ ESI-Group, Delft, Radex Innovation Centre, Rotterdamseweg 183 C, 2629 HD Delft,
The Netherlands. pgr@esi-group.com, www.esi-group.com

² School of Engineering, Cranfield University, Bedfordshire, MK43 0AL, UK.
j.campbell@cranfield.ac.uk, www.cranfield.ac.uk

³ AIRBUS MILITARY - Structural Dynamics and Aeroelasticity Dept., P^o John Lennon, s/n,
28906 Getafe, Spain. Luis.Benitez@military.airbus.com, www.airbusmilitary.com

⁴ German Aerospace Center (DLR) - Institute of Structures and Design, Pfaffenwaldring 38-40, 70569
Stuttgart, Germany. martin.siemann@dlr.de, www.dlr.de/bt/en

Key Words: *Fluid-structure interaction, Smoothed Particle Hydrodynamics, Aircraft Ditching, Hydrodynamic Phenomena.*

Abstract. Smoothed Particle Hydrodynamics (SPH) has been used to simulate the complex fluid flow phenomena involved in ditching of aircraft. Innovations in the SPH method that allow simulating the interaction of structures impinging on water with increased accuracy and enhanced computer performance will be presented. Interaction between the structure and the fluid is treated by coupling finite elements for the structure with SPH for water. The effectiveness of this approach will be demonstrated for vertical wedge impact and ditching of a subscale aircraft model.

1 INTRODUCTION

Aircraft ditching is an emergency situation that needs to be studied during aircraft design and certification phases. The physical phenomena involved in ditching are highly complex and the use of simulation tools is becoming a current practice in research and industry fields [1-4]. The capability of simulating the aircraft structure by means of non-linear Finite Element (FE) Method coupled with water modelled with SPH (Smoothed Particle Hydrodynamics) methods has received attention in recent years. This paper presents recent advances in SPH methods for modelling water media that provide significant advances in ditching simulation in terms of pressure fields acting on the structure and kinematic behaviour of the aircraft at the impact. The high velocity creates a local, but very pronounced pressure peak which generates waves. Another effect which is aggravated by the high horizontal velocity is the break-up of the free surface and generation of spray. Effects due to aerated water and cavitation may also need to be considered. There exists also a tendency of the water

flow to remain attached to the aircraft skin after initial contact; this suction effect is known to strongly affect the aircraft kinematics. For simulation of the entire event the computational domain has to be quite long which makes numerical simulation computationally demanding.

Since it is not feasible to conduct ditching tests at full scale under experimentally controlled conditions, the only tests that can be done, and have been done, are scaled tests [3, 5]. The fluid flow phenomena mentioned above make it, however, impossible to define a consistent set of scaling rules [6]. Moreover, due to the high loads the structural flexibility of the fuselage will have to be accounted for and the subsequent hydro-elasticity cannot be captured with classical reduced size experimental models. Thus numerical simulation of real size aircraft models is relevant for the aeronautical industry once the methodology is well established. The combination of ditching simulations at full scale using numerical methods validated against scaled experiments with the experimental results at representative scales and velocities for skin panels [4] provides a unique capability for future aircraft design able to withstand relevant ditching scenarios. Although significant progress has been made in analytical and semi-analytical approaches for ditching [7], such methods are unable to capture all physical complexities of ditching which warrants the application of advanced numerical tools.

The mentioned complexities are very difficult to capture at sufficient accuracy using mesh-based methods, hence the mesh-less Smoothed Particle Hydrodynamics (SPH) method is proposed here for the fluid flow. Due to the Lagrangian nature of SPH, there are no problems to track the deforming free surface or other interfaces. In order to reduce complexity and computational cost the air is neglected in the model, with the region above the water treated as void. Simplified lift/drag models can be included to model the overall aerodynamic loads on the aircraft if deemed relevant

The requirements for the numerical method to describe the aircraft during ditching are that the structure is accurately represented, that the constitutive models for the material include elastic and plastic deformation with strain-hardening, strain-rate effects and allow for damage and failure. Such requirements are best met by explicit finite element (FE) codes such as used for crashworthiness studies. In addition, it should be possible to model the interaction with the fluid represented by particles. The FE software tools discussed are the commercially available VPS/PAM-CRASH code and the 3D SPH solver developed at Cranfield that is coupled with the LLNL-DYNA3D FE code [8].

In the following two sections a brief overview of the SPH method including fluid-structure interaction coupling and the innovations in the computational approach will be presented. Section 4 presents the models for two simplified test cases and selected results. Finally, some conclusions will be drawn.

2 COMPUTATIONAL APPROACH – STANDARD FORMULATION

This section provides a brief description the Weakly Compressible Smoothed Particle Hydrodynamics (WC-SPH) method and the numerical coupling with structural parts as by the software tools employed in these studies.

2.1 Smoothed Particle Hydrodynamics

The SPH method is an interpolation method in which each “particle” describes a fixed amount of material moving in a Lagrangian frame of reference. SPH is a solution method for the continuum mechanics equations of fluids and solid materials. Since the method requires no mesh, SPH facilitates the simple handling of the motion and topology changes of material surfaces. As shown in figure 1, the evaluation of relevant field variables such as density and velocity at the location of a selected particle is conducted by interpolation over all neighbour particles in a region of influence. The size of the sphere (or circle in 2D) of this region is defined by a smoothing length (h). This allows the spatial derivatives of field variables such as velocity and pressure to be evaluated using the known derivative of the smoothing kernel.

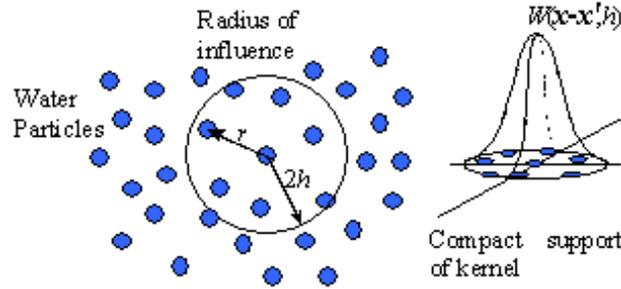


Figure 1: Two-dimensional representation of the region of influence and the smoothing kernel.

The continuity equation provides the time derivative of the density at particle i :

$$\frac{d\rho_i}{dt} = \sum_j m_j (\bar{v}_i - \bar{v}_j) \cdot \nabla_i W_{ij} \quad (1)$$

where the sum extends over all neighbour particles j , each having a mass of m_j . The vectors \bar{v}_i and \bar{v}_j are the velocities of the i^{th} and j^{th} particles, respectively, and W_{ij} is the smoothing kernel evaluated at the distance between the i^{th} and j^{th} particles. The updated velocity may be obtained from the momentum equation:

$$\frac{d\bar{v}_i}{dt} = -\sum_j m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Pi_{ij} \right) \cdot \nabla_i W_{ij} \quad (2)$$

where p_i and p_j represent the particle pressures. The artificial viscosity is defined as:

$$\Pi_{ij} = \frac{2}{\rho_i + \rho_j} \left(-\alpha \frac{c_i + c_j}{2} \mu_{ij} + \beta \mu_{ij}^2 \right), \quad (3)$$

with c_i and c_j being the local sound speed, and with

$$\mu_{ij} = \frac{1}{2}(h_i + h_j) \frac{(\bar{v}_i - \bar{v}_j) \cdot (\bar{r}_i - \bar{r}_j)}{|\bar{r}_i - \bar{r}_j|^2 + \epsilon^2 h^2} \quad \text{when } (\bar{v}_i - \bar{v}_j) \cdot (\bar{r}_i - \bar{r}_j) < 0 \quad (4)$$

and zero otherwise; h_i and h_j are the smoothing lengths. The parameters α and β determine the strength of the artificial viscosity required to suppress numerical shocks. These artificial viscosity parameters should be set as low as possible to avoid the flow becoming too viscous.

The smoothing kernel W is a bell-shaped function of the distance between particles for which in most simulations the cubic spline or the Wendland kernel [4] will be used.

The flow is assumed to be nearly incompressible implying that the pressure field is obtained from an equation of state (EOS) model. A polynomial EOS may be used for water, but it is convenient to use an alternative: the Tait model [4], whose EOS is given by:

$$p = {}^0p \left[\left(\frac{\rho}{{}^0\rho} \right)^\gamma - 1 \right] \quad (5)$$

where 0p is the reference pressure, ${}^0\rho$ the reference density and γ the adiabatic exponent which is usually set to 7.0 for water. A cut-off pressure may be included as a simple approximation to cavitation.

Particles are assumed to interact mutually only if they are sufficiently close to each other. This is established by a nearest neighbour (NN) search. Second-order accurate leap-frog time stepping is used for the explicit time integration of equations 1 and 2. The numerical time step is set at a fraction of the well-known Courant (or CFL) criterion based on the current size and sound speed of all particles and finite elements present in the model.

Despite obvious advantages of SPH to simulate fluid flow involving free surfaces or interaction with a structure or other fluids, there also exist a few drawbacks of the standard SPH method. These include the occurrence of irregular particle distributions ('clumping'), the scatter in computed pressures, and limitations in spatially varying SPH solutions. These limitations have been largely overcome by correction terms in the SPH formulations as discussed [9, 13] and in section 3 below.

2.2 Coupling with structures

Numerical simulation of structural mechanics is conveniently conducted using the Finite Element Method (FEM). The type of elements may include shells, solid elements, beams and bars. Since the interaction of the structure with the fluids considered here is highly dynamic, an explicit solution for the structural dynamics should be considered.

One of the FEM codes involved is VPS (of which the explicit part dedicated to crashworthiness is also known as PAM-CRASH), with its embedded SPH solver. Interaction between particles, representing a fluid, and finite elements, representing moving or deformable structures, may be modelled by one of the sliding interface contact algorithms available in VPS [10]. The sliding interfaces employed are based on the well-known penalty formulation, where geometrical interpenetrations between slave nodes and corresponding master faces are penalized by counteracting forces proportional to the penetration depth. The contact algorithm detects when a particle penetrates a master segments representing the outer surface of the finite element model of the structure. This type of contact has been validated by simulating the vertical motion of floating bodies.

An alternative contact treatment between the FE mesh and SPH particles is to treat the FE nodes as SPH particles within a particle-particle contact algorithm. In this approach an SPH particle interacts with all FE nodes within its neighbourhood and a repulsive force, based on a contact potential is generated in between the particle and each node. This approach has been

successfully applied to a range of water-structure interaction problems including the behaviour of floating bodies and impact of deformable structure on water [8]. When analysing a deformable structure, a contact based on a sliding interface requires the SPH resolution to be higher than the FE resolution ensuring that all contact segments that should be in contact with the fluid are interacting with SPH particles. By contrast for the particle-particle approach the SPH resolution should not be higher than the FE resolution and can be lower.

3 COMPUTATIONAL APPROACH – INNOVATIONS

This section describes the innovations which have been employed and further developed within the context of the SMAES project.

3.1 Pressure correction

One of the well-known drawbacks of the WC-SPH solution as discussed above is the rather large variation in pressure, both in time and space. A modified equation for the new density may be proposed as [9]:

$$\frac{\rho^{n+1} - \rho^n}{\Delta t} = \sum_j m_j \cdot (\bar{u}_i^n - \bar{u}_j^n) \cdot \nabla_i W_{ij} - 2 \cdot \xi \sum_j \frac{m_j}{\rho_j} \frac{(P_i^n - P_j^n) \cdot \bar{r}_{ij} \cdot \nabla_i W_{ij} \cdot \Delta t}{(r_{ij}^2 + \epsilon \cdot h^2)} \quad (6)$$

in which the superscript refers to the time step, $\bar{r}_{ij} = \bar{r}_i - \bar{r}_j$ and Δt is the time step.

This pressure correction provides a significantly smoother pressure distribution than the reference SPH simulation but without notable effects on the free surface location and velocities. The total fluid volume (or average density) is also not modified.

An additional approach to improving the accuracy of the SPH method for flow simulations is the usage of regularization methods. Such methods mitigate clumping and generation of other irregular particle distributions that are known to deteriorate the accuracy of the solution during the flow simulation. Various algorithms have been investigated [9] and have been shown to deliver more accurate flow simulations while also reducing the strength of the artificial viscosity required.

3.2 Boundary conditions and damping

In numerical simulation of aircraft ditching the computational domain for the water is usually limited by rigid wall boundaries. Reflection of waves emanating from the impact region may adversely influence the solution if these boundaries are defined too close to the impact region. One possibility to reduce the computational effort is to employ finite elements for the (exterior) parts of the water in which there is only limited flow. For ditching where the high horizontal velocity necessitates an extended SPH domain, an appropriate condition for further reduction of computational effort is that of periodic boundaries [4]. This feature allows particles leaving one end of the domain to enter at the opposite end with the same velocity. Opposing boundaries may be allowed to translate according to the (horizontal) motion of the ditching aircraft, without introducing additional velocities to the particles themselves. To avoid that free surface disturbances in the wake of the aircraft will modify the incoming flow,

it is also possible to enter the particles at their initial equilibrium conditions. Due to the interaction between nearest neighbour particles that extends on both sides of a periodic boundary, disturbances from the initial conditions at the inflow end may still arise in case the wake at the opposite end is violent. This is accounted for by a special damping zone as explained below.

A simple method to reduce the reflection of free surface waves from a rigid wall boundary or for transmission at a periodic boundary is by defining damping zones in front of these boundaries [9]. To avoid a sudden jump in damping introducing a discontinuity, giving rise to artificial reflections, an option to introduce a smooth transition to full damping has been implemented. This method allows to damp free surface waves effectively, but may not be adequate to let particles in the spray created by the ditched aircraft to settle down to act as an appropriate inflow distribution. For that case a new type of damping has been introduced in which particles traversing a section with a length of at least two h are gradually returned to their initial positions.

Figure 2 illustrates how the combined features of the periodic boundary condition and the new damping zone may be used to significantly reduce the size of the domain to be filled with small particles for the ditching impact of a deformable plate. Many particles have left the domain on the ‘downstream’ side but despite the huge amount of spray-like distortion and the reduction of free surface level directly behind the plate, no disturbance is observable in the particles that have re-entered at the front face. With the domain translating at the same instantaneous horizontal velocity as the plate (for instance linked by one reference node), the distance from the plate to the domain boundaries remains constant throughout the simulation.

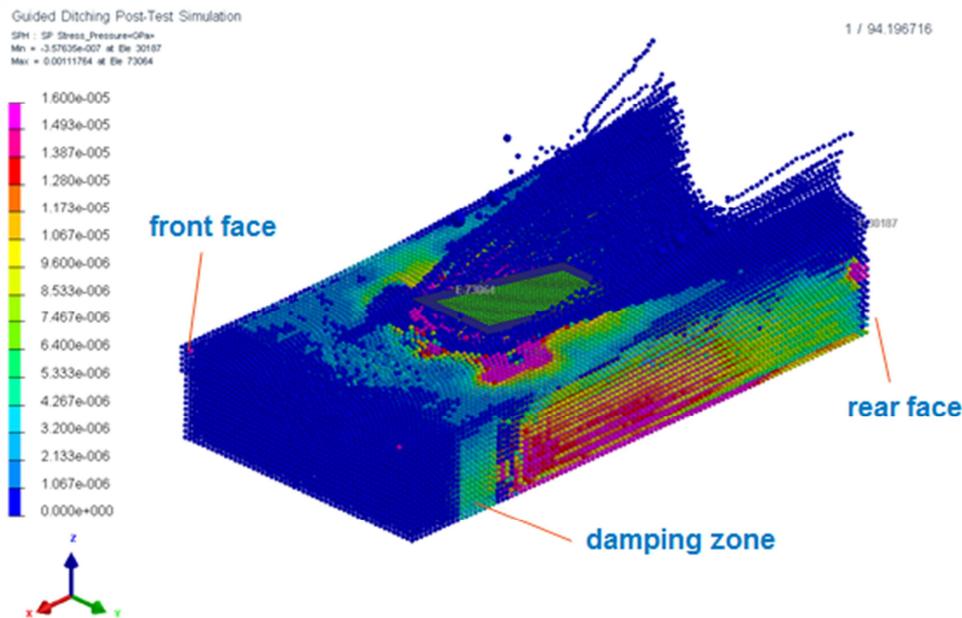


Figure 2: Example of particle distribution with pressures for plate impact demonstrating the usage of periodic boundaries and the damping zone.

3.3 Non-uniform particle distributions

A shortcoming of the standard SPH approach is the difficulty to employ a non-uniform volume discretization. For flow simulations such a feature allows using a fine discretization for the regions where this is relevant, as for instance where there is contact with structures. The difficulty is related to the requirement that the particle distribution should locally be sufficiently uniform as well as isotropic and without sudden discontinuities in particle size. If, for example, a rectangular domain is filled with particles placed at equal distance from each other in horizontal directions, but at an increasing distance in vertical (downward) direction, the bigger particles near the bottom may find more neighbors in horizontal direction than in vertical direction. As may be anticipated [11], such a distribution may give rise to unphysical flow phenomena.

To overcome this problem, the Weighted Voronoi Tessellation (WVT) method originally proposed by Diehl et al. [12] has been extended and implemented in VPS/PAM-CRASH. With Voronoi tessellation a set of points (seeds) is specified beforehand and for each seed there will be a corresponding region consisting of all points closer to that seed than to any other. Diehl et al. argue that the Voronoi tessellation can be extended to spatially adaptive (non-uniform) particles and that placing the SPH particles at the centroid of the polygons produces the optimal configuration. The WVT technique they propose is to start from an arbitrary particle configuration, impose displacements based upon the distance to its nearest neighbour, and iterate towards a converged distribution. The extensions include an option to allow particles to grow in size and to account for boundaries. The latter allows filling a domain of arbitrary size and shape with particles starting from a reference distribution somewhere in the interior which may simplify the generation significantly [11].

To judge the quality of the resulting non-uniform particle distribution other than by visual inspection, it is proposed to assess the distribution of the smoothing length and the ratio of anisotropy. For a particle distribution that remains stable under hydrostatic conditions, one could consider each particle ‘i’ to be the center of a body containing all its neighbors and the following inertia tensor may be proposed:

$$\hat{I}_i = \sum_j \frac{m_j}{\rho_j} W_{ij} \cdot (\vec{r}_j - \vec{r}_i)(\vec{r}_j - \vec{r}_i)$$

Assuming that the density is constant (and equal to 1) this tensor represents the inertia tensor of a rigid body weighted by the smoothing kernel. The eigenvalues of this tensor provide valuable information regarding the local anisotropy. When the particle distribution is isotropic, the ratio of the lowest and highest eigenvalue equals one. If the distribution is not isotropic this ratio will be less than one and the distribution may not be stable.

For a 2D test case in the x-y plane an initial, rectangular particle distribution in a 3 m long and 1 m high domain has been created employing a small increment in size in both directions. The WVT algorithm, including the option that the particles become larger until they have reached the boundaries, has been used to fill a 3.75 wide and 1.6 m high box. Figure 3 shows the initial particle distribution and the eigenvalue ratio. It may be observed that the distribution is isotropic only in a region close to the diagonal. Despite the fact that the smoothing length (which is proportional to the particle size in the figure) is smooth, such a distribution loaded under gravity is known to generate significant flow [11] including

upheaval of the free surface. After about 1300 steps of the WVT iteration (not optimized at the time) the domain has been filled with a particle distribution that not only exhibits a smooth variation in particle size, but for which the ratio of anisotropy is very close to one (except along the boundaries), as to the right in figure 3.

This distribution has been exposed to gravity; the resulting flow is quite small with a maximum velocity of 0.1 m/s during 8 seconds which may be attributed to settling of the particles near the boundaries. The particle and pressure distribution - which is close to the hydrostatic one - shows that the distribution is stable enough to be used for relevant flow simulation. With a minimum distance between adjacent particles of about 12.6 mm, 37200 particles would have been required to fill the entire domain in cubic arrangement whereas the current model contains only 4800 particles (ratio ca. 8/1). This reduction of the number of particles may even be larger for 3D SPH simulations.

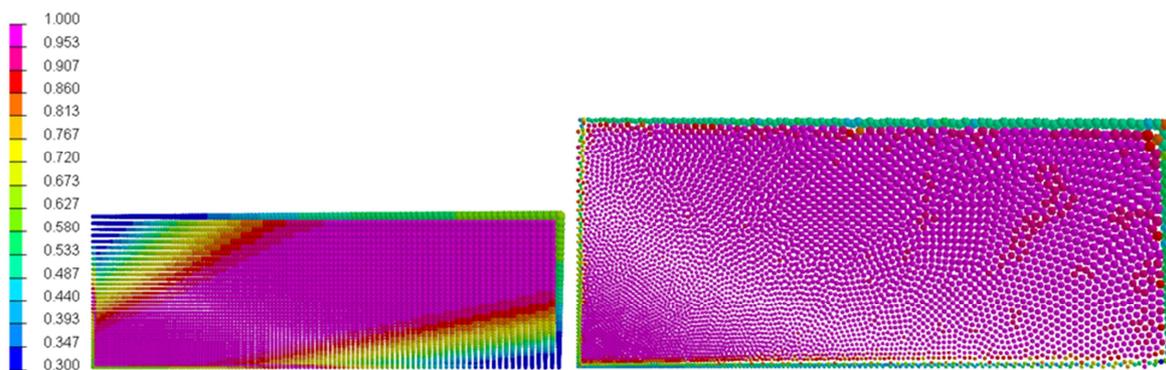


Figure 3: Particle configuration and the ratio at the start (left) and the end (right) of the WVT iteration.

3.4 Hydrostatic initialization and pressure gauges

Another feature that helps to reduce the CPU effort when attempting to obtain more accurate results is that of a ‘hydrostatic equilibrium condition’. Adding the hydrostatic pressure to the dynamic pressure from the start of the simulation avoids the need to conduct an initialization-simulation to obtain hydrostatic pressure equilibrium dynamically.

Moreover, a ‘gauge’ feature provides a mechanism to monitor the pressures and, in relevant situations, the free surface level. A ‘pressure gauge’ may be considered as the computational equivalent to physical pressure gauge in an experiment [13]. They may be positioned anywhere within an SPH domain without influencing the results. Due to the averaging conducted over nearby particles, the pressures obtained suffer less from oscillations observed for regular particles.

3.5 Suction effect

Suction is defined as the tension force when the water remains attached to a surface while being lifted out the original water domain and is a factor that significantly influences the dynamic response of an aircraft during ditching. Simulation of ditching based on first principles would require not only a cavitation model but also surface tension effects. Since it is not feasible to use a fine enough scale to model this for ditching, the separation stress

feature of the contact definition from VPS/PAM-CRASH is used instead. After contact between a node and a segment is established, it is maintained as long as the ‘penalty stress’ remains smaller than the user defined separation stress. For this algorithm there is a search radius which equals the product of separation thickness factor SEPTHK and contact height HCONT as shown in Figure 4.

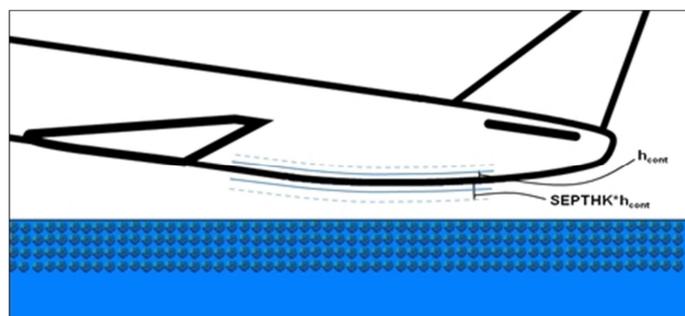


Figure 4: Modelling suction forces using the separation stress feature within VPS/PAM-CRASH [2].

Using this suction feature, it has been demonstrated a good correlation of pressure results between test and simulation both in overpressure and suction regions for ditching tests [2]. Moreover, using similarity rules for the phenomenological separation parameters properly, the same kinematic and pressure behaviour was reproduced at different scales [6].

3.6 Additional flow phenomena

Cavitation may also be relevant to ditching and numerical models to account for the presence (and generation) of vapour have been implemented for SPH [14]. For the presented simulations only the simplified pressure cut-off model has been used to model cavitation.

In order to model aerodynamic forces acting on the aircraft during the free flight phase a VPS/PAM-CRASH user subroutine has been created [15]. The aerodynamic forces are concentrated at the aircraft centre of gravity by means of classical lift, drag and pitch moment of longitudinal flight. This approach allows considering aerodynamic effects in terms of basic aircraft aerodynamic coefficients at the ditching conditions. It is recommended to include gravity and aerodynamics when long simulations with several rebounds are performed [2]. However, because aerodynamic loads are low compared to hydrodynamic loads, the effects of aerodynamics may be neglected for single impact simulations.

4 TEST CASES

4.1 Vertical wedge impact

One example of the validation procedure for the coupled SPH-FE approach discussed above is the test case of a two-dimensional wedge impact from Battley et al. [16], who performed motion-controlled vertical impact experiments using a rigid wedge of 10 degrees dead-rise angle impacting with constant velocity of 3 m/s. Experimental pressure results are available for three positions along the centre line between keel and chine.

The numerical model consists of a wedge structural model and a two-dimensional water

domain filled with particles with smoothing length of $h = 2 \text{ mm}$ and it is symmetrical with respect to the vertical axis [13]. Pressure gauge particles are positioned on the actual surface location of the wedge. Numerical pressure results for the gauge particles are validated against experimental data in figure 5; this comparison shows good correlation of peak values. However, after the peaks diminish, the numerical residual pressures are higher compared to the experimental ones. It is believed that the general overestimation of pressures in the simulation is caused by the two-dimensional nature of the numerical model. Nevertheless, it has been demonstrated that the coupled SPH-FE approach allows for reasonable results for this test case.

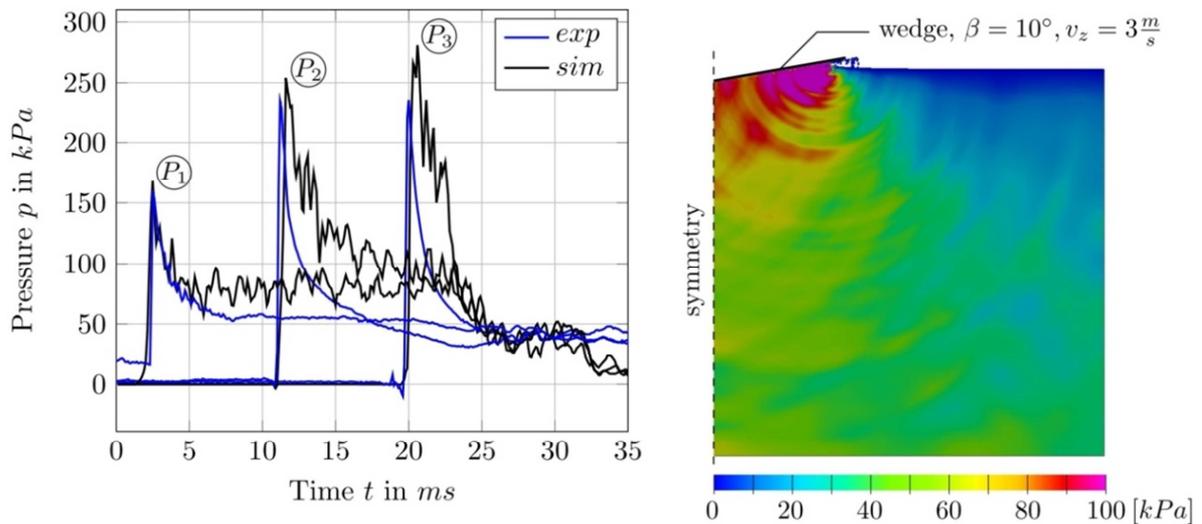


Figure 5: Comparison of experimental and numerical pressure results: Time histories at positions P1-P3 and pressure contour plot at $t = 20$ ms.

4.2 CN235 aircraft rigid subscale ditching

During the Airbus Military CN235 ditching certification a test campaign was performed with a subscale rigid mock-up [5]. The reference test case corresponds to 14 m/s and 8 degrees pitch angle. The numerical model of the CN235 rigid mock-up has been prepared in VPS. It is a symmetric model at 8 degrees initial pitch attitude. The numerical model of the water block consists of SPH particles and an external block of classical continuum finite elements to reduce the computational effort. This model includes the previously described suction model and aerodynamic forces using a user subroutine.

Figure 6 shows the impact sequence for the Airbus Military CN235 subscale rigid model. The impact occurs at the rear fuselage and the mock-up slightly nose-up. Some particles seem to stick to the fuselage which would be nearly impossible to model with mesh-based methods. The suction effect provides a realistic nose-up effect observed in ditching tests. Figure 7 demonstrates that inclusion of suction is essential for a correct prediction of the aircraft trajectory.

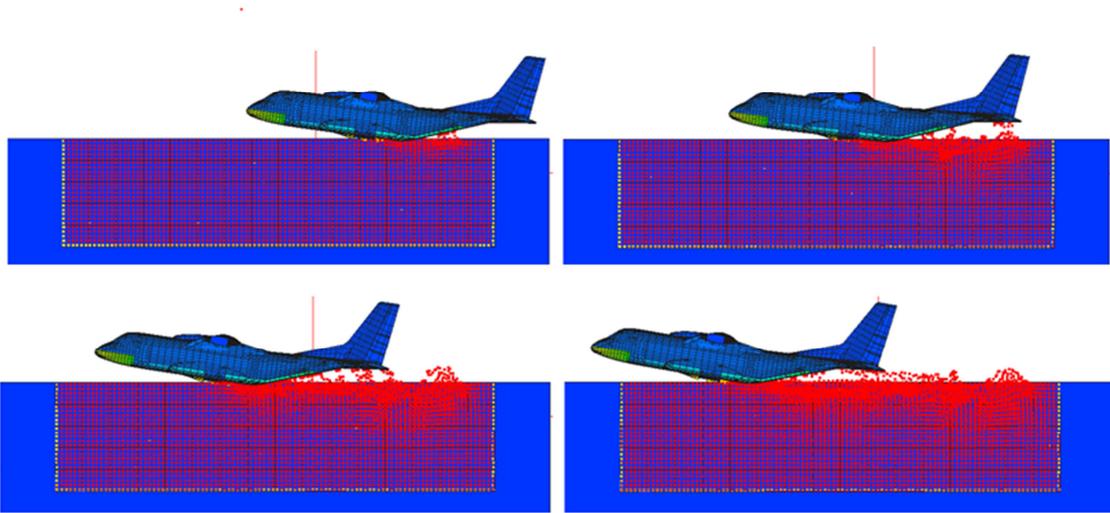


Figure 6: Airbus Military CN235 subscale rigid model with suction effects at 0.05, 0.10, 0.15, 0.20 seconds from left to right, top to bottom.

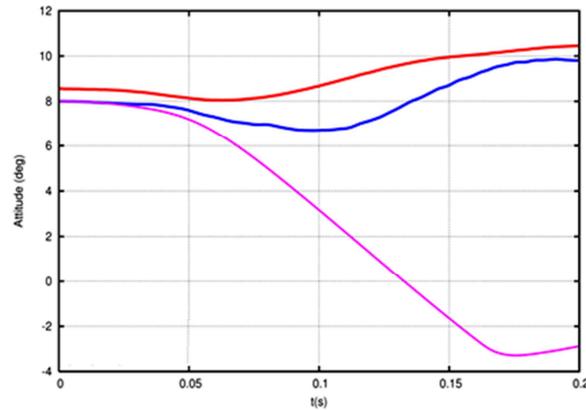


Figure 7: Comparison of pitch angle time history for Airbus Military CN235 subscale aircraft in experiment and numerical simulation with and without suction modelling. (Red: Test, Magenta: No suction, Blue: with suction)

5 CONCLUSIONS

This paper emphasizes the advantages of advanced numerical simulation methods for assessment of the survivability of aircraft during ditching. The complex physics of the free surface flow and the interaction with deformable structures may suitably be simulated by a combined SPH-FE approach. Various innovative features have been introduced into the SPH method to improve the accuracy of the obtained solutions and to reduce the total number of particles required in the simulation while maintaining a fine enough distribution in the impact regions. The relevance to include suction has been demonstrated and the similarity rules for scaling were retrieved with the phenomenological approach for suction adopted [6].

Two test cases have been presented to demonstrate the capabilities of the approach. For the CN235 subscale model it was demonstrated that inclusion of suction in the model is essential for simulation of the correct kinematics of ditching. Further work on the numerical modelling approach and an experimental validation based on guided ditching tests is presented in [4].

ACKNOWLEDGMENTS

Most of the work leading to the results presented has received funding from the European Commission's Seventh Framework Programme under grant agreement no FP7-266172 and was performed within the project SMAES — SMart Aircraft in Emergency Situations.

REFERENCES

- [1] J. Campbell, J. 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).
- [2] Benítez Montañés, L., Climent Máñez H., Siemann, M. and Kohlgrüber D. Ditching Numerical Simulations: Recent Steps in Industrial Applications, in: *Proc. of Aerospace Structural Impact Dynamics International Conference*, Wichita, USA, 2012.
- [3] Climent, H., Benítez, L., Rosich, F. Rueda, F. and Pentecote, N. Aircraft Ditching Numerical Simulation, *25th International Congress of the Aeronautical Sciences* (2006).
- [4] Siemann, M. Kohlgrüber D., Benítez Montañés L., and Iafrazi, A., Numerical Simulation And Experimental Validation Of Guided Ditching Tests, in: *Proc. of 11th WCCM/5th ECCM*, Barcelona, Spain (2014).
- [5] EADS CASA, 'CN235: Scaled Ditching Test. Shared Information within SMAES Program', Memorandum N° ME-T-MM-110104, 2011/07/03.
- [6] AiMil Contribution to the SMAES WP2 Month 24 Progress Report (2013)
- [7] Gropengießer, W., Tassin, A., Korobkin, A., Cooker, M.J., Martin, L. and Rung T. Advancement of semi-analytical Methods for Simulation of Aircraft Ditching, in: *Proc. of 11th WCCM/5th ECCM*, Barcelona, Spain (2014).
- [8] Campbell J.C. and Vignjevic, R. Simulating Structural Response to Water Impact. *International Journal of Impact Engineering* (2012) , Vol. 49, pp.1-10
- [9] Groenenboom, P. H. L. and Cartwright, B. K. Pressure-corrected SPH with innovative particle regularization algorithms and non-uniform, initial particle distributions. *8th SPHERIC Workshop*, Trondheim, Norway, (2013).
- [10] Virtual Performance Solution 2013 Solver Notes Manual, ESI Group, Paris, France, (2013).
- [11] Groenenboom, P. H. L. Particle filling and the importance of the SPH inertia tensor, accepted for the *9th SPHERIC Workshop*, Paris, France, (2014).
- [12] Diehl, S., Rockefeller, G., C., Fryer, C.L., Riethmiller, D. and Statler, T.S. Generating Optimal Initial Conditions for Smooth Particle Hydrodynamics Simulations, *Preprint submitted to Monthly Notices of the Royal Astronomical Society*, 2012.
- [13] Siemann, M. and Groenenboom. P. Pressure Evaluation at Arbitrary Locations in SPH Water Impact Simulations, in: *Proc. of 3rd International Conference on Particle-based Methods*, Stuttgart, Germany, 2013.
- [14] Groenenboom, P. H. L., SPH for two-phase fluid flow including cavitation, in: *Proc. of 7th International SPHERIC Workshop*, Prato, Italy, 2012.
- [15] Benítez L. and Napoli C., SMAES 12 Months Review: Ditching Numerical Simulation, Airbus Military ME-T-NT-120018 (Mar 2012).
- [16] Battley, M.; Allen, T.; Schierlink, J.; Lake, S. and Pherson, P. Hydroelastic Behaviour of Slam Loaded Composite Hull Panels. *3rd HPYDC*, Auckland, New Zealand, (2008).