Numerical Simulation of Aircraft Ditching of a Generic Transport Aircraft: Contribution to Accuracy and Efficiency

Maria Inês Costa Cadilha
Instituto Superior Técnico, University of Lisbon
Supervisors: Dipl.-Ing. Martin Siemann, Prof. Afzal Suleman
November 2014

***Abstract*—Planned water impact should be covered for the certification of an aircraft operating over sea. In that case, the manufacturer should show compliance to the specific regulations relating to ditching. Most certification testing is currently experimentally based, due to the interdisciplinary character of the physical problem and complex hydrodynamic effects arising from the high forward velocity of the impacting body. However, test campaigns are expensive and time-consuming, and thus the industry is now turning its attentions to numerical tools to assist in crashworthy structural design.**

Of key importance for ditching investigations is the ability for the numerical method to accurately capture the high deformations of the fluid domain in such events. To that aim, the present thesis focuses on the classical finite-element (FE) method and the meshless Smoothed Particle Hydrodynamic (SPH) techniques, for the modeling of the water and aircraft structure. Contact is facilitated as both formulations are of Lagrangian nature. Recently, the hybrid FE-SPH explicit code VPS (formerly known as PAM-CRASH) from ESI-Group was extended by various features to simulate hydrodynamic effects. These innovations have been validated for guided ditching tests in which representative plates simulate ditching conditions. The present research aims to transpose the acquired knowledge to the problematic of full-scale, fixed wing aircraft involving high-forward velocity.

Within the framework of this task, a computational pre-processing tool is developed for automated model generation, and a comparison of different modeling approaches is performed in terms of quality of the results and computational requirements. Water modeling techniques are investigated with the objective of reducing computational effort. For the aircraft, different modeling schemes are considered, ranging from simple rigid body approaches to flexible FE models. A separation stress feature is tested to simulate suction forces between fuselage and water domain. Finally, shared memory processing and multi-model coupling computational schemes are considered.

Key words: Smoothed Particle Hydrodynamics, Fluid-Structure Interaction, Fixed-Wing Aircraft Ditching, Water Impact

I. Introduction

On January 15, 2009, US Airways Flight 1549 brought aircraft ditching into public attention. The airplane was struck by a flock of geese soon after takeoff from New York's La Guardia airport, losing thrust in both engines. The pilot told his passengers to brace for a hard impact and then set the plane down safely on the Hudson River. All 155 occupants safely evacuated the plane. More than the "Miracle on the Hundson", as it became known, it was a testament to the skill of the crew, and a proof of the robustness of the airframe of the Airbus A320.

In fact, since many airports are close to water and many flights are operating partly overseas, impacting on water is a possibility that could be encountered by most aircraft. Even tough this is (fortunately) a very rare event, regulations address ditching in the certification of an aircraft operating over the sea. The manufacturer should show compliance to the specific airworthiness regulations, which can be roughly summarized as in the following: the aircraft should be able to land on water as safely as possible and to float long enough in order to enable the passengers to evacuate.

The demonstration of compliance to ditching requirements may be realized by either testing or comparing to aircraft of similar design, of which the ditching behavior has already been certified. The use of sub-scale aircraft models, launched in a water tank, is currently more common to investigate various ditching conditions. The kinematics of the aircraft impacting on the water strongly depends on its configuration at the moment of the event. It is, therefore, required to thoroughly investigate the influence of position of flaps, location of the center of gravity, weight and/or eventual damage on the structure for the forced landing on water. External factors like a rough sea may have to be taken into account. As test campaigns are time-consuming and very expensive, the use of simulation tools capable of predicting the response of aeronautical components, or the entire aircraft, facilitates the development and the evaluation of new designs.

Numerical modeling provides a platform to evaluate new crashworthy concepts prior to expensive testing programs, and the global research focus is on developing efficient simulation tools, particularly using explicit Finite Element (FE) modeling methods. The challenge with simulating the ditching case is the high deformation of the water domain, which needs to be modeled effectively to predict the structural response. Standard FE models are optimum for structural representation, but grid based numerical methods suffer from difficulties which limit their applications to the water domain in ditching simulations involving forward velocity. One alternative which has recently been a major research focus is the use of meshfree methods for the water, which overcome the problem of restrictive mesh distortion. Exemplary is Smoothed Particle Hydrodynamics (SPH), a meshfree particle method based on Lagrangian formulation, which has been widely applied to different areas of engineering and namely to the study of aircraft water landing. One other significant advantage of SPH is the simplified contact between FE structural aircraft models and the water media for FSI, as both the structure and the water are modeled using Lagrangian formulations. Investigations using this method showed promising results, with the drawback of increased computational expense when compared to the standard FE formulation.

Within the course of the European commission's SMAES project, initiated in 2011, water impact was investigated using the FE-SPH method with the goal to permit cost effective design and entry-into-service of aircraft able to protect its occupants during a water landing. The results were validated with data originating from a guided ditching campaign, in which representative plates were experimentally investigated under realistic ditching conditions. Within this research, the hybrid SPH-FE explicit code VPS (formerly known as PAM-CRASH) from ESI-Group was extended by various features for improvement of water modeling and reproduction of FSI phenomena. The main objective of this thesis is to combine the state-of-the-art modeling techniques developed in SMAES with the aircraft modeling knowhow of the DLR Institute of Structures of Design, applying them to full-scale aircraft ditching simulations. The research also aims to explore advantages and limitations of the different modeling options currently available.

Within the framework of this task, a computational pre-processing tool was developed for automated model generation, and a comparison of different modeling approaches was performed in terms of quality of the results and computational efficiency. Accuracy and efficiency are forefront in order to provide an improvement in the predictability of loads occurring during ditching.

II. Coupled Finite Element-Smoothed Particle Hydrodynamic Formulation

The structural models in the current investigation are composed of rigid and flexible classic finite elements. The fluid domain is modeled using both weakly compressible SPH method, in the vicinity of impact where larger deformations are expected, and FE hydrodynamic volumes for the regions where less disturbance occurs.

Smoothed Particle Hydrodynamics was initially developed in 1977, by Lucy [1] and Gingold [2], for the simulation of astrophysics problems. Since then, many researchers have conducted investigations on the this method, improving it to the state of a very powerful formulation for problems governed by the Navier-Stokes equations. There are two main steps in obtaining an SPH formulation. The first step is to represent a function and/or its derivatives in continuous form as integral representation, and this step is usually termed as kernel approximation. The second step is usually referred to as particle approximation. In this step, the computational domain is first discretized by set of of particles representing the initial settings of the problem. During the simulation, field variables (e.g. density, pressure, velocity) on a particle are approximated by a summation of the values over the nearest neighbor particles [3]. A scheme is presented in Figure 1.

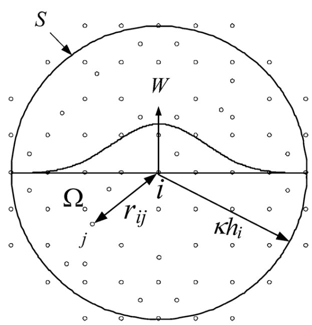

Figure 1: SPH particle approximation in problem domain Ω with a surface S. W is the smoothing function that is used to approximate the field variables at particle i using averaged summation over particles j within the support domain with a cutoff distance of κh_i [3].

The Wendland kernel function is used in the present research with a radius of $2h$ influence of twice the smoothing length h. The numerical stability is treated by using the standard Monaghan-Gingold artificial viscosity term [5]. The system of Navier-Skotes differential equations is closed by adding an equation of state (EOS) relating pressure to density. The frequently used Tait EOS (also referred as Murnaghan law) [6] with reference pressure p_0, speed of sound c_0, ratio of current over initial mass density ρ/ρ_0 and adiabatic exponent of the fluid γ reduces the computation time by artificially increasing compressibility, and is therefore used in the present thesis:

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

Coupling of the SPH water particles and the aircraft FE model is achieved by using a node-to-segment penalty contact formulation.

Previous publications highlight the advantage of modeling the water domain closer to impact with SPH elements due to high deformations perceived, and to use inexpensive classic hydrodynamic FE volumes in the areas of less disturbance. However, computational times remained excessive and suction forces could not be modeled using SPH for the water domain. Simulating aircraft ditching required a list of improvements over the state of the art to enhance the accuracy and efficiency of existing numerical tools. Within the EC-FP7 SMAES project, the German Aerospace Center, Airbus Military and ESI-Group teamed up to simulate the hydrodynamic effects with the hybrid FE-SPH code VPS from ESI-Group. To this end, the SPH module within VPS has been extended by various features including a separation stress option to mimic suction, periodic boundary conditions and particle regularization methods. Validation has been performed in studies using both sub-scaled models and guided plates impacting in water. The next logical step was to transfer this knowledge into full-scale aircraft ditching simulations, to further understand the potential of applying this methodology in numerical tools to assist in design and certification.

III. Automated Model Generation Tool for Numerical Investigations

Finite element analysis can be separated in three distinct phases: pre-processing, solution and post-processing. The pre-processing phase consists of developing an appropriate geometry, finite element mesh, designate suitable material properties, and apply boundary conditions as restraints and loads. When dealing with hybrid FE-SPH analysis, this also includes appropriate particle distribution and contact definition. Preparation of large models can thus be very time consuming and consistency between different modeling approaches is easily compromised due to the large number of parameters involved.

The VPS explicit solver requires a specific format of input data organized in so-called input cards. The setup of the model can be performed by using the windows-based graphical interface Visual-Crash PAM and exporting the resulting geometry files, or by directly typing in these cards into a text file from scratch. This latter option requires good knowledge of solver commands and corresponding keywords.

The automation of this input generation was performed by developing an intuitive pre-processing script dedicated to modeling of ditching scenarios. The tool will automatically generate all the necessary VPS cards (e.g. nodal points, materials, function curves, contact definitions) for the model specified by the user. This includes the setup of several aircraft and fluid domain configurations. If specified, the tool will directly feed the input files to the solver and start the referred simulation, and all what is left to the user is the post-processing of the results.

The code was developed predominantly in Python programming language (Version 2.7.2), making use of the commercial software ANSYS (Version 14.5) to perform certain geometry meshing tasks. The VPS input generated is specified for version 2010, due to restrictions imposed by the development version available.

The user is allowed to choose between a graphic interface or a fully automated mode in the tool. Input is made through water parameter definition and desired aircraft geometry. Several different modeling approaches for the aircraft are allowed, and are described in the following paragraphs.

General and Detailed Finite Element Models (GFEM/DFEM) generation is entirely performed by an incorporated pre-existing in-house tool, AC-CRASH. This fuselage modeling tool was developed to automatically generate a global FE model using elastic beam elements for preliminary sizing purposes (GFEM). To be also used in crash simulations this model was extended in a way that certain regions where high plastic deformations or failure is expected may be modeled by the use of shell elements and a finer mesh (DFEM). Those local refinements are named detailed regions. By this approach the advantages of both discretization methods are unified: the global aircraft stiffness is mapped correctly by cheap beam elements and the areas of interest can be computed using shell elements, thus benefiting from their advantages over beam elements (see Figure 2).

Rigid Body Models (DMM): The RBM is a simplified, undeformable version of the detailed FE model presented before. It is created by constraining the shell elements of

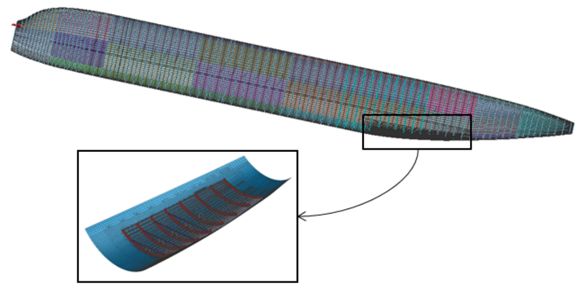

Figure 2: DFEM with local refinement in the area first impacting the water.

the fuselage skin into a single rigid entity, and deleting all the remaining structural items (frames, stringers, passenger seats). Total mass, center of gravity and rotational inertia are transferred to the simplified geometry. The entire transformation process is automated by the developed pre-processing tool, through mesh file analysis. The main advantage to this approach is the considerable reduction in computational expense of the model.

Dynamic Master Models do not allow for skin deformation, but present an alternative to compute energy propagation through the fuselage. Aircraft simplified beam FE models, also known as stick models, are commonly used in civil aircraft design. Accurate prediction of bending and twisting deformations of the aircraft structure depends on extracting the stiffness properties of the main structure and applying it to a set of beam elements. The model considered in the present thesis (DMM) takes from this condensed beam concept and adapts it to be utilized in the context of ditching simulations. This is done by connecting a set of rigid shell elements to the beam structure, in order to represent the contact surface with the water domain. Stiffness distribution is computed with recurse to another external tool, which computes local stiffness properties for each section in between frames, directly from the GFEM or DFEM model (see Figure 3).

The correct mass distribution, originating from nodal mass data, is applied after the geometry generation is completed. The point mass distributions to be applied to the different aircraft models are defined in separate text files. The developed toolbox offers several examples of these distributions but the user is free to modified

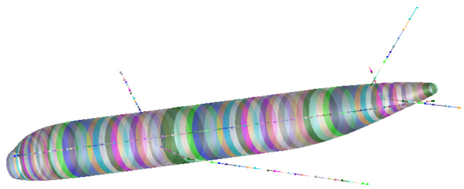

Figure 3: Example of Dynamic Master Model

them in order to test different mass load cases. The tool also provides a beam model of the wing and empennage structures. This segment can be left out, attached to any of the models as a rigid structure or attributed a variable stiffness distribution. Its purpose is to evaluate the influence of the flexibility and mass of these structures on the fuselage only, and consequently no contact between these parts and the water is included.

Water generation follows. Concerning water features, one can mention the application of translating periodic boundaries, which allows moving the SPH domain with a velocity linked to the COG of the aircraft (see Figure 4), or the active box, where particle location is checked at each time step and particles outside the domain of interest (further from the impact area) are deactivated. For the latter option, an algorithm was implemented to linearly vary the length of the active region throughout the simulation.

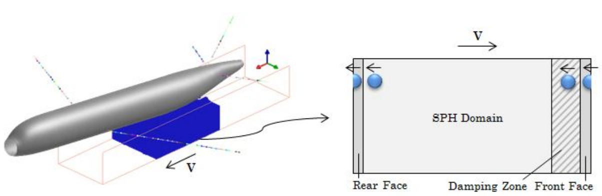

Figure 4: Example of Periodic Boundary Conditions

Coupling between FE and SPH elements for the water domain and in between structure and water was also automated, and the separation stress feature can be activated to portray suction forces. Finally, different memory processing schemes are allowed, such as shared memory processing or Multi-Model Coupling (MMC). The latter allows for the structure and water to be separated and computed with different time steps, and requires a specific data layout to be implemented. The pre-processing tool successively achieves the required input file architecture for all models, so long as combination between approach is allowed for the VPS solver.

IV. Comparison of Modeling Approaches

In order to provide a test to the capabilities of the developed pre-processing tool, and the potential of recently implemented features in the VPS explicit solver, several modeling options were investigated.

The aircraft used in the simulations performed represents a generic single aisle, 37.5m long airplane with a capacity to hold up to 150 passengers. Maximum fuselage height reaches 4.1 m and main wing is defined with a span of around 34 m and mean sweep back angle of 25 degrees. Vertical tail extends to 9.5 m height (from lowest fuselage point). Wings are modeled as representative beam elements and no engines nor landing gear are included in the geometry.

Mass distribution originates from available mass point data representative of a 50% payload and 50% fuel upon impact, totaling at around 72500 kg. All simulations are performed considering null yaw and roll angles, and an angle of attack of 8 degrees on approach. Initial horizontal and descent velocities are fixed as $v_x = 65m/s$ and $v_z = -1.5m/s$, respectively.

In order to portray realistic conditions, gravity is applied to both structure and water and a simple linear lift model is applied to the aircraft model. The latter consists of linearly decreasing the lift force from balance (equal to gravity force) to zero during the first second of impact. Air is not modeled.

It is important to highlight that no experimental data is available for comparison, and as such the presented results should not be seen as an accurate representation of the aircraft dynamics upon ditching. The objective is to compare the different modeling approaches in terms of consistency, accuracy and time efficiency.

Investigations Concerning Water Modeling

In this first part of the research, the RBM approach is used for all simulations for time efficiency. The classic approach using a full pool with FE and SPH elements (see Figure 5) was used as a reference, and the influence of different parameters was studied.

In the reference analysis, splash formation and different phases over impact could be observed. The aircraft first

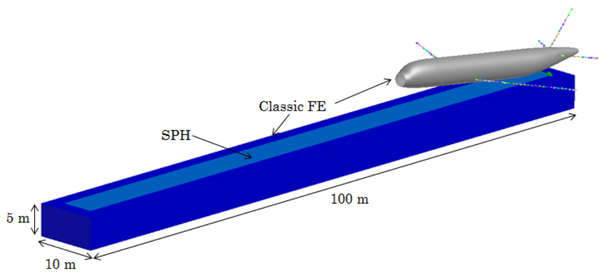

Figure 5: RB aircraft model and full length FE-SPH option for water basin.

impacts the water domain and then bounces, landing a few meters ahead on a second impact. The disturbance caused by the impacting body is plausible, and splash effects can be reproduced due to the nature of the SPH method applied in the central region of the water domain. The classical Lagrangian finite elements placed in the outer region of the basin still deform, but not enough as to cause a dramatic decrease in critical time step.

Cross-section of the pool should be enough as to provide sufficient impulse and remove boundary interference. The investigations performed show that the FE volumes used around the particles for extension are computationally cheap and as so cross-section can be increased with no excessive CPU expense. The contact between particles and FE volumes is also investigated, testing different particle element to FE size ratios. A penalty contact is also tested for contact computation, as oppose to the tied connection typically used for this purpose. Results show that even though no significant differences are observed for the different referred ratios and contact algorithms, further investigations should be performed on the subject. One other important aspect is the long CPU times, due to the required extensive length of the SPH domain. Translating periodic boundaries address this issue.

For the translating periodic boundary conditions, it was found that the translating SPH domain on its own does not provide time improvements as the reduction in length is overcome by the necessary increase in cross-section. The problem was addressed to the ESI-Group, which developed an algorithm to allow for a U-section of FE hydrodynamic volumes to be used in coupling with the translating particle domain. The algorithm locates the FE volumes outside the region adjacent to the particles at each time step, and sets accelerations of these elements and velocities to zero. As a result, the FE which are not in contact with the particles do not collapse under their own weight. This option reveals to be effective in producing constant results and achieves a considerable reduction (54 %) in CPU time.

The active box via nodes option was also considered together with the FE volume U-section mentioned. It was shown that the ability to vary the active domain over the period of simulation brings further improvements in terms of computational time (65% overall reduction in CPU time), and results are practically the same in comparison the full-length approach (see Figures 6 and 7).

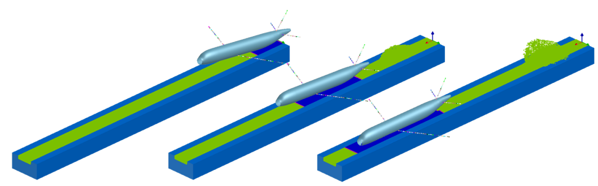

Figure 6: Adaptive Active Box over simulation (from left to right, $t = 0ms$, $t = 450ms$ and $t = 900ms$). In the water basin: hydrodynamic FE volumes (light blue), activated particles (dark blue), deactivated particles (green)

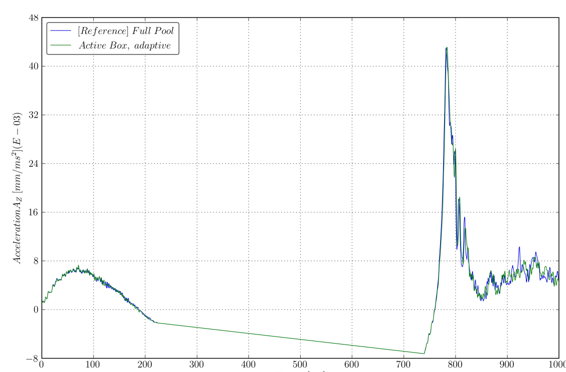

Figure 7: Comparison of COG acceleration histories for full-length pool and adaptive active box options (CFC180).

Another important observation was the effect of particle refinement in the acceleration history of the aircraft. Different particle sizes were tested, maintaining the particle length to FE size ratio. It is observed that a big discrepancy arises between larger and smaller particle

Figure 8: Mesh Refinement, Active box constant size, with time step constraint (CFC180)

modeling. The effect of time step reduction in contact computation is considered. It is proven that by artificially reducing critical time step, results show convergence. Results for 100 mm and 70 mm particles with and without artificial time step constraint are shown in 8, for 300 ms of simulation.

The accelerations showed that when restraining the critical time step to the same maximum value, the results are much closer between simulations. Other than this, the results showed the initially expected reduction of oscillations with smaller particle size. This is a very relevant observation, since it indicates that time step is a critical factor to the accuracy of the results, other than particle refinement or modeling approach for the water basin. One possible explanation could be that due to the violent impact of the large structure, the critical time step defined by the individual elements is maybe not low enough to accurately portray the contact between aircraft and water. Nonetheless, this is an unsure conclusion, as it should not be the case. Further investigations are required on the subject as not enough simulations were performed in order to understand the relation between impact force and necessary time step for contact.

Investigations Concerning Aircraft Modeling

analysis is performed by using the same input files and alternating within the four modeling possibilities allowed by the tool, from simple rigid body to fully detailed finite element models. The different models are first compared in terms of global mass and inertias to ensure consistency. It is proven that even though the developed pre-processing tool used different methods for mass distribution in each of the models, global properties are successfully consistent. Modeling approaches are tested to simpler to more detailed: first the RBM, followed by the DMM and GFEM schemes, and finally the DFEM. It is shown that in this particular case using flexible beams for wing and empennage structures does not have a significant influence in kinematics.

The geometry tested contains a total number of 87 frames located length wise in the fuselage, and as so the DMM approach tested contains the same number of independent "rings". The dynamic master model correctly portrays structural deformation through the fuselage structure, but for the tested case this parameter does not seem to significantly influence acceleration time histories. In Figure 9 the model after 120 ms of simulation is displayed. Due to its small magnitude, deformation is amplified by a factor of 200 in the referred image. It could be observed that the energy arising from the impact is propagated by the elastic elements present, resulting in a quasi-sinusoidal dislocation of the skin rings across the fuselage.

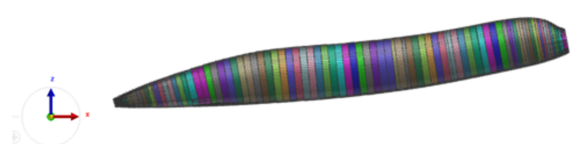

Figure 9: Deformation of DMM at $t = 120ms$. Structural deformation is amplified in all directions by a factor of 200. Beam elements and water basin are hidden for clarity purposes.

The GFEM approach is tested with coarser and finer meshes, and it is observed that a fine mesh is necessary in the impact region to accurately portray skin deformation and eventual panel failure. To this aim, the DFEM option was considered. Overall conclusion was that skin deformation and panel failure appear to be a decisive factor in the contact force and acceleration histories, as results from the latter approach present a considerable difference to those using rigid elements for the fuselage skin (see Figures 10 and 11). Time increase is also remarkable: the latter approach took up to 12 times the necessary CPU effort of the rigid body model.

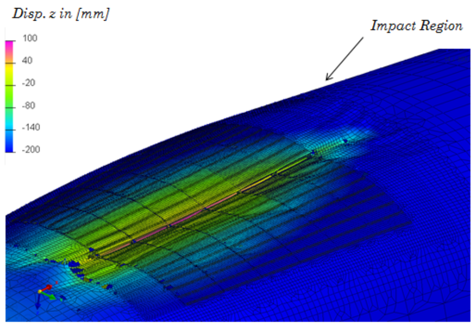

Figure 10: Displacement in Z-direction of DFEM refinement region, at t=390ms.

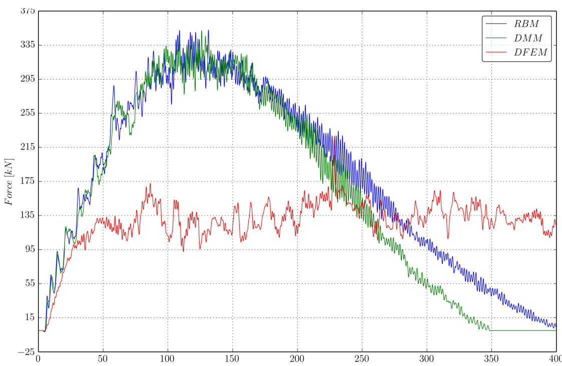

Figure 11: Contact force time histories in Z-directions for RBM, DMM and DFEM models (CFC180).

	RBM	DMM	GFEM
Computational time [h]	6.8	18.5	82.9

Table I: Global finite element investigations: computational times of rigid and flexible wing options.

In terms of the consistency of the results, several observations were made. First, previous investigations (for the water basement particle refinement) showed that considerable differences in critical time step of the simulations may have an effect on the contact between the water particles and the FE fuselage structure. The RBM presents a critical time step of around one order of magnitude higher than the DFEM, due to element size and elasticity. It is therefore considered that the significant decrease in time step, visible when comparing the three approaches with rigid wing in 11, may be a factor for the disparity of the results.

Another factor for the different results between modeling approaches was the considerable deformation and consequent failure of the skin elements portrayed by the DFEM. Even though one could visualize structural deformation throughout the fuselage in the DMM, the results of the DFEM indicate that skin deformation is of high significance for fluid-structure interaction during the first impact. However, aircraft pitch angle evolution shows the same tendency over the three results, although with a less smooth behavior for the DFEM.

It is therefore advised to weigh in accuracy and efficiency costs, as the several days necessary for the solution of the DFEM may be impractical for an analyst. If global pitch kinematics is to be evaluated, the RBM approach gives a reasonable first approximation with considerable time benefits. For harder impact conditions (such as higher downward velocity upon impact), the DMM approach may be considered as overall structural deformation is portrayed. On the other hand, if fuselage skin rupture is analyzed or accurate portray of accelerations is required, it is advised to opt for the DFEM approach. Nonetheless, the lack of experimental data for validation of the aircraft structural models in the present investigations should be recalled, and as such previous conclusions may need further analysis. Still, the comprehensive methodology followed is valid for any model, and the automated preprocessing tool facilitates the testing of all options in coping with the best approach for the ditching analysis in question.

Investigations Concerning Separation Stress Feature

In some ditching experimental test cases, suction forces arise in the rear part of the fuselage, caused by an acceleration of the water flow around the impacted area, leading to a drop of the water pressure in the region. The referred forces are located with at a fairly large distance from the structure's center of gravity, and therefore even if the force magnitude is not elevated the momentum produced is of significance. In order to assess the importance of the mentioned effect in the present full-scale ditching simulations, investigations in [8] are taken as reference. In the latter research, a standard transportation aircraft geometry (of similar length as to the one used in the present investigations) is tested with the separation stress feature available in the VPS FE-SPH module. The model used in the simulations is scaled-down by a ratio of 1:8 applying Froude scaling rules, and a rigid body approach is used. Separation stress $SPSTR$ and separation thickness factor $SPTHK$ are investigated until suction is realistically portrayed. As models are similar in shape, Froude rules were employed to the separation feature results in order to apply them to the present research.

In the reference investigation, the aircraft model remained attached to the water surface during the first 350 ms of impact. Similar results were expected for the full-scale case now tested. However, the behavior of the aircraft is not as predicted. At $t = 300$ ms, the aircraft was no longer in contact with the water surface, similarly to what had been observed in previous investigations were the separation stress featured was not used. The main difference is a layer of particles that remain attached to the structure, contrary to the objective of simulating suction forces. It was thought that the disparity arose from the lack of scaling of the particle size, which was fairly coarser in the reference simulations. To this aim, $SPSTR$ and $SPTHK$ were increased to compensate for this effect. However, no improved results are obtained. Finally, the same was tested with a larger particle size. The results were presented in Figure 12 in comparison with the same simulation without the separation stress feature.

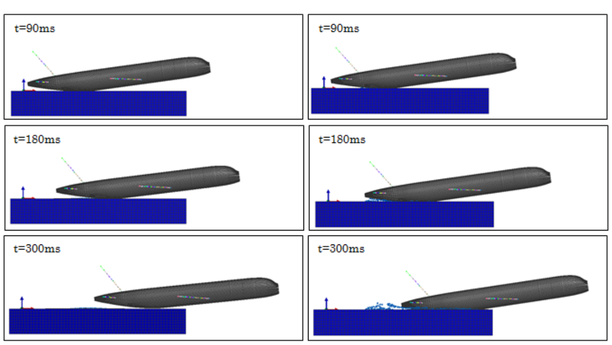

Figure 12: Separation stress investigations: simulation without separation stress feature activated (left) and with $SPSTRS = 1E3GPa$, $SPTHK = 1.75$ (right) for 300 mm particle spacing.

It could be observed that in the latter case, the application of the separation stress feature resulted in a longer attachment of the aircraft to the pool. It seems as if the relation between individual particle size and mass of the impactor plays an important role for this to be observed. Further investigations should be conducted on the subject as larger particle size may lead to less accurate results in terms of pressure distribution. However, the main objective behind this investigation was to prove if suction forces could be portrayed for full-scale aircraft models impacting on water recurring to the FE-SPH method. Despite initial difficulties in parameter scaling, it is thought that the results prove that this statement is true.

Investigations Concerning Computational Aspects

In the previous benchmarking sections, computational efficiency has been address by the modeling options point of view. A shared memory processing (SMP) scheme was used in the previous investigations, running the simulations in one computational node over eight cores. However, computational configuration may have an even larger effect in CPU requirements. For the referred reason, the present section aimed to explore the capabilities of the Multi Model Coupling (MMC) option. MMC allows for the simulation domain to be separated in aircraft and water, and for each of the components to run on its own time step. Contact is then achieved over super-cycles, where data from both parts is interchanged. The prospects of this feature are promising, especially for the more detailed aircraft modeling approaches, where the time step of the structure is considerably lower than the one of the water particles. Another feature is the possibility of assigning a different number of CPUs for each part.

The MMC parameters were previously studied in [9] for the vertical impact of a fuselage section on water, using a computational cluster assembly. These investigations concluded that optimum results were achieved when attributing the same number of nodes to both structure and water, and that TSR $\Delta t_{water}/\Delta t_{struct.}$ should be kept under 30 to ensure correct contact computation. Taking these regards as a reference, the MMC option was tested in the thesis for the present model with the GFEM approach.

One important notice is that to date the MMC option was only available for the commercial version of the VPS solver. Features such as active box via nodes were exclusively available in the development version used in previous simulations. This means that both, until the moment of the present research, could be combined.

The methodology followed for the MMC simulations was similar to the one performed in [9]. One computational node is used and 4 cores were attributed to each the structure and the water. First, no time step ratio was artificially imposed, to verify the natural TSR of the full model. In a second step, TSR was artificially constrained by imposing the corresponding value in the water section of the simulation. In the present case, $\Delta t_{water}/\Delta t_{struct.} = 30$ and a more conservative $\Delta t_{water}/\Delta t_{struct.} = 5$ were tested. Finally, after the simulations were completed, the contact force history was checked for ensure consistency. This data is plotted in Figure 13.

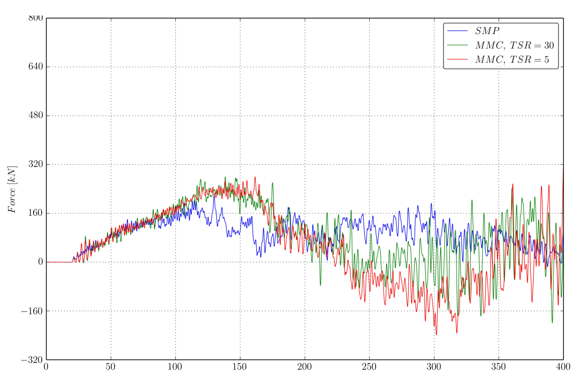

Figure 13: MMC investigations: GFEM model using SMP and MMC with time step ratios TSR=30 and TSR=5.

	SMP	MMC, TSR=30	MMC, TSR=5
Comp. time [h]	132.5	24.4 (-81.5%)	49.8 (-62.4%)

Table II: Computational times for SMP (No MMC) and MMC computational schemes with TSR=30 and TSR=5.

It is in fact very encouraging to observe that even with fairly low TSR, time savings overcome 60%. Although this comes with a penalty in accuracy, the remarkable decrease in computational expense should be taken into consideration. It was concluded that as computer technology evolves, memory processing schemes should not be disregarded as it was obvious that this could lead to a significant reduction in time spent analyzing more detailed aircraft models.

V. Conclusions

Summarizing, the contribution of the present work for the state of the art in FE-SPH ditching simulations can be separated into two main achievements. First, a parametrized pre-processing tool was developed for generation of complete ditching models, including advanced modeling options concerning aircraft, water, contact and memory processing schemes. The developed tool is now available for comprehensive studies to be efficiently performed in the future, and possibly for the ditching load case to be incorporated in larger computational chains. In second place, the state-of-the-art features developed during the SMAES project for VPS concerning water and FSI modeling, the pre-existing in-house tools of the DLR dedicated to structural modeling, and an adapted condensed beam approach (DMM) implemented by the author have been successfully applied to the case of full-scale fixed-wing aircraft ditching simulations, with very encouraging results.

References

[1] Lucy, L. B. (1977). "A Numerical Approach to the Testing of Fusion Processes". In *The Astronomical Journal, 82*, pp. 1013-1024.

[2] Gingold, R.A. and Monaghan, J.J. (1977). "Smoothed Particle Hydrodynamics: Theory and Application to Non-Spherical Stars". In *Monthly Notices Royal Astronomical Society, Vol. 181*, pp. 375-389.

[3] Liu, M. B. and Liu, G. R. (2010). "Smoothed Particle Hydrodynamics (SPH): an Overview and Recent Developments". In *Archives of computational methods in engineering, 2010, vol. 17, no 1*, pp. 25-76.

[4] Monaghan, J. J. (2012). "Smoothed Particle Hydrodynamics and Its Diverse Applications". In *Annual Review of Fluid Mechanics, 44(1)*, pp. 323-346.

[5] Monaghan, J. J. (1992). "Smoothed particle hydrodynamics". In *Annual review of astronomy and astrophysics, 30*, pp. 543-574.

[6] Tait, P. G. (1965). "Report on some of the physical properties of fresh water and of sea water". Johnson Reprint Corporation.

[7] Schwinn, D., Scherer, J., Kohlgrüber, D., and Harbig, K. (June, 2013). "Development of a Multidisciplinary Process Chain for the Preliminary Design of Aircraft Structures". In *NAFEMS World Congress 2013*.

[8] Benítez, L. M., Montañés, H. C., Siemann, M., and Kohlgrueber, D. (November, 2012). "Ditching Numerical Simulations : Recent Steps in Industrial Applications". In *Aerospace Structural Impact Dynamics International Conference*, pp. 12-40.

[9] Hyra, P. (September, 2012). "Investigation of Multiscale Modeling/Multi Model Coupling - Numerical simulations of aircraft structures impacting on water". DLR-Interner Bericht.