Numerical Simulation of Aircraft Ditching of a
Generic Transport Aircraft

Contribution to Accuracy and Efficiency

Maria Inês Costa Cadilha

Thesis to obtain the Master of Science Degree in

Aerospace Engineering

Supervisors: Prof. Afzal Suleman,
Dipl.-Ing. Martin Siemann

Examination Committee

Chairperson: Prof. Filipé Szolnoky Ramos Pinto Cunha
Supervisor: Prof. Afzal Suleman
Members of the Committee: Prof. André Calado Marta

November 2014
Acknowledgments

First of all, I would like to express my sincere gratitude to my supervisor Martin Siemann, for the continuous support throughout my research at the German Aerospace Center (DLR). His guidance helped me in all the time of research and writing of this thesis. I could not have imagined a better mentor.

Secondly, I would like to thank Dominik Schwinn, Julian Scherer, Jannik Waldner and all the remaining colleagues at the DLR Institute of Structures and Design, for their contribution and their help with the simulations.

In addition, I would like to thank my supervisor at IST, Prof. Afzal Suleman, for his advice during the execution of this thesis.

I also have to thank all my friends, who in some way or another made my past 5 years of studies in Aerospace Engineering both enjoyable and fun. Those who know me since we were in school, those I met in Técnico and those from all around the world I met during my studies in the Netherlands and in Germany have turned this journey into an experience I will never forget.

I also thank Joan, who encouraged, supported and helped me throughout all the moments of this work.

Finally, I would like to dedicate this thesis to my parents. Without their unconditional support, none of this would have been possible.

Lisbon, November 2014

Maria Inês Costa Cadilha
Abstract

Since many airports are close to water and many flights operate partly over the sea, planned impact on water, known as ditching, is an emergency condition which can be encountered by most aircraft. For this reason, manufacturers must show compliance to specific certification regulations concerning this subject. The present thesis aims to contribute to the state of the art in numerical simulations of full-scale aircraft ditching.

Particularly, this research focuses on the use of the classical finite element (FE) method for the modeling of the aircraft structure, and in the meshless Smoothed Particle Hydrodynamics (SPH) technique, especially suitable for problems dealing with large deformations, for modeling of the water domain upon impact. The explicit code VPS, used in this work, had several new features implemented within the European Commission’s project SMAES (SMart Aircraft in Emergency Situations) for the modeling of fluid-structure interaction. The main objective of this thesis is to combine the 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.

In the framework of the task, a Python-based pre-processing tool is developed for automated model generation. In a second part, the developed tool is successfully used in order to conduct a series of numerical studies on different modeling approaches for a generic full-scale transport aircraft. Water modeling techniques with the objective to reduce the computational effort are investigated. For the aircraft, different modeling schemes range from simple rigid body approaches to detailed FE models. A separation stress feature is tested to simulate suction forces between the fuselage and the water domain. Finally, shared memory processing and multi-model coupling computational schemes are considered. The final conclusions are drawn in terms of the accuracy and the efficiency of the state-of-the-art modeling techniques investigated.

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

Uma vez que muitos aeroportos estão perto de água e muitos voos operam em parte sobre o mar, o impacto planeado em água, conhecido como amaragem, é uma condição de emergência que pode ser encontrada pela maioria das aeronaves. Por esta razão, os fabricantes devem demonstrar conformidade com regulamentos específicos de certificação referentes ao impacto em água. A presente tese tem como objetivo contribuir para o estado da arte em simulações numéricas de amaragem de aeronaves em escala real.

Particularmente, esta pesquisa centra-se na utilização do método clássico de elementos finitos (FE) para a modelação da estrutura da aeronave, e na técnica sem malha Smoothed Particle Hydrodynamics (SPH), especialmente adequada para problemas que lidam com grandes deformações, para a modelação da água no momento do impacto. O código explícito VPS, utilizado neste trabalho, teve várias novas funcionalidades implementadas no âmbito do projecto da Comissão Europeia SMAES (SMart Aircraft in Emergency Situations) para a modelação de interação fluido-estrutura. A presente abordagem combina os conhecimentos adquiridos no SMAES com o conhecimento de modelação estrutural de aeronaves do Instituto de Estruturas e Design do DLR. O objetivo principal é investigar as vantagens e limitações de diferentes opções de modelação.

No âmbito desta tarefa, uma ferramenta de pré-processamento baseada em Python é desenvolvida para a geração automatizada de diferentes modelos. Numa segunda parte, a ferramenta desenvolvida é utilizada com sucesso a fim de realizar uma série de estudos numéricos sobre diferentes abordagens de modelação para uma aeronave de transporte genérica em escala real. Técnicas de modelação da água com o objetivo de reduzir o esforço computacional são investigadas. Para a aeronave, diferentes esquemas de modelação variam de simples abordagens de corpo rígido a modelos detalhados. Uma opção de stress de separação é testada para simular forças de sucção entre a fuselagem e o domínio da água. Finalmente, sistemas computacionais de processamento de memória compartilhada e Multi Model Coupling são considerados. As conclusões finais são delineadas em termos de precisão e eficiência das técnicas de estado da arte investigadas.

Palavras chave: Smoothed Particle Hydrodynamics, Interacção Fluído-Estrutura, Amaragem de Aeronaves de Asa Fixa, Impacto em Água
Contents

List of Symbols

List of Figures III

List of Tables VI

1 Introduction 1
   1.1 Motivation .............................................. 1
   1.2 Objectives and Structure of the Thesis .................... 2

2 Background on Fixed-Wing Aircraft Ditching Analysis 5
   2.1 Airworthiness Regulations ................................ 5
   2.2 Acting Forces upon Impact ................................ 7
   2.3 Current Research Activities ................................. 9

3 Coupled Finite Element-Smoothed Particle Hydrodynamics Formulation 15
   3.1 Smoothed Particle Hydrodynamics (SPH) Method ............ 15
      3.1.1 Fundamentals ........................................ 15
      3.1.2 Governing Equations for Fluid Domain ................. 18
      3.1.3 Coupling with Finite Elements ....................... 20
   3.2 State of the Art on Coupled FE-SPH Simulations of Impact on Water 21
   3.3 Critical Review and Definition of the Thesis ............. 23

4 Tool for Automated Model Generation 25
   4.1 Introduction ............................................. 25
      4.1.1 Development Methodology ............................. 27
      4.1.2 User Input and Graphical Interface .................. 28
   4.2 Aircraft Model Generation ................................ 29
      4.2.1 General and Detailed Finite Element Models ........... 30
      4.2.2 Rigid Body Models ................................... 31
      4.2.3 Dynamic Master Models ............................... 32
      4.2.4 Wing and Empennage Model ............................ 34
   4.3 Water Model Generation ................................... 35
      4.3.1 Classic Approach and Particle Packing ............... 36
      4.3.2 Translating Periodic Boundary Conditions ............ 37
      4.3.3 Active Box via Nodes .................................. 38
   4.4 Modeling of Interaction Between Fluid and Structure .... 39
   4.5 Computational Aspects .................................... 40
   4.6 Final Considerations ..................................... 42
# List of Symbols

## Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAST</td>
<td>Crashworthiness of Helicopters on Water (EU project)</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>COG</td>
<td>Center of Gravity</td>
</tr>
<tr>
<td>CRAHVI</td>
<td>Crashworthiness of Aircraft for High Velocity Impact (EU project)</td>
</tr>
<tr>
<td>CAST</td>
<td>Crashworthiness of helicopter on water: design of structures using advanced simulation tools (EU project)</td>
</tr>
<tr>
<td>CSD</td>
<td>Computational Structural Dynamics</td>
</tr>
<tr>
<td>DFEM</td>
<td>Detailed Finite Element Model</td>
</tr>
<tr>
<td>DLR</td>
<td>German Aerospace Center (German: Deutsches Zentrum für Luft- und Raumfahrt e.V.)</td>
</tr>
<tr>
<td>DMM</td>
<td>Dynamic Master Model</td>
</tr>
<tr>
<td>DMP</td>
<td>Distributed Memory Processing</td>
</tr>
<tr>
<td>EASA</td>
<td>European Aviation Safety Agency</td>
</tr>
<tr>
<td>EOS</td>
<td>Equation of State</td>
</tr>
<tr>
<td>EU</td>
<td>European Union</td>
</tr>
<tr>
<td>FAA</td>
<td>Federal Aviation Agency</td>
</tr>
<tr>
<td>FAR</td>
<td>Federal Aviation Regulations</td>
</tr>
<tr>
<td>FDM</td>
<td>Finite Difference Method</td>
</tr>
<tr>
<td>FE</td>
<td>Finite Element</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>FSI</td>
<td>Fluid-Structure Interaction</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite Volume Method</td>
</tr>
<tr>
<td>GFEM</td>
<td>Global Finite Element Model</td>
</tr>
<tr>
<td>JAR</td>
<td>Joint Aviation Regulations</td>
</tr>
<tr>
<td>MDO</td>
<td>Multi Disciplinary Optimization</td>
</tr>
<tr>
<td>MMC</td>
<td>Multi Model Coupling</td>
</tr>
<tr>
<td>RB</td>
<td>Rigid Body</td>
</tr>
<tr>
<td>RBM</td>
<td>Rigid Body Model</td>
</tr>
<tr>
<td>SMAES</td>
<td>SMart Aircraft in Emergency Situations (EU project)</td>
</tr>
<tr>
<td>SMP</td>
<td>Shared Memory Processing</td>
</tr>
<tr>
<td>SPH</td>
<td>Smoothed Particle Hydrodynamics</td>
</tr>
<tr>
<td>SPSTR</td>
<td>Separation Stress</td>
</tr>
<tr>
<td>SPTHK</td>
<td>Separation Thickness Factor</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Meaning</td>
</tr>
<tr>
<td>--------------</td>
<td>----------------------------------------</td>
</tr>
<tr>
<td>TPBC</td>
<td>Translating Periodic Boundary Conditions</td>
</tr>
<tr>
<td>TSR</td>
<td>Time Step Ratio</td>
</tr>
<tr>
<td>VPS</td>
<td>Virtual Performance Solutions (explicit solver)</td>
</tr>
</tbody>
</table>

**Latin Symbols**

<table>
<thead>
<tr>
<th>Symbols</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c$</td>
<td>Speed of sound</td>
</tr>
<tr>
<td>$e$</td>
<td>Internal energy</td>
</tr>
<tr>
<td>$E$</td>
<td>Young’s modulus</td>
</tr>
<tr>
<td>$f$</td>
<td>Continuous field function</td>
</tr>
<tr>
<td>$h$</td>
<td>Smoothing length</td>
</tr>
<tr>
<td>$h_{cont}$</td>
<td>Contact thickness</td>
</tr>
<tr>
<td>$i, j$</td>
<td>Particle notation</td>
</tr>
<tr>
<td>$I$</td>
<td>Inertia</td>
</tr>
<tr>
<td>$L$</td>
<td>Length</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
</tr>
<tr>
<td>$N$</td>
<td>Set of neighboring particles</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$r$</td>
<td>Position vector</td>
</tr>
<tr>
<td>$S$</td>
<td>Integral surface</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$v$</td>
<td>Velocity</td>
</tr>
<tr>
<td>$W$</td>
<td>Smoothing kernel</td>
</tr>
</tbody>
</table>

**Greek Symbols**

<table>
<thead>
<tr>
<th>Symbols</th>
<th>Designation</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha, \beta$</td>
<td>Coordinate directions</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Dirac delta function</td>
</tr>
<tr>
<td>$\delta_{cont}$</td>
<td>Penetration depth</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>Shear Strain</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Adiabatic exponent</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>Scaling constant of smoothing function</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic Viscosity</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Stress</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Isotropic stress</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>Integration domain</td>
</tr>
</tbody>
</table>
List of Figures

1.1 Ditching of US Airways Flight 1549 in the Hudson River on January 15, 2009. ...................... 1

2.1 Phases of ditching [10]. ................................................................. 6
2.2 Differences in load transfer for impacts on rigid surfaces and water [14]. ......................... 7
2.3 Exemplary suction and overpressure regions originating in fixed-wing aircraft ditching
    (force amplitude and extent not to scale) [20]. .................................. 8
2.4 EADS-CASA CN235-300M Experimental Ditching Campaign [11]. .............................. 10
2.5 Excessive mesh distortion with Lagrangian modeling approach for water domain [27]. .... 11
2.6 Qualitative validation of SPH model with images from experimental test of wedge with a
    25 degree deadrise angle, at t= 0.01 s (left) and t= 0.02 s (right) [13]. ................. 12
3.1 Particle approximation in problem domain $\Omega$ with a surface $S$ [37]. ......................... 17
3.2 Penalty contact scheme. ........................................................................... 20
3.3 Simulation of the half model WG30 subfloor structure impacting on water using a combined
    FE-SPH for the fluid domain [12]. ......................................................... 21
3.4 Importance of suction modeling by comparison of the simulated kinematics of an Airbus
    Military CN235 rigid sub-scale model [49]. ........................................ 22

4.1 Examples of VPS input format ............................................................... 25
4.2 Overview of the code of the developed pre-processing tool (emphasis on modular setup). .... 27
4.3 Examples of graphical interface of the developed pre-processing tool. .............................. 28
4.4 Computation scheme of available aircraft modeling approaches. .................................. 29
4.5 Examples of refinement in the ditching impact zone. ................................................. 30
4.6 Scheme of condensation of RBM from AC-CRASH FE model. ...................................... 31
4.7 Example of computations for the generation of the Rigid Body Model. .......................... 31
4.8 Scheme for Stiffness Computation .................................................................... 32
4.9 Nodal and beam mass scheme for DMM model. ..................................................... 33
4.10 Dynamic Master Model computations scheme ...................................................... 34
4.11 Example of Dynamic Master Model approach. ..................................................... 34
4.12 Scheme of wing and empennage attachment (in red) for GFEM and DFEM approaches. . 35
4.13 Scheme of wing and empennage attachment (in red) for DMM models. ...................... 35
4.14 Example of classic FE-SPH approach for the water domain. ...................................... 36
4.15 Available SPH packing schemes. Images from [61]. ............................................. 36
4.16 Exemplary scheme of Periodic Boundary Conditions. ........................................... 37
4.17 Example of Active Box via Nodes .................................................................... 38
4.18 Options for contact between aircraft and water domain [48]. ................................... 39
4.19 Schematic architecture of parallel processing methods, [64, 65] .................................. 40
4.20 Scheme of Multi Model Coupling Method [67] ..................................................... 41
4.21 File structure for simulation models to be used with MMC. 41
4.22 File structure for simulation models to be used with SMP or standard DMP. 42

5.1 Scheme of Rigid Body model used in water modeling investigations. 46
5.2 RB aircraft model and full length FE-SPH option for water basin. 47
5.3 Simulation over 1000 ms using RB aircraft model and full length FE-SPH water basin. 47
5.4 Height and width dimensions of the pool for investigations concerning pool size. 48
5.5 Comparison of the acceleration time histories of the COG of the aircraft model with different dimensions of water pool cross-section (CFC180). 48
5.6 Influence of FE to SPH element size ratio and contact between both (CFC180). 49
5.7 Translating periodic boundary conditions options, without (left) and with (right) shell container, SPH modeling of water domain. 50
5.8 TPBC without shell container: acceleration histories of different SPH domain dimensions. 51
5.9 TPBC with shell container: acceleration histories of different SPH domain dimensions. 51
5.10 Translating Periodic Boundary Conditions with FE water model. 52
5.11 TPBC SPH domain with FE hydrodynamic volumes compared to the classic full-length approach for the water (CFC180). 53
5.12 Fixed active box SPH domain with FE hydrodynamic volumes compared to the classic full-length approach for the water (CFC180). (CFC180). 54
5.13 Adaptive Active Box over simulation (from left to right, t = 0 ms, t = 45 0ms and t = 900 ms).
In the water pool: hydrodynamic FE volumes (light blue), activated particles (dark blue), deactivated particles (green). 55
5.14 Comparison of COG acceleration histories for full-length pool and adaptive active box options (CFC180). 55
5.15 Particle refinement investigations. Different particle spacings (CFC180). 56
5.16 Particle refinement investigations. Different particle spacings with and without time step constraint (CFC180). 57
5.17 Contact force time histories in Z-direction of rigid body fuselage with rigid and flexible wing/empennage (CFC180). 60
5.18 Dynamic master model used in aircraft model investigations, with 87 length-wise sections. 61
5.19 Contact force time histories in Z-direction of DMM and RB models (CFC180). 61
5.20 Deformation of DMM at t = 120 ms. Structural deformation is amplified in all directions by a factor of 200. Beam elements and water basin are not displayed for clarity purposes. 62
5.21 Contact force time histories in Z-direction of DMM with rigid and flexible wing (CFC180). 62
5.22 GFEM fuselage with coarser and finer meshes for aircraft modeling investigations (frames highlighted in red). 63
5.23 GFEM with coarser mesh: displacement in Z-direction | Zoom on impact region for t = 390 ms. 64
5.24 GFEM with coarser mesh: displacement in Z-direction | Zoom on impact region for t = 390 ms. 64
5.25 Contact force time histories in Z-direction of GFEM with coarser and finer meshing (CFC180). 65
5.26 Displacement in Z-direction for the DFEM approach after t=390 ms. 66
5.27 Contact force time histories in Z-direction for DFEM approach (CFC180). 66
5.28 Contact force time histories in Z-directions for RBM, DMM and DFEM models (CFC180). 67
5.29 Pitch angle time histories for RBM, DMM and DFEM models (CFC180). . . . . . . . . . . 68
5.30 Separation stress investigations: simulation with SPSTR=320 GPa and SPTHK=1.75. . . 69
5.31 Separation stress investigations: simulation without separation stress feature activated
(left) and with SPSTRS=1E3 GPa, SPTHK=1.75 (right) for 300 mm particle spacing. . . 70
5.32 MMC investigations: GFEM model with time step ratios TSR=30 and TSR=5. . . . . . . 71

A.1 Example of CPACS file in XML editor. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 83
B.1 Computed stiffness of GFEM approach, transferred to DMM model. . . . . . . . . . . . 85
List of Tables

4.1 Aircraft modeling capabilities of the tool and compatible VPS version (see §4.2). 26
4.2 Water modeling capabilities of the tool and compatible VPS version (see §4.3). 26
4.3 Computational processing capabilities of the tool and compatible VPS version (see §4.5). 26

5.1 Mass and inertial properties of Rigid Body model used in water modeling investigations. 46
5.2 Comparison of computational time with different dimensions of water pool cross-section. 48
5.3 TPBC without shell container: computational time with different sizes of SPH domain. 50
5.4 TPBC with shell container: computational time with different sizes of SPH domain. 51
5.5 Particle refinement investigations: computational time with different particle spacings (dx). 56
5.6 Particle refinement: computational time with different particle spacings and time step constraint. 57
5.7 Aircraft modeling investigations: mass, COG and inertia comparison (percentage is given in comparison with reference values shown in Table 5.1). 59
5.8 Rigid body investigations: computational times of rigid and flexible wing options. 60
5.9 Dynamic master model investigations: computational times of RB and DMM simulations. 61
5.10 DMM investigations: computational times of rigid and flexible wing options. 63
5.11 Global finite element investigations: computational times of rigid and flexible wing options. 65
5.12 Computational times for RBM, DMM and DFEM models. 67
5.13 Data concerning separation stress investigations in [48]. 69
5.14 Computational times for SMP (No MMC) and MMC computational schemes with TSR=30 and TSR=5. 72
1 Introduction

1.1 Motivation

On January 15, 2009, the 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 and being forced to pursue an emergency landing. The pilot told his passengers to brace for a hard impact and then set the plane down safely on the Hudson River. With the Manhattan skyline as background, all 155 passengers and crew safely evacuated the plane. More than the "Miracle on the Hudson", 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 [1, 2].

![Evacuation of the occupants](image1.jpg)

(a) Evacuation of the occupants [3].

![Recovered airframe](image2.jpg)

(b) Recovered airframe [4].

Figure 1.1: Ditching of US Airways Flight 1549 in the Hudson River on January 15, 2009.

In fact, since many airports are close to water and many flights are operating partly over water, planned water landing is a possibility that could be encountered by most aircraft. Even tough this is a very rare event, regulations address ditching in the certification of aircraft operating over sea. The manufacturer must show compliance to the specific airworthiness regulations, from structural aspects to emergency equipment: the study of water impact is of great importance to designers and engineers.

New aircraft undergo a range of analyses and tests to show that a water landing event is feasible without critical damage and injury to the occupants, and that the aircraft will subsequently float for a time compatible with evacuation [5]. In addition to calculations verifying sufficient structural strength, tests with scale models ditched in a water tank are performed when necessary to evaluate the general behavior of the aircraft during a water landing, and to investigate optimum approach conditions.

The reason why most crashworthiness testing in aircraft design has been experimentally based is due to the complex nature of the fluid-structure interaction (FSI) phenomena involved, in particular due to the high forward velocity of the impacting aircraft. The challenging physical problem involves a wide range of technical disciplines, such as aerodynamics, hydrodynamics and structural dynamics.
However, test campaigns are expensive and time-consuming, and thus the aerospace industry is now turning its attentions to numerical tools to assist in ditching investigations. The increase in computer power and the development of numerical models contribute to the potential of such assessments, along with the prospect of reduced costs and faster evaluations of novel designs. Several advances have been made in this field over the years, yet the simulation of aircraft ditching still presents a very interesting computational modeling challenge.

In February 2011, the project SMAES (Smart Aircraft in Emergency Situations) was initiated with the goal to develop a set of simulation tools to permit cost effective design and entry-into-service of aircraft able to protect its occupants during a water landing. The project covered two main research areas: numerical prediction of the ditching loads; and predictive aircraft models that incorporated dynamic structural behavior while being coupled with hydrodynamic models [6]. The work was co-funded by the European Commission and brought together both industry leaders and research institutions, e.g. Airbus Military, the ESI-Group and the German Aerospace Center (DLR). This project represents the starting point of the present thesis.

1.2 Objectives and Structure of the Thesis

During the course of the SMAES project, investigations were conducted to simulate aircraft ditching using a particular numerical approach which combines classic Finite Element (FE) and Smoothed Particle Hydrodynamics (SPH) formulations. Within this research, the hybrid FE-SPH module of the VPS explicit solver (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 is developed for automated model generation, and a comparison of different modeling approaches is performed in terms of quality of the results and required computational effort. Accuracy and efficiency are forefront in order to provide an improvement in the predictability of loads occurring during ditching. The thesis has the following structure:

- **Chapter 2** provides the context for the current research. Importance of ditching is highlighted by summarizing relevant airworthiness regulations. Particularities of the fluid-structure interaction upon impact are described, with emphasis on the hydrodynamic effects arising from the presence of high forward velocity of the impacting structure. Finally, current research works are described, focusing on numerical investigations. The motivation for applying a combined finite element-smoothed particle hydrodynamics method to this problematic is given.

- **Chapter 3** proceeds to further deepen the FE-SPH formulation used in this research. Theoretical fundamentals on discretization of the governing equations with the SPH method are summarized. The coupling algorithms to couple this method to classic FE are briefly described. Finally, a review on the state of the art and an identification of open questions follows, closing with a detailed definition of the objectives of the present thesis.
• **Chapter 4** provides an overview on the computational pre-processing tool developed for automated model generation and constitutes one of the main contributions of this work, as it is a fundamental requirement to allow for comprehensive investigations. The tool focuses on generation of input for the VPS explicit solver, aiming to incorporate new features and different pre-existing tools into one automated chain. Several advanced modeling techniques concerning the aircraft structure and the water domain are presented, and different computational arrangements are explained. A discussion on the final outcome of the developed tool is given at the end of the chapter.

• **Chapter 5** uses the tool developed in Chapter 4 to investigate the potential of the different modeling features available, presenting the results and discussing them in terms of accuracy and computational efficiency. A generic full-scale transport aircraft model is used for these investigations.

• **Chapter 6** summarizes the main results of the work and discusses the different achievements. Suggestions for further improvements are listed in the Future Work section.
2 Background on Fixed-Wing Aircraft Ditching Analysis

This background review analyses how aircraft ditching is currently regarded by researchers and industry. In the first part, ditching is described as an aircraft emergency situation which ends with the planned landing on water. Transport aircraft must satisfy several requirements if they are to be certified for over-water operations. A summary of the relevant ditching regulations is given here, based on the three phases of planned water landing. Moreover, the impact phase is defined as subject of the present research.

High hydrodynamic loads on the structure are present during water impact. In addition, the presence of a high forward velocity in fixed-wing aircraft ditching affects the pressure distribution and the interrelated hydrodynamic effects acting on the fuselage. The referred phenomena are explained and their relevance as key features of this hydrodynamic impact is highlighted.

In the last section, the methods currently in use to show compliance to ditching regulations are identified. An overview of the numerical modeling techniques commonly used for water crashworthiness simulations is presented in the last part, outlining the results of research in this field related to aerospace structures.

2.1 Airworthiness Regulations

Transport aircraft water impacts are classified into two basic categories: planned and unplanned water contact. As described by the certification regulations, ditching is a planned water impact, an event where the flight crew intentionally lands the aircraft on a body of water such as in the open ocean, a lake or a river. When proper ditching procedures are followed, the occupants should have enough time to prepare for the impact, and the pilot should maintain substantial control of the aircraft. In contrast, when the impact on water is unplanned, the crew is not aware of the imminent crash. Runway overruns, due to landing gear failure or poor weather conditions, are an example for the latter. This very frequently causes higher impact loads and subsequently higher probability of severe injury levels.

In order to improve the crashworthiness in a water environment, both ditching and unplanned water impact accidents should be considered. As the focus of this research is planned water landing, no further review will be conducted on the referred unplanned events. Investigations on the latter type of impacts can be consulted in [7] or [8].

For the past decades, the assessment of the safety standard for certification has been emerging in the crashworthiness area and is being extended to the specific problem of planned impact on water. Specifications concerning ditching are compiled e.g. in the Federal Aviation Regulations (FAR), in the European Joint Aviation Regulations (JAR) and in the Certification Specifications (CS) of the European Aviation Safety Agency (EASA). The regulations include design guidelines, equipment requirements and evacuation procedures which are intended to allow maximum passenger survivability.
As crew and passengers should have time to prepare for the impact, it is necessary that the touchdown follows a prudent approach and acceptable procedures. If these procedures are followed then, according to the certification regulations, the airframe must [9]:

- Provide structural integrity to protect all occupants;
- Ensure that no excessive decelerations will be experienced by the occupants;
- Provide sufficient time for safe evacuation from the damaged aircraft.

To fulfill the above mentioned requirements, three main stages are accounted for in a complete ditching evaluation: aircraft conditions before impact (approach phase), structural response during the impact (impact phase), and the subsequent flotation phase, as shown in Figure 2.1.

![Approach Impact Flotation](image)

**Figure 2.1:** Phases of ditching [10].

The approach phase covers the initial conditions for the impact scenario. The kinematics of the aircraft during the impact on water strongly depend on its configuration at the moment of the event, and so the manufacturer is required to thoroughly investigate factors such as flight path angle, pitch attitude and aircraft speed to identify optimum approach conditions. Aircraft mass properties regarding weight, center of gravity (COG) and inertia as well as the landing gear position and engine thrust are also of relevance. Simultaneously, external factors such as wind or sea state may be considered. Under reasonable environmental conditions, recommendations should be made by the manufacturer and incorporated into the airplane’s cabin crew manual of emergency procedures [9].

The present thesis is mainly devoted to the study of the impact phase. At this stage, the high pressures derived from the impact yield local deformations along the fuselage and may cause rupture of the structure, putting at risk the safe evacuation of the occupants [11]. In accordance with the regulations, the airframe must be designed in order to maintain as far as possible these crash forces below the level of severe injury of the occupants. External doors and windows must be designed to withstand the maximum load pressure in order to avoid immediate water ingress in the cabin. Finally, direct structural deformation of the passenger space is to be avoided, to allow the the crew and the passengers to reach as easily as possible the emergency exits.

The evacuation of the passengers and crew of the aircraft takes place within the flotation phase. Airworthiness regulations address this phase by stating requirements for fuselage buoyancy, flotation time and aircraft trim with respect to the local sea conditions which are likely to occur. The number of exits should be sufficient and their size adequate to allow a rapid evacuation of the airplane. In addition, for various COG positions and flooding conditions, the position of the exits relative to the waterline and the time each exit remains above water must be analyzed [9, 12].

In summary, each design measure, compatible with the general characteristics of the airplane, must be taken to minimize the probability of immediate injury to the occupants or the impossibility of escape. As the forces acting on the body upon impact are determinant for the integrity of the aircraft structure, further review will be focused on the impact phase.
2.2 Acting Forces upon Impact

Considering impact of aeronautical structures on water, it is useful to first assess the force mechanisms associated with the physical problem. The aim is to have a better understanding of the complexity of the event, and not to give an exhaustive list of all factors involved.

**Vertical Impacts**

The structural behavior for impact onto rigid surfaces such as hard ground is different to that required to absorb the loading due to impact on water. The difference is exemplified for vertical impact in Figure 2.2. When crashing onto rigid surfaces, the impact loads are transferred to the frames and beams which absorb significant energy through controlled deformation. When crashing into water, the impact loads are distributed along the skin panels, which must transmit them to the energy absorbing structural components. Typical skin panels are not designed to withstand out-of-plane impulsive loads, and therefore may rupture easily during a water impact [13].

![Diagram showing differences in load transfer for impacts on rigid surfaces and water](image)

(a) Impact on hard surface (load concentrated towards stiff structural elements).  
(b) Impact on water (uniformly distributed load).

**Figure 2.2:** Differences in load transfer for impacts on rigid surfaces and water [14].

In this event, inertial forces are preponderant, as the high loads acting on the structure are primely caused by the reaction of accelerating a certain mass of fluid which moves with the body. Von Kármán [15] and Wagner [16] were the pioneers in determining an analytical estimate of this so-called induced mass, and therefore of the impact force, proposing solutions for wedge-shaped rigid bodies. The basis of these theories was conservation of momentum, as the momentum lost by the impactor is converted into momentum of the induced mass of water. The induced mass is described as the mass of water which would have the same kinetic energy as that of the flow if it traveled with the body speed. Historically, these works were the basis of estimation of the forces applied to seaplanes at landing.

Other than the described inertial forces, additional factors may play a role in the event of water impact. For the vertical impact of aerospace structures, a short study in [12] shows that in a first approximation viscous forces of the water, its compressibility and surface tension effects can be seen as negligible for the analysis of load transfer. The influence of the layer of air between the impactor and the free water surface has also been object of study for bodies impacting on water (namely in [17] and [18]), as it may cause a cushioning effect. The latter, commonly named as air entrapment, may be particularly important when flat or nearly flat-bottomed bodies strike the water surface.

**Oblique Impact**

The presence of a high forward velocity in fixed-wing aircraft ditching events highly affects the pressure distribution and the hydrodynamic effects acting on the fuselage, adding to the complexity of the problem.
Two states of motion, both leading to high accelerations and loads, are feared in aircraft ditching: "nosedive", i.e. the nose of the aircraft submerges and the aircraft overturns, and "skipping", the repeated water contact [10]. The latter may lead to a loss of control of the aircraft, resulting in uncontrolled motion and possibly higher accelerations. To describe the full aircraft dynamics correctly, flight aerodynamic has to be accounted for. Furthermore, in addition to the physical phenomena described for vertical water entry, significant fluid-structure interaction effects include suction, overpressure, cavitation and ventilation, described below.

The pressure gradient along the wetted fuselage is strongly affected not only by the horizontal velocity component of the aircraft, but also by its shape. These two factors govern the intensity of the suction effects and can either result in distinct overpressure (positive pressure), or suction (relative negative pressure) regions. Suction forces typically arise in the rear part of a conventional fuselage, caused by an acceleration of the water flow around the impacting structure, leading to a drop of the water pressure in the region. Regions where the velocity is lower due to the shape of the impacting body, lead to overpressure. These forces can be high enough to change the attitude of the aircraft and therefore its kinematics.

If the water pressure decreases under the vapor pressure, cavitation (state at which vapor or gas- or vapor-filled bubbles, named cavities, appear in a liquid) occurs. It is also presumed that the presence of air in the impact region has an influence on the loading acting on the structure. When the structure is moving forwardly, air may be fed into the water, which changes the properties of the continuum at this point. This effect is named by the literature as ventilation or aeration.

The pressure field due to hydrodynamic forces is studied in [19], showing clear regions of suction and overpressure measured by pressure transducers in the fuselage of a scale model. The phenomenon is represented in Figure 2.3. Overpressure results in nose-down pitch moment, whereas suction leads to a nose up pitching moment. Even a small suction region can cause a change of global kinematics, due a much larger lever arm to the center of gravity compared to the overpressure region.

**Figure 2.3:** Exemplary suction and overpressure regions originating in fixed-wing aircraft ditching (force amplitude and extent not to scale) [20].

The above factors highlight the need to develop optimized features for both rigid and water surface impact, through the use of high performance energy absorbing materials and alternative structural concepts. Also, fixed-wing ditching presents a very different set of phenomena due to the high forward velocity of the impactor, which are not encountered during a vertical impact on water as in helicopter accidents.
2.3 Current Research Activities

Most water impact testing in aircraft design has been experimentally based due to the complex nature of the fluid-structure interaction phenomena involved. Currently, to substantiate ditching behavior of a new aircraft model, airworthiness regulations state that the manufacturer should emphasize a combination of accident data review, comparison with previous designs and model tests. However, during the last decades numerical simulation has experienced a great development, and therefore it may decisively contribute for determination of test conditions.

A summary description and corresponding advantages and disadvantages of each of current analysis methods is presented below.

Comparison with airplanes of similar configuration

One method to show compliance with imposed ditching requirements is to use for comparison the hydrodynamic characteristics of prior certified designs and/or to evaluate ditching accidents involving comparable aircraft [12]. The manufacturer may do this by showing that the design is similar in both geometry and size to existing designs which have already demonstrated satisfactory hydrodynamic behavior. It can also be shown that the additional features of the new structure are only beneficial to the ditching behavior compared to the existing aircraft. For example, if the new aircraft has a similar design in fuselage geometry and size but a longer wing span, it can be proven that the new design is expected to provide additional buoyancy, therefore providing better flotation characteristics [9].

There are a number of constraints in the analysis of aircraft ditching accidents. First, ditching is a very rare event and occurs for different aircraft under different circumstances, which means no meaningful statistical evaluation is possible. Other than this, not necessarily all the material gathered during the official accident investigation is released, due to confidentiality constraints. Finally, the accident analysis offers little insight into the physics of ditching and thus makes it difficult to reconstruct a cause-and-effect chain [10].

For some novel aircraft with very unusual design features, comparison with previous airplanes may not be possible. While this analysis is confined to similarly designed aircraft under special conditions, ditching model tests and numerical simulations allow to investigate aircraft ditching performance during the design phase considering the influence of various parameters.

Model testing

The manufacturer may perform ditching analysis through experimental model testing. Full scale models have been used before (e.g. the full-scale drop tests performed on the Hudson and Botha rivers by the British Marine Aircraft Experimental Establishment, 1941-1942, in [21] and [22]). However, for economical and also practical reasons, nowadays the use of aircraft scaled models launched in a water tank is more common when investigating various ditching conditions. This is particularly interesting since the airworthiness authorities recognize model test results to prove the kinematic behavior, the flotation and the trim characteristics of a full-scale aircraft.

The scaled models are built from design data sent by the manufacturer and accurately represent the aircraft’s external shape, being designed with access panels on their sides to enable equipment (e.g. accelerometers, strain gauges, pressure transducers) to be fitted. For the test, a rigid model is mostly launched as a free body by means of a catapult and has a short free flight before contacting the water surface (guided model tests are also possible, in which one or more degrees of freedom are restricted).
Damage may be taken into account by incorporating scale strength elements, which fail when a specified load level is reached. This aims to represent the damage of particularly sensible parts of the aircraft (e.g. engine mountings), which may have an influence on its further behavior during the event. Ditching conditions are varied by modifying pitch, roll and yaw angles, velocity, COG position, and possible sea states (calm and rough sea) [12]. An example showing the ditching campaign performed by EADS-CASA in early 2004 for certification of the military aircraft CN-235-300M [11] is given in Figure 2.4.

Years of experience and accreditation by the authorities are the main advantages of this analysis approach. However, model production and appropriate testing facilities are still considerably expensive, and thus only a limited number of geometries can be tested during the design stage of the aircraft. Other than this, ensuring precise motion in flight in free-flight tests is often challenging, leading to repeatability issues.

Another well known disadvantage of scaled model testing arises from scale effects. Because of the dominance of inertia forces and the gravitational force at this impact situation, all known ditching tests are carried out considering Froude scaling for the model properties and impact conditions. As only this scaling is applied, scale effects due to the other involved scaling laws are introduced. Certain hydrodynamic effects such as cavitation are not currently portrayed due to the violation of the respective scale law, which compromises the direct scaling of loads and aircraft motion between model and full-scale conditions. The latter becomes particularly relevant for aircraft with pronounced longitudinal curvature and high forward velocity [10].

Numerical simulations, on the other hand, allow for the investigation of different designs and impact conditions without additional cost, and overcome scale effects.

**Numerical Simulation**

Numerical ditching analysis ranges from simple analytical water impact formulas to numerical simulation methods taking into account the complete aircraft and the near field of the surrounding fluids.

Simple numerical methods and semi-empirical formulas form hybrid techniques, based on known results from tests or analytical studies for simple bodies. Notable examples are the DRI-KRASH code [23], modified under FAA sponsorship, or the DITCHER code [24], developed by D2M for Eurocopter France. More recently the hybrid tool "Ditch" has been developed by the Hamburg University of Technology, and is compared to experimental data in [10].

Computational numerical methods, on the other hand, rely on the discretization of the intricate differential and integral equations that describe the physical problem. Examples of disciplines that make use of such methods are Computational Structural Dynamics (CSD), or Computational Fluid Dynamics (CFD).
The present thesis focuses on a particular computational method, and as such forthcoming reviews will focus on this category.

Simulation of fluid-structure interaction is complex, as the ideal code must be able to handle non-linearities and predict thin walled structural collapse, in addition to capturing the physical response of water. Efforts to achieve some of these ideals have been made, using both Eulerian and Lagrangian grid-based formulations. These latter differ from each other due the reference coordinate system in which the equations of motion are written, as explained in [25]:

- Eulerian algorithms are widely used in fluid dynamics. Here, the computational mesh is fixed and the continuum moves with respect to the grid. Examples are the Finite Difference Method (FDM) or the Finite Volume Method (FVM);

- Lagrangian algorithms, in which each individual node of the computational mesh follows the associated material particle during motion, are mainly used in structural mechanics. One example of this description is the Finite Element Method (FEM);

- One can also refer to the Arbitrary Lagrange-Euler (ALE) technique. In this method, an arbitrary referential coordinate system is introduced, and features of both the Lagrangian and the Eulerian approaches are combined. The mesh is allowed to displace and the material particles can be displaced through the moving grid.

In aircraft ditching simulations, it is necessary to ensure the compatibility between the methods used to model the water medium and the common structural models, namely FE models available in the different aeronautical disciplines (e.g. static and dynamic load analysis). The structure deformations are commonly computed with the classical finite element method due to its proved robustness. The difficulties remain on the one hand in the computation of the fluid flow and on the other hand in the coupling between the structure and the fluid in order to compute interactions.

For the simulations performed using the conventional FE formulation for the water, large mesh distortions were observed as the impact event progressed. These lead to significant reduction in the solution time step and further numerical instabilities occur. Despite this, good experimental/numerical correlation was observed for the initial moments after impact, prior to the occurrence of mesh instabilities at the fluid-structure interface [12, 26, 27]. Although computationally efficient, the FE approach was found to be suitable only for the initial stages of the water impact, prior to the onset of excessive mesh distortion. An example of this effect is given in Figure 2.5 for better visualization of the problem.

![Figure 2.5](image.png)

**Figure 2.5:** Excessive mesh distortion with Lagrangian modeling approach for water domain [27].
Techniques coupling different formulations have also been tested. They involve the coupling (known as the Euler/Lagrange coupling) between the finite volume method for the fluid (Eulerian formulation) and the finite element method for the structure (Lagrangian formulation). With the Eulerian description of the water, the deformation of the water was well captured [26, 28]. Better stability was observed than the Lagrangian approach above and very good correlation with experimental test results was observed [26]. However, the very long computational times when using an entirely Eulerian approach would prove impractical for full-scale ditching models. Also, the complex coupling algorithm between formulations present a key issue as it is responsible for the transmission of the contact forces and/or displacements between the fluid and the structure.

The ALE representation of the water domain generally resulted in good experiment-simulation correlation [29, 30]. Even for an appreciable mesh density, the model solution times were less than those of a purely Eulerian formulation. Nonetheless, the ALE simulation results were highly mesh dependent and certain instabilities occurred in the models with poor mesh distributions [29]. Even so, ALE has shown strong potential for water impact simulations, and is favored by many researchers for efficient modeling of highly deformable domains.

In summary, 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. In the SPH method the continuum is represented by a set of particles, each carrying individual mass. An interpolation function of the neighboring particles is used for the computation of field properties. Exemplary of a simulation using SPH for the water domain in a water impact analysis is Figure 2.6.

![Figure 2.6: Qualitative validation of SPH model with images from experimental test of wedge with a 25 degree deadrise angle, at t= 0.01 s (left) and t= 0.02 s (right) [13].](image)

From the different strategies for water modeling in investigations related to aircraft ditching, ALE and SPH have demonstrated the best potential. Yet, ALE still displays high mesh dependence and numerical stability due to mesh consistency is not ensured. By contrast, in SPH mesh distortion is overthrown. One other significant advantage of the latter is the simplified contact between FE structural aircraft models and the water media, 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.

Following the literature review, two Lagrangian techniques are taken in this research work. To model the water medium, classic FE volumes are used for the regions with less disturbance, and SPH particles are employed in the area closer to impact. The impacting structure is modeled with the classic FE method, in the same way as standard crash simulations.
Due to the specificity of the SPH method, which is not as well known as the classic FE technique, some theoretical background is given in the following chapter (Section §3.1). Applications of the FE-SPH approach are subsequently reviewed in Section §3.2.

For the problems of interest in this work, the explicit software VPS from ESI-Group is used. The code provides the ability to simultaneously use FE and SPH within the same model, and has proven its capacity to treat nonlinear dynamic crash analysis of structures (namely for the automotive industry). Other than this, state-of-the-art features have been recently implemented which are of interest for the present investigations.
3 Coupled Finite Element-Smoothed Particle Hydrodynamics Formulation

The previous chapter provided the framework for the Smoothed Particle Hydrodynamics approach to simulate water in aircraft ditching events. Here, key principles will be first reviewed in order to provide some background in the formal description of the SPH method. The discretization of the governing equations for the water domain according to the SPH formulation is then presented. Finally, the principles of algorithms in use for coupling FE models and the water media for fluid-structure interaction are described.

In the second part, a review of the state of the art on how this method has been applied to simulations of impact on water is presented for validation purposes. Recent improvements and code implementations are described. Finally, a discussion of the literature and identification of open questions are given, in order to provide the frame for the current thesis.

3.1 Smoothed Particle Hydrodynamics (SPH) Method

Smoothed Particle Hydrodynamics was initially developed in 1977, by Lucy [31] and Gingold & Monaghan [32], for the simulation of astrophysics problems in three-dimensional open space. Since then, many researchers have conducted investigations on the method, improving it to the state of a very powerful formulation for problems governed by the Navier-Stokes equations. Its application has extended to subjects of continuum solid and fluid mechanics, such as river hydrodynamics [33], heat conduction problems [34], underwater explosions [35] or even blood flow analysis [36].

Despite redundancy, the term SPH particle will be used instead of SPH "element" in the following text, in order to emphasize the particle character of the method in contrast to the classic finite element formulation.

3.1.1 Fundamentals

There are basically two 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 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 [37].
Kernel Approximation

The kernel approximation in the SPH method involves the representation of a function and its derivatives using a smoothing function. The kernel approximation of a function \( f(x) \) continuous in the integration domain \( \Omega \) starts from the following identity:

\[
f(x) = \int_{\Omega} f(x') \delta(x - x') dx'
\]  

Equation (3.1) implies that \( f(x) \) can be replaced by an integral form. Here \( x \) is the location where \( f \) is evaluated by interpolating its known values in \( x' \) over the domain. The Dirac delta function \( \delta(x - x') \) is given by Equation (3.2). Since the Dirac delta function is used in (3.1), the integral representation is exact, as long as the function is defined and continuous in \( \Omega \).

\[
\delta(x - x') = \begin{cases} 
1 & \text{if } x = x' \\
0 & \text{if } x \neq x'
\end{cases}
\]

In the SPH formulation, the Dirac delta function is replaced by a smoothing function \( W(x - x', h) \), over a finite spatial dimension \( h \). The smoothing function \( W \) should satisfy a number of conditions, which will be referred later in this section. Note that as long as \( W \) is not the Dirac Delta function, \( \langle f(x) \rangle \) in (3.3) can only be an approximation, named the kernel approximation:

\[
\langle f(x) \rangle = \int_{\Omega} f(x') W(x - x', h) dx'
\]  

The integral representation of the derivative of a function is obtained by substituting \( f(x) \) with \( \nabla f(x) \) in (3.3). After applying the divergence theorem to the integration domain, one obtains the result in (3.4):

\[
\langle \nabla f(x) \rangle = \int_{\Omega} \nabla f(x') W(x - x', h) dx'
\]

\[
\Rightarrow \langle \nabla f(x) \rangle = - \int_{\Omega} f(x') \nabla W(x - x', h) dx'
\]

It is clear that the differential operation on a function is transformed into a differential operation on the smoothing function. In other words, the SPH kernel approximation of the derivative of a field function allows the spatial gradient to be determined from the values of the function and the derivatives of the smoothing function \( W \), rather than from the derivatives of the function itself. Kernel approximation of higher order derivatives can be obtained in a similar way by substituting \( f(x) \) with the corresponding derivatives in (3.3).

Particle Approximation

The second step of SPH method is the particle approximation, which involves representing the problem domain using a set of particles, and then estimating field variables on this set of particles. A problem domain \( \Omega \) filled with a set of particles (see Figure 3.1 for illustration in a two-dimensional domain) can be considered. The state of the system is represented by these particles, each associated with field properties. These particles can be used not only for integration, interpolation or differencing, but also for representing the material. The volume of a subsection is lumped on the corresponding particle.
Each particle $i$ is therefore associated with a fixed lumped volume $\Delta V$ without fixed shape. If the particle mass and density are concerned, the lumped volume can also be replaced with the corresponding mass to density ratio $m_i/\rho_i$.

After representing the computational domain with a finite number of particles, the continuous form of kernel approximation expressed in (3.3) can be written in discretized form as a summation of the neighboring particles $N$ as follows:

$$\langle f(x) \rangle = \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(x_j) W(x - x_j, h),$$

(3.5)

where $N$ is the total number of particles within the influence area of the particle at $x$. It is the total number of particles that are within the support domain which has a cut-off distance, characterized by the smoothing length $h$ multiplied by a scalar constant $\kappa$. This procedure of summation over the neighboring particles is referred to as particle approximation, which states that the value of a function at a particle can be approximated by using the average of the values of the function at all the particles in the support domain weighted by the smoothing function.

![Figure 3.1: Particle approximation in problem domain $\Omega$ with a surface $S$][37].

Following the same procedure of particle approximation, the continuous integral representation of the derivative of a function $f(x)$ can be written as:

$$\langle \nabla . f(x) \rangle = - \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(x_j) \nabla W(x - x_j, h)$$

(3.6)

Equation (3.6) shows that the value of the gradient of a field function at a particle $i$ is approximated by the sum of this field function at all particles in the support domain weighted by the gradient of the smoothing function.

**Smoothing Functions**

Other than an even function usually being chosen (particles from same distance but different positions have equal effect on a given particle), there are a certain number of conditions the smoothing functions have to obey. The first is the compact condition, given in (3.7).

$$W(x - x', h) = 0 \text{ when } |x - x'| > \kappa h$$

(3.7)
where \( k \) is a constant related to the smoothing function for a particle at \( x \). Using this compact condition, integration over the entire problem domain is localized as integration over the support domain of the smoothing function.

The normalization condition states that the integration of \( W \) should produce the unity:

\[
\int_\Omega W(x - x', h) dx' = 1 \tag{3.8}
\]

The second condition is the Delta function property that is observed when the smoothing length approaches zero:

\[
\lim_{h \to 0} W(x - x', h) = \delta(x - x') \tag{3.9}
\]

This property ensures that as the smoothing length tends to zero, the approximation value tends to the function value, i.e. \( \langle f(x) \rangle \to f(x) \). Another condition states that the smoothing function value should monotonically decrease with the increase in the length: this assures that a nearer particle has a bigger influence on the particle under consideration.

A quintic Wendland kernel with radius 2h is used in this research because it was found to reduce or even alleviate the problem of standard SPH where particles tend to coincide. For more detailed reviews on the fundamentals and the development of the SPH method the reader is referred to publications by Monaghan [38] and Randles & Libersky [39].

### 3.1.2 Governing Equations for Fluid Domain

Using the approximation techniques referred before, and with the necessary mathematical operations, it is possible to derive the weakly compressible (WC)-SPH formulations for partial differential equations governing the physics of fluid flow. The basic laws for the present problem of fluid dynamics are mass, momentum and energy conservation, in (3.10), (3.11) and (3.12), respectively:

\[
\frac{D\rho}{Dt} = -\rho \frac{\partial v_\beta}{\partial x_\beta} \tag{3.10}
\]

\[
\frac{Dv_\alpha}{Dt} = \frac{1}{\rho} \frac{\partial \sigma^{\alpha\beta}}{\partial x_\beta} \tag{3.11}
\]

\[
\frac{De}{Dt} = \frac{\sigma^{\alpha\beta}}{\rho} \frac{\partial v_\alpha}{\partial x_\beta} \tag{3.12}
\]

where \( \alpha \) and \( \beta \) are used for the coordinate directions and repeated indexes stand for summations, \( v \) the velocity, \( \rho \) the density, \( x \) the position, \( \epsilon \) the internal energy and \( \sigma \) the total stress tensor. No external forces are included in (3.11).

The total stress tensor \( \sigma \) is then decomposed into an isotropic pressure \( p \) and an isotropic stress \( \tau \), as in (3.13):

\[
\sigma^{\alpha\beta} = -p\delta^{\alpha\beta} + \tau^{\alpha\beta} \tag{3.13}
\]
where $\delta^{\alpha\beta}$ is the delta function. Equations (3.10) to (3.13) are general and apply for both solid and fluid mechanics problems.

For Newtonian fluids, as water, the stress is proportional to the strain rate:

$$\tau_{ij} = \mu \dot{\varepsilon}_{ij}$$

(3.14)

where $\mu$ is the dynamic viscosity and the shear strain rate $\dot{\varepsilon}_{ij}$ is given by the following relation:

$$\dot{\varepsilon}_{ij} = \frac{\partial v_j}{\partial x_i} - \frac{2}{3}(\nabla \cdot v)\delta_{ij}$$

(3.15)

Substituting the SPH approximations for a function and its derivatives (see Equations 3.3 and 3.4), and taking into an artificial viscosity factor, the discretization of the conservation laws presented can be written as follows:

$$\frac{D\rho_i}{Dt} = \sum_{j=1}^{N} m_j (v^\beta_j - v^\beta_i) \frac{\partial W_{ij}}{\partial x^\beta_i}$$

(3.16)

$$\frac{Dv^\alpha_i}{Dt} = - \sum_{j=1}^{N} m_j \left( \frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Pi_{ij} \right) \frac{\partial W_{ij}}{\partial x^\alpha_i} + \sum_{j=1}^{N} m_j \left( \frac{\mu \dot{\varepsilon}^{\alpha\beta}_i}{\rho_i^2} + \frac{\mu \dot{\varepsilon}^{\alpha\beta}_j}{\rho_j^2} \right) \frac{\partial W_{ij}}{\partial x^\beta_i}$$

(3.17)

$$\frac{De_i}{Dt} = \frac{1}{2} \sum_{j=1}^{N} m_j \left( v^\beta_j - v^\beta_i \right) \left( \frac{\partial W_{ij}}{\partial x^\alpha_i} \right)$$

(3.18)

where

$$\dot{\varepsilon}^{\alpha\beta}_i = \sum_{j=1}^{N} m_j (v^\beta_j - v^\beta_i) \frac{\partial W_{ij}}{\partial x^\alpha_i} + \sum_{j=1}^{N} m_j (v^\alpha^j - v^\alpha_i) \frac{\partial W_{ij}}{\partial x^\alpha_i} - \frac{2}{3} \sum_{j=1}^{N} m_j (v_j - v_i) \nabla_v W_{ij} \delta^{\alpha\beta}$$

Here $i$ and $j$ are referring to the particles evaluated. The Monaghan-Gingold artificial viscosity term $\Pi_{ij}$ is introduced in the momentum and the energy equations for numerical stability. This is a common numerical artifice and helps to diffuse sharp variations in the flow, to dissipate the energy of high frequency terms and to prevent inter-penetration of particles.

Material Model

The system of differential equations (3.16) to (3.18) comprises five equations (equation (3.16) contains the three-dimensional velocity vector), but has six unknowns, thus requiring one further equation to close the system. This is achieved by adding an equation of state (EOS) relating pressure to density.

As water has a high sound velocity ($\approx 1.450 \text{ m/s}$), the computational time step in the case of an explicit solution scheme is very small compared to the typical duration of the flow phenomena. In addition, assuming the liquid to be incompressible requires a solution of the Poisson equation over the entire flow field and an implicit solution. The frequently used Tait EOS (also referred as Murnaghan law) [41] reduces the computation time by tackling this problem, and is therefore used in the present thesis. It allows for the representation of a fluid with artificially increased compressibility, with the formulation presented in (3.19).
\[ p = p_0 + \frac{c_0^2 \rho_0}{\gamma} \left[ \left( \frac{\rho}{\rho_0} \right)^\gamma - 1 \right], \tag{3.19} \]

where \( p_0 \) is the reference pressure, \( c_0 \) speed of sound, \( \rho / \rho_0 \) the ratio of current over initial mass density and \( \gamma \) the adiabatic exponent of the fluid. This approach is feasible for cases where flow velocities \( v \) remain well below the corresponding speed of sound, satisfying Equation (3.20), and hence compressibility effects are insignificant.

\[ \frac{c_0^2 \rho_0}{\gamma} \geq \frac{100 \rho_0 v_{\text{max}}^2}{\gamma} \tag{3.20} \]

where \( v_{\text{max}} \) is the expected maximum material flow speed. In this case, the inequality (3.20) states that the sound speed of the medium is at least a factor of 10 times higher than the bulk flow velocity.

### 3.1.3 Coupling with Finite Elements

Interaction between particles representing a fluid and moving or deformable structures may be modeled by one of the contact algorithms available in the VPS solver. Such algorithms, while allowing sliding at the interface, prevent penetration between selected structures. Their implementations are based on the well known penalty formulation (see Figure 3.2), where geometrical inter-penetrations between so-called slave nodes and adjacent master faces are penalized by counteracting forces. At each time step, each slave node (the SPH particles in this case) is checked and the nearest master is located. The contact thickness \( h_{\text{cont}} \) indicates the distance away from a contact face where physical contact is established. If the slave enters this zone, forces are applied to prevent further penetration. These forces are essentially proportional to the penetration depth \( \delta_{\text{cont}} \), and depend on the properties of the elements on each side of the contact.

![Figure 3.2: Penalty contact scheme.](image)

In addition, it is possible to define a tied contact in which virtual spring elements are automatically defined between particles that are sufficiently close to segments in the initial configuration. Such a contact acts as a rigid connection between the two parts [42]. More information on how each one of these two contact methods were used in the present thesis will be given in Chapter §4.
3.2 State of the Art on Coupled FE-SPH Simulations of Impact on Water

Finite elements and smoothed particle hydrodynamics have been used to simulate a variety of hydrodynamic impacts of simple geometries, and validation has been performed based on experimental results. The following review will be restricted to the specific case of simulating aircraft/rotorcraft impacting on water, focusing on relevant conclusions for the present research.

Significant efforts have been made for vertical impacts of aeronautical structures on water, targeting the helicopter industry. Toso [12], based on the knowledge gained in FE-SPH analysis of simple rigid bodies impacting on water, applied the method to the impact on water of deformable helicopter models. The quality of the calculated results was judged by comparing them to test measurements and observations gained within the CAST project [43]. The vertical impacts on water of a WG30 helicopter sub-floor structure and then of a full-scale WG30 model were investigated [44]. The simulation results remained conservative compared to the test data and the mean pressure level could be predicted, though the pressure peaks were over-predicted in stiffer locations.

In these simulations, two different approaches of water domain representation were investigated: one using a full grid-based FE model and one using a combined FE-SPH model for the water basin (in Figure 3.3 the results of latter are illustrated). Both presented similar quality in results. However, the simulation with the combined FE-SPH mesh for the water domain required 16 times the computational time (more than 10 days), and this with the half-sub-floor structure model.

Figure 3.3: Simulation of the half model WG30 subfloor structure impacting on water using a combined FE-SPH for the fluid domain [12].

Based on the experience gained with the WG30 helicopter, it was made clear that the investigation of the impact on water of deformable full-scale structures would require very large computation times. Although the FE mesh for the water was preferred for vertical impacts (due to similar results and faster computations), it was referred that if a horizontal velocity at impact has to be considered, the use of a SPH discretization for the water medium in the impact region was inevitable. The use of a classical FE mesh led to a dramatic decrease of the time step.

In [30], an assessment of the ALE and SPH fluid-structure interaction methodologies was conducted using test data from two vertical drop tests into water. The tests featured a 1.5 m diameter, 1.5 m long composite fuselage section, alone and with four blocks of a composite honeycomb energy absorber, impacting a 4.5 m diameter, 1 m deep pool. The SPH method was used to simulate both test configuration using three
different mesh densities. Findings were unexpected as the finest SPH mesh did not yield the most accurate results, but acceleration histories from the intermediate mesh spacing were in good agreement with the experimental test.

Forward velocity was considered starting in 2003, as investigations within the course of the CRAHVI European project report the results of FE-SPH ditching simulations of a rigid model of an Airbus A321 over the first second of impact [45]. A similar yet generic aircraft model is investigated with rigid FE and mesh refinements close to the region first impacting the water in [46]. Later, in [11], the method is considered by EADS-CASA, with a deformable FE sub-scale model of the CN-235-300M aircraft. All these investigations report considerable computational effort. Moreover, the latter research highlights the inability of the SPH formulation to portray suction forces in the rear part of the fuselage (see §2.2) when horizontal velocity is present. As the presence of suction forces (and possible cavitation) is critical in the evaluation of pitch behavior of the aircraft during the first seconds of impact, and in the consequent pressure distribution in the bottom of the fuselage, this handicap is seen as one of the major flaws of the method with respect to the ditching application.

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, initiated in 2011, 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, a model for cavitation [47], periodic boundary conditions and particle regularization methods. Some of these recent advances are summarized below.

The separation stress feature is a contact definition used to simulate suction forces (further explained in Section §4.4). In [48] the separation stress is used to account for suction in the CN235 aircraft model. The results in Figure 3.4 demonstrate that suction causes a very significant effect on pitch angle history and spray formation, and comparison to experimental results shows considerable improvement.

Figure 3.4: Importance of suction modeling by comparison of the simulated kinematics of an Airbus Military CN235 rigid sub-scale model [49].
One other concern regarding the standard weakly-compressible SPH formulation was the high level of noise in pressure results. This deficiency may be counteracted by application of pressure correction methods which aim to yield a more regular pressure distribution. Two methods, the density re-initialization by Shepard filtering and the Rusanov flux, were investigated in [20] and [49] during SMAES. These pressure corrections provide a significantly smoother pressure distribution than the reference SPH simulation but without notable effects on the free surface location and velocities.

Improvements to the VPS code concerned also the amount of particles necessary for the simulation, which caused excessive computational time. The ditching event is computationally challenging because it requires a sufficiently fine particle resolution but also a large fluid domain. For ditching where the high horizontal velocity necessitates an extended SPH domain, an appropriate option for further reduction of computational effort is that of periodic boundaries, recently implemented in the code. This feature allows particles leaving one end of the domain to enter at the opposite end. Opposing boundaries may be allowed to translate according to the motion of the ditching aircraft, without introducing additional velocities to the particles themselves. In this manner, the fluid domain near the impact region is always modeled, while reducing the length-wise dimension of the SPH domain. One other alternative algorithm implemented is the "deactivation" of particles, at each time step, further away from the impact region. The latter option is named active box via nodes (more information about these options will be given in Sections §4.3.2 and §4.3.3).

Above mentioned enhancements of the SPH module within VPS were successfully validated based on the experimental test campaign of Guided Ditching Tests also performed during the SMAES program [50, 51]. The activity was motivated by the need of achieving a better comprehension and more reliable experimental data for the aircraft ditching phase [52]. Tests consisted of high velocity impacts of metallic and composite plates with conditions representative of full-scale aircraft ditching. The problem was also addressed numerically using an FE model of the panel and supporting structure and SPH particles for the fluid domain. The numerical models included above mentioned features as particle regularization methods, periodic boundary conditions and others such as pressure gauges.

Initial comparisons with rigid structures in terms of water pressures acting on the plate showed promising results, in particular in non-steady pressure fields. Numerical results followed experimental tests at maximum values, time delay and spatial distribution [49]. The correlation between experimental and numerical results proves that, in fact, the advances made during the SMAES project with the FE-SPH module of VPS provide considerable improvements to the method in terms of both efficiency and accuracy.

### 3.3 Critical Review and Definition of the Thesis

The literature available highlights ditching as a relevant factor considered by aircraft manufacturers. The impact of the aircraft with water is crucial because the structure should withstand impact loads to allow a safe evacuation of the occupants. Most ditching analyses have been experimentally based, due to the complex physical phenomena involved, but the industry is now turning its attentions to numerical tools to aid in the design stage of safer aircraft. The difficulty in such analyses is not only the large deformation of the fluid domain, but the appropriate capture of relevant fluid-structure interaction phenomena.
Smoothed particle hydrodynamics is a particularly interesting method for modeling the water domain as it allows for easy coupling to standard FE aircraft models, widely used in crash simulations, since both formulations are Lagrangian in nature.

In this mesh-free method, particles which carry physical properties model the continuum water domain in discrete points. One particular code that allows for SPH and classic FE formulations to be used in the same simulation is VPS, released by the ESI-Group. Within the recent SMAES project, several new state-of-the-art algorithms have been implemented in this code, improving both the accuracy and the efficiency of FE-SPH analysis. Validation has been performed in studies using both sub-scaled models and guided plates impacting in water. The next logical step is 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.

In order to achieve this particular goal, several factors must be taken into account. First, FE aircraft modeling is a challenge by itself. Detailed finite element models are usually considered computationally expensive to be used in dynamic analyses by the industry. These are frequently condensed to so-called stick models, in which simple beams represent structural stiffness properties, and used to test several load cases. To determine ditching loads and analyze FSI phenomena, this approach is impractical as the outer geometry of the aircraft is not represented. Several researchers use full rigid FE models instead, where no skin deformation is portrayed, leading to lower computational requirements. Each one of these approaches has its advantages and disadvantages, and its application to ditching events should be weighed in to assess which is the most suited to study the referred case. Other than this, factors concerning contact between the structure and the water, such as the referred separation stress parameter, have yet to be tested to full-scale simulations. Finally, as in any other computational assessment, the type of memory processing plays an important role in efficiency. Literature studies regarding the application of shared/distributed algorithms to these particular simulations are scarce, but the potential efficiency gain could be significant.

The present thesis aims to apply acquired knowhow from SMAES project, i.e. state-of-the-art modeling as well as simulation techniques for the SPH fluid domain, to full aircraft ditching simulations in order to investigate the accuracy, efficiency and limitations of the different modeling approaches. The methodology incorporates the aircraft modeling knowledge of the DLR Institute of Structures and Design and makes use of the recently implemented algorithms in the VPS code.

One key element to allow for consistent benchmark studies and for incorporation of the ditching load case in larger tool chains is the automation of the pre-processing phase of the analysis. With this objective in mind, the author proposes to create a fully automated pre-processing script that allows for different models to be generated with reduced effort for the analyst.

Later in this research, the developed tool is tested by taking a standard full-scale transportation aircraft geometry and testing the several options available. Aircraft modeling techniques include detailed FE models, rigid bodies and a proposed alternative to standard stick models; SPH options include coupling with FE hydrodynamic volumes in the areas further from impact and advanced features such as periodic boundary conditions or active box; the separation stress feature is tested; finally, an advanced computational setting (Multi-Model Coupling) is analyzed.
4 Tool for Automated Model Generation

In this chapter the development of a parametrized, Python-based pre-processing toolbox will be presented. The described process chain is subsequently used to setup all the ditching simulations presented in the next chapter (Chapter §5).

The tool was developed using a standard desktop PC, Intel Core i5-3570 CPU @3.40 GHz, with 8GB of RAM memory and a 64-bit Windows 7 SP1 operative system. Studies concerning efficiency of the tool were performed in the same local machine.

4.1 Introduction

Typically, finite element analysis is separated in three distinct phases: pre-processing, solution and post-processing. The pre-processing phase consists of generating an appropriate geometry, finite element mesh, designate suitable material properties, and apply boundary conditions as restraints and loads to name a few. When dealing with hybrid FE-SPH analysis, this also includes an 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.

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 PAM¹ 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. Correct alignment/sequence of keywords and input is also of utmost importance, as exemplified by Figure 4.1.

| CAD | 1 | 0.01 | 0.01 |
| CAD | 2 | 0.01 | 0.01 |
| DOE | 3 | 0.01 | 0.01 |
| NOE | 4 | 0.01 | 0.01 |
| DOE | 5 | 0.01 | 0.01 |
| NOE | 6 | 0.01 | 0.01 |

(a) Geometry Definition Cards (b) Contact Definition Cards

Figure 4.1: Examples of VPS input format

During the course of this thesis, 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, elements, 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 (solution phase), and all what is left to the user at this stage of the development is the post-processing of the results.

¹More information about this dedicated environment for crash simulation can be consulted in [53].
It is important to highlight that some of the options included are only implemented in the VPS development (special) version made available to the DLR, but are still to be implemented in the commercial version released by the time of this research. The script incorporates pre-existing in-house modeling tools and concedes the opportunity to expand and make use of an available library of aircraft models, amongst which one can find examples of the several modeling approaches in Table 4.1.

Table 4.1: Aircraft modeling capabilities of the tool and compatible VPS version (see §4.2).

<table>
<thead>
<tr>
<th></th>
<th>Commercial Version</th>
<th>Special Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>General Finite Element Models (GFEM)</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Detailed Finite Element Models (DFEM)</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Dynamic Master Models (DMM)</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Basic Rigid Body Models (RBM)</td>
<td>X</td>
<td>X</td>
</tr>
</tbody>
</table>

Subsequently, the three dimensional water domain is generated by the tool. The modeling variants include different material models and several SPH state-of-the-art options such as the ones referred in Table 4.2.

Table 4.2: Water modeling capabilities of the tool and compatible VPS version (see §4.3).

<table>
<thead>
<tr>
<th></th>
<th>Commercial Version</th>
<th>Special Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cubic/Hexagonal Particle Distribution</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Active Box via Nodes</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Periodic Boundary Conditions</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Combination of SPH and FE Solid Elements</td>
<td>X²</td>
<td>X</td>
</tr>
</tbody>
</table>

It is also important to highlight the ability of the developed tool to allow for advanced computational methods due to the modular nature of the input files generated. Different computational settings are available.

Table 4.3: Computational processing capabilities of the tool and compatible VPS version (see §4.5).

<table>
<thead>
<tr>
<th></th>
<th>Commercial Version</th>
<th>Special Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shared Memory Processing (SMP)</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>Distributed Memory Processing (DMP)</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>Multi-Model Coupling (MMC)</td>
<td></td>
<td>X</td>
</tr>
</tbody>
</table>

The automation of such process can possibly allow for coupling to other disciplines in the future, as in Multi Disciplinary Optimization (MDO) chains. With this in mind, time efficiency of the code and standardization of user input to fit general patterns were taken into account.

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.

²Not compatible with periodic boundaries or active box in the commercial version, only when coupled to full-pool option.
4.1.1 Development Methodology

ESI-Group constantly releases new versions and features for the VPS solver, and as such the tool was developed having a fully parametrized layout in mind. This allows for new functionalities to be added or modified in the future and avoids limiting different combinations between them. Incorporation of pre-existing in-house tools was also necessary. Figure 4.2 offers an overall view of the code’s final setup.

![Diagram of code setup](image)

**Figure 4.2:** Overview of the code of the developed pre-processing tool (emphasis on modular setup).

The top left block represents the computations related to the aircraft model. The head choice consists of either to generate a new model or to use one previously developed, properly stored in a digital library. If the latter option is selected, aspects as aircraft attitude, positioning or initial conditions can still be modified, but all the aircraft computations regarding geometry and mesh generation are spared thus saving a considerable amount of time usually required for such complex structural models.

If the user chooses to generate a new global FE aircraft model, a pre-existing in-house design tool is automatically deployed. The referred tool, AC-CRASH, generates the complete FE model of the fuselage structure (skin panels, frames, stringers) corresponding to any configuration specified. The subsequent modules of the developed tool can later transform any model generated by AC-CRASH into rigid body or an adapted condensed beam approach. New models developed are finally stored in the referred library, together with a short summary of the modeling process, and can be used in further simulations. All the models are posteriorly attributed loads, such as nodal mass distributions or a lift model.

The unit below represents the water related computations and is divided in several subsections regarding particle distribution or SPH special options available. If modeling of classic FE entities for the water is requested, ANSYS is deployed for mesh generation and the resulting files are automatically converted to VPS input format and integrated in the overall model. The dotted line that connects the aircraft and water modules represents the interaction necessary for active box or periodic boundary conditions (see §4.3.3 and §4.3.2). If none of these two options are activated, these two modules are completely independent.

The last unit represents the general module that brings together the aircraft and water modules. If requested, the simulation is automatically started once the pre-processing computations are finished.
4.1.2 User Input and Graphical Interface

User input is required in three different manners: through a CPACS (Common Parametrized Aircraft Configuration Schema) file for aircraft geometry definition, through a Python file organized in a series of Python dictionaries, which is then fed to the different modules of the tool, and optionally through a graphical interface.

The geometric data of any initial aircraft configuration is retrieved from a CPACS database, a parametrized XML-based schema where the user can manually alter parameters raging from fuselage geometry and structural element definition to detailed material definitions. An example of such file can be observed in Figure A.1 (Appendix B). This hierarchic format was developed by DLR in the course of the TIVA (Technology Integration for the Virtual Aircraft) project in 2005 as a standardized aircraft description which serves as in and output for different analysis tools. More information about the CPACS format can be consulted in [54].

The Python input file contains the definitions regarding the chosen water basin. Selections range from dimension (width, height and length of the domain) to advanced SPH aspects. As the same aircraft model can be used for more than one simulation, parameters concerning aircraft initial conditions (attitude, velocity, positioning relative to water domain) can be changed here. This leaves the CPACS file exclusively used for definition of aircraft geometry properties. The user should also decide between different contact options. The last section of the referred Python file concerns computational aspects and VPS output controls. In addition, the user can opt to make use of the graphical interface developed.

**Graphic Interface Mode** The graphical interface allows for easier, more intuitive setup of aircraft modeling options. The several choices are displayed along side a short description of each command. In addition, the user can have an overview of the models available in the library. This mode is required to have access to the full range of options incorporated in the tool. An example is given in Figure 4.3.

![Figure 4.3: Examples of graphical interface of the developed pre-processing tool.](image)

**No Graphic Interface Mode** Upon choosing this option, the tool will get a default model (previously stored in the library and indicated in the Python input file) and the simulation will be deployed automatically upon termination of the pre-processing phase. This mode is appropriate if the user wishes to run the tool remotely (e.g. RCE applications) and hence no graphical interface is available, or if the tool is to run as a segment of a wider process chain.

---

3 Dictories consist of pairs of keys and their corresponding values. Also known as associative arrays or hash tables.

4 Remote Component Environment - distributed platform for integration of applications developed by the DLR, see[55].
4.2 Aircraft Model Generation

The process chain for generation of the different aircraft modeling approaches is described in Figure 4.4. It can be observed that consistency between the models, in terms of geometry, total mass, COG position and inertial properties, is guaranteed as the input is the same for any of the modeling approaches.

As previously referred, the information regarding the aircraft fuselage geometry is contained in the CPACS file provided by the user. The script analyzes this data and allows for the generation of four different models based on the given geometry description. The first two are Global and Detailed Finite Element representations (GFEM/DFEM, left in Figure 4.4), the most complete FE models. The GFEM uses deformable shell elements for the skin and beams for most of the remaining structures, while the DFEM includes local refinements in areas of interest. If requested, either models can be simplified into a single rigid body (RBM), as it is shown in the center of the figure above. If the global stiffness distribution is to be considered, then an adapted condensed beam approach, referred as the Dynamic Master Model (DMM), is generated. This is performed by another external in-house tool which analyzes the detailed FE model and outputs the stiffness properties of the structure, and by the developed toolbox which then generates the necessary mesh and applies the corresponding properties. An example of this modeling technique can be seen on the right side of Figure 4.4.

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 toolbox offers several examples of these distributions but the user is free to modified 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.

The next subsections address how the developed pre-processing tool generates each one of the aircraft modeling options referred above.
4.2.1 General and Detailed Finite Element Models

The generation of the GFEM and DFEM is entirely performed by AC-CRASH. This fuselage modeling tool was developed to automatically generate a global FE model for preliminary sizing purposes, using deformable shell elements for the skin and beams for most of the remaining structures (GFEM). To be also used in crash simulations this model was extended in a way that certain regions, where highly non-linear behavior was expected, could be modeled by the use of shell elements and optionally a finer mesh (DFEM).

In case of using the global approach, the fuselage model is meshed using shell elements for the skin and elastic beam for structural components (e.g. frames, stringers, crossbeams). However, beam elements do have the disadvantage of not being suited for displaying local stress concentrations. To approach this discrepancy a global model (using elastic beam elements) with local refinements (using shell elements) was developed for AC-CRASH V3 [56]. Those local refinements are named detailed regions. By this approach the advantages of both discretization methods are combined: the global aircraft stiffness is represented correctly by computationally cheap beam elements and the areas of interest can be modeled using shell elements, thus benefiting from their advantages over beam elements.

The detailed region modeled in the course of this thesis is the first zone to get in contact with the water upon ditching, highlighted in Figure 4.5. This mixed discretization method requires the adaption of the mesh at the common interfaces. This includes the interfaces between shell and beam mesh as well as, for the fuselage skin, the mesh size adaption from detailed regions to the global mesh size. The second item is not necessarily required as it is possible to mesh the complete fuselage skin with one single mesh size. However, as detailed structural behavior in terms of non-linear deformation may not be displayed when using large elements, the fuselage skin in detailed regions is modeled finer than in the remaining structure. Connecting the beam node with the corresponding shell nodes at their common interfaces is performed by coupling translational and rotational degrees of freedom. Regarding the interface between finer and coarser mesh areas, transitions zones are defined. These are basically areas around the detailed region giving space to a stepwise adaptation of the mesh sizes, thus avoiding high gradients. An example is shown in Figure 4.5 (b).

Benchmark tests in [56] show that the mesh generation for fuselage sections of up to 28m of length, with approximately 1.6 million nodes, requires less than 90 minutes to be preformed by AC-CRASH, predestinating it for the use within parameter studies.

Figure 4.5: Examples of refinement in the ditching impact zone.
After the generation of the fuselage model by AC-CRASH is completed, all the output files are taken by the toolbox for further computations. Due to the complexity of the models, the nodal mass distribution was not automated for the GFEM or DFEM approaches during the course of this research. The methodology followed to apply specific load cases was therefore manual, with the aid of Visual-Crash PAM. First, the difference between COG position and total mass of the selected nodal mass point distribution case and the resulting total structural weight of the fuselage model was assessed. The remaining mass (corresponding payload and fuel) was distributed through the passenger seats, in order to achieve correct COG placement and inertial properties corresponding to the desired load case.

4.2.2 Rigid Body Models

The rigid body modeling approach (RBM) is a simplified, undeformable version of the detailed or global FE models presented before. It is created by constraining the shell elements of the fuselage skin into a single rigid entity, and deleting all the remaining structural items (e.g. frames, stringers, passenger seats). Total mass $m$, center of gravity COG and moments of inertia $I$ are transferred to the simplified geometry as input scalars (see Figure 4.6). The main advantage of this approach is the considerable reduction in computational expense of the model.

![Figure 4.6: Scheme of condensation of RBM from AC-CRASH FE model.](image)

The transformation process is performed automatically by the developed pre-processing tool. Below, a brief overview will be given on how the Python code performs the operation. Figure 4.7 illustrates the geometry files of the original FE model to be transformed.

![Figure 4.7: Example of computations for the generation of the Rigid Body Model.](image)

The tool analyzes this data and locates the group containing the list of element numbers which form the fuselage skin, as exemplified in the top segment of the image. Each of these entries represent shell elements. The code subsequently searches for the location of the corresponding element definition cards (second block of the referred figure). These cards contain, amongst other definitions, four node ID’s linked to the nodal points which constitute each of these shell elements. The script proceeds to locate the indicated
nodal definition cards (last segment of Figure 4.7 on the preceding page), where the correspondent nodal positions (in x, y and z axis) are defined. The process is repeated for every element in the fuselage skin group. All the information is internally stored to be included in the RBM mesh file. Remaining items are discarded. For full-scale aircraft models this type of file search can comprise millions of points, confirming the importance of code efficiency in such computations.

A uniform thickness distribution and a null material (VPS material Type 100) is attributed to the resulting shell elements. The null material is an efficient means (large time steps possible) for modeling of contact surfaces when deformations of these surfaces are not of interest, and thus suited for this aircraft modeling approach.

All the elements are connected by a rigid body definition card. The total mass and full inertia tensor of the model are automatically computed from direct analysis of the nodal mass distribution chosen, and transferred to the center of gravity of the model in question by the developed tool. In the standard desktop PC used for these studies, the RBM model generation is concluded in a matter of seconds, following AC-CRASH mesh generation.

The resulting RBM is a very simplified approach when compared to the original detailed FE model, yet it retains the same outer shape and inertial properties while leading to a significant reduction in simulation time. However, no deformation of the elements or flexibility of the overall structure are reproduced, and consequently no energy abortion is portrayed. The next model comes as a suggestion for compromise between computational efficiency and stiffness representation.

4.2.3 Dynamic Master Models

Aircraft simplified beam FE models, also known as stick models, are commonly used in civil aircraft design. Accurate prediction of bending and torsion of the aircraft depends on extracting the stiffness properties of the main structure and applying it to a set of beam elements extending along the structure [57]. The Dynamic Master Model (DMM) approach considered in the present thesis takes from this condensed beam concept and adapts it to be utilized in the context of ditching simulations. This is done by connecting several sets of rigid shell elements to a flexible beam structure, in order to represent the contact surface with the water domain.

The generation process for this model follows several stages. The first step is the computation of stiffness properties of the original detailed FE model. To this aim, an additional external tool developed by DLR (consult [58]) is incorporated into the pre-processing toolbox. The process chain of the referred tool is illustrated in Figure 4.8 below.

![Figure 4.8: Scheme for Stiffness Computation](image-url)
The structural analysis tool referred takes the detailed FE model of the fuselage and locates several sections along the longitudinal axis, in between the frame positions. It then calculates, for every stringer and shell element contained in each section, the bending stiffness around y- and z-axis and stores it in arrays of data. These values are calculated with respect to the global coordinate system and center of gravity of each section. The calculation of the axial and torsional stiffness follows. The final result is a complete set of section-wise condensed structural properties, which are output in the form of text files. The stiffness data is later fed to the pre-processing toolbox for the next stages of model generation.

Next, along the horizontal axis, a beam geometry is created for each section in between the fuselage frames, and the corresponding stiffness properties previously computed are attributed to the beam elements. At the end of this stage the group of nodes and beams generated is able to reproduce the global stiffness distribution of the structure.

It is worth mentioning the effort made during the projection of this model approach in increasing the stable time step of the beam elements generated. For beam elements such as the ones used for the DMM the stable time step $\Delta t_s$ is limited by length $L$, Young’s modulus $E$ and density $\rho$ [59]:

$$\Delta t_s = \frac{L}{\sqrt{E/\rho}}$$  \hspace{1cm} (4.1)

The length of each beam is limited by the space in between frame positions, and the Young’s modulus is defined to represent the accurate stiffness properties of each section. Therefore, these two factors cannot be altered, but the density of the material can be artificially increased for a higher time step without affecting overall mechanical properties. However, the mass of the fuselage structure is entirely represented by nodal mass distribution; increasing the beam density leads to an increase in total mass of the model which cannot be neglected. This effect is portrayed in Figure 4.9.

![Figure 4.9: Nodal and beam mass scheme for DMM model.](image)

This issue is solved by the pre-processing tool by computing the mass of each beam segment and subtracting it from the nodal mass of its two nodes. However, if the beam density is increased to higher values in order to achieve a higher time step, it might happen that the beam mass becomes larger than the nodal mass applied, which limits the process described. A compromise between these two factors is therefore necessary: higher density for higher time step and consequently faster computations, lower density to avoid an incorrect mass distribution. Several values are tested and a constant optimum density is chosen to be applied to the beam model. Nonetheless this is a point for improvement, since a non-constant density distribution could lead to higher reductions in computational time.

For the modeling of fluid-structure interaction, the beam model by itself is not sufficient. For this, the tool takes the undeformable fuselage skin model generated for the RBM (see Section 4.2.2) and cuts it into several sections.

The fuselage skin geometry is sectioned at the exact positions of the aircraft frames. The nodes at the intersection of the different sections are doubled and attributed to the corresponding part, in order to avoid the different sections from sharing any nodes, as can be seen in Figure 4.10(a). This allows
for independent movement of the segments. The connection between the shell and beam elements is performed by coupling all degrees of freedom of the surface to one of the nodes of the corresponding beam (Figure 4.10(b)). This rigid connection allows for the movement of each section to be dictated by the corresponding beam, that accounts for global stiffness.

![Diagram showing shared and duplicated nodes, flexible beams, and rigid body connections.](image)

(a) Cutting process  
(b) Link between beam and shell section

**Figure 4.10**: Dynamic Master Model computations scheme

Similarly to the RBM approach (see section 4.2.2), all the geometry generation for the DMM is performed by the Python toolbox from direct analysis of extensive mesh files, as those previously shown in Figure 4.7. An example of a DMM, after the pre-processing chain is concluded, is illustrated in Figure 4.11.

![Example of Dynamic Master Model approach.](image)

**Figure 4.11**: Example of Dynamic Master Model approach.

Ideally, the location of the beams for structural stiffness representation should be coincident with the elastic axis of each section. However, at this stage of the development, the incorporated in-house tool does not output the data relative of the position of the elastic axis, and therefore as a first approximation the beams are placed along the longitudinal axis of the fuselage. It is thought that this approximation is reasonable considering the purpose of ditching investigations. However, it is a factor which should be improved in the future.

### 4.2.4 Wing and Empennage Model

The modeling process chain described in the previous sections concerns only the fuselage structure. For wing and empennage representation, the toolbox offers simplified beam models. The process into obtaining the stiffness distribution of these elements, if desired, is similar to the one followed for the fuselage in the DMM approach. On the other hand, if stiffness distribution is considered to be unnecessary, a rigid model of the structures can also be used. In both cases, contrary to the DMM, no fluid-structure interaction is portrayed as no shell sections are connected to the referred beams. The purpose of these wing and empennage models is to achieve global realistic mass and inertial properties of the aircraft, or to analyze the effect of their flexibility in the fuselage structure during the ditching event. In the future, these models may be used to attach engines which may affect the ditching behavior.

For the models generated by AC-CRASH (GFEM and DFEM approaches), the available wing and empennage structures are attached to a range of selected frames of the fuselage model, as in Figure 4.12.
The connection is made through rigid body definitions, i.e., the selected fuselage frames are artificially stiffened in order to allow for the attachment of the referred structures. If the wing is modeled as a set of rigid beams, then both the frames and all the beam structure are constricted in all degrees of freedom. On the other hand, if the wing is modeled as a flexible structure, only the beams in the vicinity of the selected frames are rigidly connected to the latter, and the remaining wing elements are free to deform.

For the RBM approach (see §4.2.2), the wing and empennage models are directly attached to the rigid fuselage skin. Similarly to the previous example, the structures might be constricted in all degrees of freedom of the fuselage or allowed to deform freely. For the DMM, the attachment is conducted by connecting these structures to a range of selected sections of the fuselage structure, as in Figure 4.13.

In the latter case, the fuselage beams in the interior of the selected sections are also artificially stiffened, as the connection is performed by rigid links between these elements and the elements of the wing. Both in the AC-CRASH and DMM fuselage approaches, the attachment of the wing and empennage beams alters the original stiffness distribution of the fuselage input by the user. This factor may lead to inaccuracies in selected regions and should be taken into account.

### 4.3 Water Model Generation

The modeling of the water domain includes the setup of appropriate material properties, correct geometry generation, SPH features regarding smoothing length and kernel function, in between many other parameters. Other than this, the water domain has been modeled and validated by a hybrid approach combining SPH elements and classic FE volumes before [12], and as such this approach is available to be generated by the automated pre-processing tool.

Over the past few years, research has been focusing on improving accuracy of this method and reducing computational time. For this, the SPH module within VPS has been extended to accommodate several novel features such as periodic boundary conditions, damping zones and deactivation of particles outside the domain of interest. These new options are particularly interesting for ditching of large structures where high forward velocity is involved, as they allow for the water medium to be extended without a huge penalty in simulation time. The next sections will focus on these new modeling aspects, and how the tool conjugates them to fit to the ditching purpose.
4.3.1 Classic Approach and Particle Packing

The classic approach to modeling of the water domain is to use a SPH particle distribution in the impacting area, where splash and larger deformations are expected, and a FE volume mesh (solid elements) further from the impact site. This mixed FE-SPH approach allows for the extension of the water pool while providing acceptable computation time, as the computational effort per time step is less for FE volumes compared to SPH particles. For this work a homogeneous mesh for the hydrodynamic FE volumes is considered. An example is shown in Figure 4.14.

![Example of classic FE-SPH approach for the water domain.](image)

**Figure 4.14:** Example of classic FE-SPH approach for the water domain.

The transition between the SPH particles and the volumes is made by a tied connection, in which virtual springs are defined between particles that are sufficiently close to the volume elements (see §3.1.3). In other words, the displacements of the SPH particles near the border are linked in all directions to the neighboring nodes of the FE volumes, ensuring that there is little reflection from the coupling interface.

Concerning the SPH domain, initial particle position is of upmost importance since each particle’s properties are interpolated upon its neighbors relative positions. It is therefore essential that the particle arrangement is locally isotropic [60]. Several arrangements are possible and have been object of study. The present pre-processing script allows for particle distributions in two schemes: cubic and hexagonal packing as in Figure 4.15.

![Available SPH packing schemes. Images from [61].](image)

**Figure 4.15:** Available SPH packing schemes. Images from [61].

Probably the simplest and fastest way to set up a uniform particle distribution is to arrange the particles on a cubic lattice. One of the problems with this method is that it may lead to pronounced preferred directions along the x, y, and z-axis [60]. A more compact structure is produced when using a hexagonal packing. This configuration is one of the optimal ways to pack uniform spheres together. Similar to the previous arrangement, it has the problem of having preferred directions, but these have been proven to be less pronounced than for the cubic packing configuration [60]. Nonetheless, this configuration involves an increase in the number of particles per volume and larger number of neighboring particles, both leading to higher computational cost for the same volumes with identical particle spacing.
According to the packing chosen, the particle position is computed by the tool, and the individual particle volume is retrieved by dividing the total SPH water volume by the total number of particles. Generation of the FE domain follows and coupling between the domains is defined. Material model and properties are chosen by the user and automatically attributed to both types of elements. The option to attribute the initial particles a hydrostatic pressure distribution based on the free-surface level is also allowed, saving the computer time expended to reach hydrostatic equilibrium under gravity in the water before the ditching simulation starts.

The extension of the water pool with FE volumes allows for a reduction of the necessary SPH domain cross-section wise, and therefore for a considerable reduction of computational time. However, the length of the pool still needs to be relatively large in order to accommodate for the high forward velocity of the aircraft model. This becomes more evident if long run times are to be simulated, in which case the water length has to be extend to several dozens of meters. The next options to be presented concern this problem, and present alternatives for extension of the SPH domain with minimum time penalty.

4.3.2 Translating Periodic Boundary Conditions

Translating periodic boundaries allow for the SPH domain to be moved with a velocity that may be linked to a node of the aircraft model. The particles leaving a user defined rectangular domain are re-entered at the opposed face, and may be reset to the initial state conditions. This allows for extension of the horizontal domain to an unlimited length without the cost of increasing the number of particles. An example of this configuration is given in Figure 4.16.

![Figure 4.16: Exemplary scheme of Periodic Boundary Conditions.](image)

In the particular case of aircraft ditching, any particle that exits the rear face is entered at the front face of the periodic domain. In the present study the particle upon re-entrance will be given the initial state conditions, and the distribution over the face where it enters is also reset to the initial one by turning the displacements in the y- and z- direction to zero. However, the front and back particles (right scheme in Figure 4.16) behave as neighbors due to the periodic nature of the approach. Since the properties of a particle are interpolated from its neighbors, the wake behind the aircraft can still influence particles at the front face. In order to remove these disturbances coming from the boundary an option has been developed for VPS that damps the particle movement and properties through a defined region, the so-called damping zone. When a particle passes this horizontal region defined by the user, its properties are linearly interpolated as a function of the amount of the traversal length of the particle through this domain. The damping region is put at the front of the domain, and allows for undisturbed water particles constantly being re-entered in the domain.
By default this periodic domain is fixed in space, but the recently implemented code for VPS allows for the movement of the SPH region to be linked to a moving node. This feature allows particles inside the fluid domain to follow the motion of a prescribed node without introduction of velocities. The pre-processing tool automatically assigns the horizontal movement of the translating periodic box to be equal to that of the center of gravity node of the aircraft model. The size of the box as well as the damping zone are user-defined.

### 4.3.3 Active Box via Nodes

The active box option consists in deactivating the SPH particles located outside a user-defined domain of interest. An example is given in Figure 4.17, where the active box option is defined by nodes N1, N2 and N3. For the inactive particles (green), the SPH algorithm is skipped which accounts for the possible time savings. Once the active domain, which is translating with the aircraft, reaches a particle’s position, the particle is activated and it is computed as default. Once a particle leaves this domain again, its properties (velocity, density, pressure) remain constant. As only a certain parcel of the SPH interaction is being computed at each time step, the computation time is considerably reduced, while the region in contact with the aircraft is always computed.

![Figure 4.17: Example of Active Box via Nodes](image)

The nodes N1 and N2 define the initial size of the active box and have a given velocity in order to follow the aircraft (the tool does so by means of a nodal constraint card to the COG of the aircraft model). Additionally, a third node (N3) has been added to allow for the active box domain to change its size during the course of the simulation. The particles which are located within the rectangular region defined by the (instantaneous) minimum and maximum coordinates of the three nodes are active. This means that if N3 is given a specific velocity or constraint to another element which motion is faster (or slower) than the other two nodes, the active region will vary in size depending on which node has the highest (and lowest) coordinate. If selected, the tool takes the prescribed initial and final dimensions of the active box to be simulated and tunes the velocity of this third node to satisfy these conditions, hence allowing for a prescribed linear growth.

The particles are deactivated over a certain transition region, represented in pink color in Figure 4.17, to provide proper neighbor particles to the active particles. To avoid non-physical results for simulations involving gravity, the hydrostatic pressure is assigned automatically to particles prior to activation when the hydrostatic equilibrium option is selected.
4.4 Modeling of Interaction Between Fluid and Structure

Interaction between the SPH particles representing the fluid and the moving aircraft structure is modeled by one of the contact algorithms available in the VPS explicit solver. This algorithm is based on the penalty formulation referred in Section 3.1.3 between so-called slave nodes and adjacent master faces. Penetration between selected entities is prevented, while still allowing sliding at the interface [62]. A scheme of the standard contact is presented in Figure 4.18 (a).

The standard contact algorithm automatically detects when an SPH particle (slave node) penetrates any segments of the outer surface of the finite element model of the structure (master). The contact thickness $h_{\text{cont}}$ indicates the distance away from the master face where contact is established and the counteracting forces are applied. It has been found that the correct position is reached when the thickness defined for the contact is in the vicinity as half the particle spacing plus the local skin thickness [62], and so such is defined automatically by the tool. By default, the scaling factor for the force is proportional to the penetration depth. However, due to the high impact forces in the considered application, the option for nonlinear penalty stiffness is applied. This parameter can be activated to avoid penetration in severe contact problems and was applied after some simulations showed unphysical penetration upon impact.

However, the current SPH formulation, the water equation of state and the traditional penalty contact are not able to physically model the suction effect explained in Section 2.2. To account for this hydrodynamic phenomenon, a separation stress feature on the contact definition may be used to mimic the effect - Figure 4.18 (b).

The latter option models a sticky contact between the SPH particles and the aircraft skin. A search radius is defined by the product of the separation thickness factor $SEPTHK$ and the contact height $h_{\text{cont}}$. After the contact is established, it is maintained as long as the penalty stress remains smaller than an input separation stress [48]. This penalty stress is defined as the algebraic distance between a node and the connected segment (it is negative when there is penetration for example) multiplied by the contact stiffness divided by the node area. If the separation stress is set to zero, the algorithm will function as the traditional penalty algorithm. If a positive number is entered, the particle will remained attached to the defined zone as long as the penalty stress is smaller than the separation stress [63]. A standard to use this option is to force a sticking contact through the input of a large magnitude for the separation stress, but this parameter may require calibration and therefore it lets as an input for the user of the tool. The same applies to the separation thickness factor.

![Figure 4.18: Options for contact between aircraft and water domain [48].](image-url)
4.5 Computational Aspects

Developments in computer science over the years have increased memory size and computational speed. Moreover, for large models that require significant computational time, parallel processing methods have been developed that allow for the distribution of a simulation process over multiple instructions on a shared memory computational system. This has a pronounced impact in overall simulation time, and should therefore be taken into account in the present studies. In order to have a deeper understanding of these options a brief theoretical introduction will be given, and the implementation in the pre-processing tool will be presented later.

Parallel computing basically consists of two different classical processing methods: shared memory and distributed memory processing. There are many structural differences between these two approaches, but the goal is ultimately the same - to perform faster and larger computations by utilizing hardware in parallel. A scheme of these two architectures can be observed in Figure 4.19.

The name of each method expresses the basic difference between them. In the case of shared memory processing (SMP), the distributed parts of the overall program that are running in parallel all share the same memory space in a node\(^5\). This provides speed when passing data between the cores and processors. Yet, the major drawback of shared memory computing is that the computing resources on a shared memory node are limited. Additional resources cannot be added when the problem size is increased or more cores should be used to reduce the number of computations per core [64, 65].

In distributed memory processing (DMP), the memory is not shared but instead is distributed among several parallel processes. These processes have to explicitly communicate with each other by sending "messages". As a consequence, communication and synchronization consumes additional time. The big advantage of distributed memory computing is that additional resources (nodes, and hence cores and memory, too) can be easily added. This is typically used for cluster machines, consisting of several nodes that function as a single resource [64, 65].

\[ \text{Figure 4.19: Schematic architecture of parallel processing methods, [64, 65]} \]

Distributed memory processing becomes even more useful when looking into the multi-domain approach. This multi-domain approach is commonly known as Multi Model Coupling (MMC). The basic idea behind this concept is that the simulation domain can be divided into sub-domains based on a difference in maximum stable time steps of the sub-models. In the context of ditching simulations the division could be between, for example, the water and the aircraft model. Each sub-domain is then attributed a fixed number of processors, optimized for equilibrated CPU usage [66].

---

\(^5\) A node is a discrete unit of a computer system that typically runs its own instance of the operating system, like a laptop or a workstation.
The objective of MMC is that each sub-domain is allowed to advance at its own (different) time step, and synchronization points establish the communication (i.e. contact) between both. With this, fine local meshes can be defined without significantly increasing the calculation time for the whole model. A scheme is presented in Figure 4.20.

The aircraft sub-model includes fine elements with a low time step $\Delta t_L$, while the water FE-SPH model has a coarser discretization with a larger time step $\Delta t_G$. The models interact at every super cycle, coinciding with the larger time step, and the contact is computed. The potential arises from the fact that the global model, with higher number of elements but larger time step, does not have to be computed every time the stable time step of the finer model is reached, and thus this may result in significantly reduced simulation time when this time step ratio is increased.

Up until the date of this thesis the VPS solver allowed for shared memory processing methods in all releases, but the distributed memory option was only available for the commercial versions of the code. Consequently, when making use of the latter approach, options like periodic boundary conditions could not be modeled, due to their availability exclusively in a development version (as referred in Table 4.3) to date.

In order to make use of these different processing methods, VPS requires a specific format of the input files that contain the model. The developed pre-processing tools allows for these different formats while maintaining a consistent file structure. This parametrized structure was thought out for readability and to allow for exchange between models.

When the Multi Model Coupling option is used, it is necessary for the output parameters, time step control and code version to be defined for each one of the sub-models. The top level file only contains information relative to the models to be included and the contact between them. This translates into the following file structure:

![Diagram](image)

**Figure 4.20:** Scheme of Multi Model Coupling Method [67]

**Figure 4.21:** File structure for simulation models to be used with MMC.
When shared memory processing or standard DMP are used, the time step is the same for both models, and no parameter definition can be repeated. The file structure may be simplified, but it was not altered in order to maintain the modular character of the developed tool also on the level of the in- and output files. The resulting structure is shown in Figure 4.22.

![Diagram of file structure]

**Figure 4.22:** File structure for simulation models to be used with SMP or standard DMP.

It is important to refer that for the Multi Model Coupling file structure, the sub-models are completely independent from each other. This means any of the sub-models can be opened and used separately if the user wishes so. The latter SMP/DMP file structure does not fully allow this, as essential data is stored in shared files.

### 4.6 Final Considerations

The developed pre-processing tool has been proven successful in generating simulation models in VPS format for all the aircraft and water configurations present in this chapter. It was shown to be particularly helpful for the benchmark studies that follow in Chapter 5. The fact that the process chain is automated and mostly programmed in Python language makes it fast and efficient: generating a pool of half a million SPH particles in combination with water FE volumes takes no more than two minutes on a standard desktop PC, for example. SPH special options implemented within the SMAES project can be easily tested for essential code validation, since only a limited number of parameters in an easy-to-read input file needs to be defined. Transforming a detailed aircraft model into one Dynamic Master Model, if done manually in a visual environment, could take several hours whereas the developed tool performs this task in just a few seconds. This now allows for studies on different fuselage geometries or fuselage partitions with minor additional effort. This was achieved by taking particular care in dealing with the large amount of data contained in the mesh files analyzed, and with the resulting information to be written onto the new VPS input cards, by surveying several coding data structures available.

Other than this, the fact that the same input CPACS file can be used to compute a GFEM/DFEM, DMM and RBM aircraft assures consistency between the three, thus allowing for direct inter-comparison of these aircraft modeling approaches. The option to automatically store these models in a digital library allows for considerable time savings when investigating the effects of different aircraft attitudes and velocities upon impact, or surveying different water modeling options while utilizing the same aircraft model. The tool becomes particularly intuitive if one makes use of the graphical interface mode, which provides a brief overview of models in any defined library, and step-by-step explanations of the stages in the aircraft model generation.
The modular layout of the resulting input VPS files makes it simple to go through the extensive generated data, and allows for the analyst to make use of advanced computational schemes as Multi Model Coupling. Furthermore, the parametrized layout of the code allows for new modules to implemented in the future, giving room for this state-of-the-art pre-processing tool to be continuously improved.
5 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 are tested. All the models analyzed are generated using the developed pre-processing tool. In a first part, different approaches to the water basin are investigated. The classic approach using a full pool with FE and SPH elements is used as a reference, and the influence of different parameters is studied. Furthermore, advanced options such as translating periodic boundaries and active box are applied to study their applicability to the problem of full aircraft ditching simulations and the possible time benefits arising.

In a second part, different aircraft modeling approaches are studied. A generic transport aircraft geometry is tested and analysis is performed by using the same CPACS input file (see subsection §4.1.2) and alternating within the four modeling possibilities allowed by the tool, from simple rigid body to fully detailed finite element models. Results are shown in terms of variation of contact force history, acceleration and time efficiency. In addition, the separation stress feature is used to assess the possibility of modeling suction forces between the water and the aircraft. Finally, regards concerning computational performance using Multi Model Coupling are made.

The aircraft used in the simulations performed represents a generic single aisle, 37.5 m 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. Further general information concerning COG. position and inertial properties of the models will be given in the following sections.

The mass distribution originates from available mass point data representative of a 50% payload and 50% fuel mass upon impact, totaling at around 72.5 tones. 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 $v_x = 65 \text{ m/s}$ and $v_z = -1.5 \text{ m/s}$, respectively.

Either in reality or model tests the aircraft may accelerate due to some aerodynamic unbalance, and so external forces should be taken into account [48]. In order to portray realistic conditions, gravity is applied to both structure and water and a simple linear lift function 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.

The computations were performed in a computational node consisting of 2 Quadcore-64bit CPUs of Xeon E5540 type (Nehalem, 2.53 GHz), and having 24 GB of RAM (DDR3-1333, 6x4 GB).

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, and to assess their limitations.
5.1 Investigations Concerning Water Modeling Approaches

To investigate different water modeling approaches the rigid body model option is used for the impacting body (both the fuselage and beam models of wing and empennage are rigid). This choice arises from the fact that this modeling technique for the aircraft is the least expensive in terms of computational effort, and the lowest time step in the simulation arises from the water particles.

As referred in Subsection 4.2.2, the generation process of the aircraft’s rigid body model includes the computation of mass and inertial properties by the code (from specified mass point distribution), properties which are then assigned to the COG of the model. To ensure consistency is achieved and results are realistic, the aircraft model is first analyzed on its own for inertial parameters through a brief test in the solver. Properties and resulting model are shown in Table 5.1 and Figure 5.1, respectively.

Table 5.1: Mass and inertial properties of Rigid Body model used in water modeling investigations.

<table>
<thead>
<tr>
<th>Total Mass [kg]</th>
<th>Position of Center of Gravity [mm]</th>
<th>Principal Moments of Inertia [kg.mm²]</th>
</tr>
</thead>
<tbody>
<tr>
<td>72547</td>
<td>16490 0 -590</td>
<td>I1 4.28E+12 I2 3.22E+12 I3 1.13E+12</td>
</tr>
</tbody>
</table>

Figure 5.1: Scheme of Rigid Body model used in water modeling investigations.

5.1.1 Full Length Approach with FE-SPH Mesh

The first analysis is performed using a full length FE-SPH mesh for the water domain. Classic finite element volumes are placed in the area where less water deformation is expected (total of 579400 solid elements), while 475000 SPH particles fill the central section of the basin. This will be used as a reference case to assess the influence of different parameters further on.

In this simulation the SPH particles are orthogonally distributed (100 mm particle spacing) and each weights at around 1 kg. The 8-node FE solids measure 200 mm×200 mm×200 mm and contact between water media is made by tied connection between SPH particles and finite elements. No separation stress is defined between structure and water (suction is not included). The model is shown in Figure 5.2.

During the simulation, different phases over impact can be observed. The aircraft first impacts the water domain and then bounces, landing several meters ahead for 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. The results in terms of displacement in Z-direction, over 1000 ms of running time, are displayed in Figure 5.3.
Investigations Concerning Pool Size

The water pool has to have sufficient total mass to provide realistic impulse for the impacting body, and boundary conditions should be placed far away from the impactor to prevent wave reflection to affect kinematics and pressure distribution. To this aim, rather large water domains are typically used, as large as computational times allow for. In order to investigate the influence of total water volume on aircraft acceleration history, total cross-section width $W$ and height $H$ are varied in this paragraph, while maintaining the dimensions of the SPH region.
The length of the water pool remains constant. The FE region is varied to fit the sizes detailed in 5.4.

<table>
<thead>
<tr>
<th>Height ($H$) / Width ($W$)</th>
<th>7m</th>
<th>10m</th>
<th>15m</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 m</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>5 m</td>
<td>X</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10 m</td>
<td>X</td>
<td>X</td>
<td></td>
</tr>
</tbody>
</table>

**Figure 5.4:** Height and width dimensions of the pool for investigations concerning pool size.

The results are presented in the form of the acceleration of the center of gravity of the aircraft model in Figure 5.5. The acceleration curves are filtered with a CFC180 filter [59]. Since the results are very similar, accelerations after second impact and peak accelerations are enlarged for better visualization. The accelerations arising from pools with smaller width ($W = 7$ m) present the highest discrepancy from the pool with the largest dimensions ($H = 10$ m, $W = 15$ m). For larger width and height values, the results seem to converge towards the latter. Nonetheless, acceleration histories are nearly coincident.

**Figure 5.5:** Comparison of the acceleration time histories of the COG of the aircraft model with different dimensions of water pool cross-section (CFC180).

Table 5.2 presents the required CPU time for the simulations. Computational expense is fairly high for all cases, averaging at 16.4 hours. However, the FE used to extend the pool are in fact very cheap in terms of computational time, as the difference between smallest and largest pool is less than 15%.

**Table 5.2:** Comparison of computational time with different dimensions of water pool cross-section.

<table>
<thead>
<tr>
<th></th>
<th>$H = 3$ m</th>
<th>$H = 5$ m</th>
<th>$H = 10$ m</th>
<th>$H = 3$ m</th>
<th>$H = 10$ m</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$W = 7$ m</td>
<td>$W = 10$ m</td>
<td>$W = 15$ m</td>
<td>$W = 15$ m</td>
<td>$W = 7$ m</td>
</tr>
<tr>
<td>Total simulation time [h]</td>
<td>15.4</td>
<td>16.0</td>
<td>17.9</td>
<td>16.2</td>
<td>16.6</td>
</tr>
<tr>
<td>Relative time to reference</td>
<td>-3.8%</td>
<td>Reference</td>
<td>+11.9%</td>
<td>+1.3%</td>
<td>+3.8%</td>
</tr>
</tbody>
</table>
The computational time comparison enhances the advantage of combining classic FE and SPH method for water modeling, as pool extension can be achieved with FE without excessive time penalty, while reproduction of large fluid deformations is still possible by SPH. However, the general CPU time is still elevated due to the large amount of particles required. As acceleration history proves a large width should be used in order to avoid boundary effects, the following simulations will be performed with $W = 10 \text{ m}$. Total height will be maintained as $H = 5 \text{ m}$, as higher values do not show significant improvement and require more computational time.

**Energy propagation between water FE and SPH particles**

As both FE and SPH particles are used to model the water domain, a concern arises in the contact between these different interfaces. Although in [12], and other publications this hybrid method is validated against experimental results for simple geometries, the violent impact in full aircraft ditching simulations suggests that this factor should be further investigated. To this aim, the ratio between particle spacing and the size of the FE 8-node solids is investigated. In addition, results are analyzed using a penalty contact (the same as the one used between water and structure, see Subsections 3.1.3 and 4.4) instead of the previously applied tied contact between particles and finite elements for the water. SPH particle spacing is maintained at 100 mm, while FE length is varied between 100 mm, 200 mm and 400 mm. Results in terms of acceleration time histories are shown in Figure 5.6. As the first impact is representative and due to time constraints, only the first 300 ms are simulated for these investigations.

![Figure 5.6: Influence of FE to SPH element size ratio and contact between both (CFC180).](image)

As computational times are very similar concerning FE mesh refinement, CPU time will not be highlighted in this research. It is interesting to observe that the volume of the outer finite elements does not seem to have a significant influence in the acceleration history of the aircraft model. However, even though the first impact is thought to be representative for these studies, the referred factor may be of significance in the second impact and so it is surely not safe to ensure that volume or mass ratio between SPH and FE is negligible. Also of interest is the fact that the penalty contact shows similar results, as this technique provides a slight reduction (around 16%) in simulation time. Once again, further conclusions are not drawn in this subject due to the lack of validation possibilities.
5.1.2 Translating Periodic Boundary Conditions

In Table 5.2 it can be observed that computational times are considerably elevated when using a full length water domain. This is due to the high number of SPH particles necessary to fill the length of the pool covered by the aircraft during 1000 ms of impact. For a sufficiently detailed particle distribution in the vicinity of the impact regions, while still maintaining an acceptable amount of water particles, an option such as translating periodic boundary conditions (TPBC) is investigated. To this aim, two approaches are analyzed: in a first part, the SPH elements are left with no boundaries. As the water domain is relatively small, particles are re-entered with initial conditions before gravity pulls them away from the aircraft, making this a viable option. In a second attempt, a rigid shell container (fixed in space) is placed around the moving SPH domain along the simulated length. As described in subsection §4.3.2, the water domain follows the movement of the COG of the aircraft.

Figure 5.7: Translating periodic boundary conditions options, without (left) and with (right) shell container, SPH modeling of water domain.

TPBC can only be applied to SPH particles, as the mesh-based formulation of the FE solids previously used in the outer regions of the pool prevents elements to be re-entered at the beginning of the box defining the periodic region. Therefore, these elements are deleted. The width and height of the SPH domain are varied and accelerations compared for both cases (without and with shell container) in Figures 5.8 and 5.9. Computational times are shown in tables 5.3 and 5.4, respectively. Once again, only the first impact of the aircraft is analyzed. The reference simulation is taken as a 35 m long FE-SPH full-length pool (SPH domain length is 30 m), similar to the one used in the previous computations. The latter simulation takes around 1.6 hours of CPU time to be completed. The length of the TPBC domain in the remaining is constant at 10 m, a third of the corresponding length of the reference simulation.

By analysis of the acceleration time histories in Figure 5.8, the higher the cross-section in the free-boundary case, the closer the results get to the reference value. This is due to the higher mass of water present, increasing impulse as volume is increased. However, the time gain is lost when reaching a cross-section of $H = 3\, m$ and $W = 7\, m$, when the time saved by limiting the length of the SPH domain is overcome by the time penalty arising from the necessary increase in cross-section. One can immediately observe the high computational cost of the SPH particles. As oppose to the classic FE entities, an increase in the cross-section of the SPH domain comes with a large increase in CPU time, limiting the advantages of a purely SPH water domain even when combined with TPBC.

Table 5.3: TPBC without shell container: computational time with different sizes of SPH domain.

<table>
<thead>
<tr>
<th>Reference</th>
<th>TPBC $H = 1, m, W = 5, m$</th>
<th>TPBC $H = 3, m, W = 7, m$</th>
<th>TPBC $H = 5, m, W = 10, m$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computational time</td>
<td>1.6 h</td>
<td>0.4 h (-75%)</td>
<td>1.9 h (+19%)</td>
</tr>
</tbody>
</table>
Figure 5.8: TPBC without shell container: acceleration histories of different SPH domain dimensions.

Table 5.4: TPBC with shell container: computational time with different sizes of SPH domain.

<table>
<thead>
<tr>
<th>Reference</th>
<th>TPBC $H = 1\text{ m}, W = 5\text{ m}$</th>
<th>TPBC $H = 3\text{ m}, W = 7\text{ m}$</th>
<th>TPBC $H = 5\text{ m}, W = 10\text{ m}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computational Time</td>
<td>1.6h</td>
<td>0.8h (-50%)</td>
<td>4.3h (+168%)</td>
</tr>
</tbody>
</table>

Figure 5.9: TPBC with shell container: acceleration histories of different SPH domain dimensions.

When adding the boundary shell around the SPH domain with TPBC, limiting movement in Y- and Z-directions, the impulse is immediately higher as particles do not "escape" as easily. The results are
immediately closer to the reference acceleration, although convergence is not as evident. However, it is interesting to notice that for the same cross-section, when comparing the simulations with and without the shell boundary condition, CPU time nearly doubles. This is mainly due to the computational cost of the violent contact between the particles and shell container. In addition, the time gain is once again lost when increasing the number of particles to fit the results originating from the full-length FE-SPH approach.

**Coupling TPBC with FE volumes for expansion of the pool**

The option of periodic boundaries for SPH flow simulation leads to reduction of the length-wise domain necessary for the analyzed ditching. However, the investigations performed show that the necessary increase in cross-section of the SPH domain overtakes the time advantage expected. It was also discussed that hydrodynamic FE volumes provide lower CPU requirements and are effective to model the water regions with lower deformations. The possibility to combine these two modeling options was therefore considered. Since there was no obvious manner to extend the TPBC to finite elements, the first approach tested was to model a U-section of FE volumes, long enough for the duration of the event, and to define only a section of limited length at the interior with the TPBC moving along with the impacting structure. This technique is similar to the one used for the shell container in Figure 5.7, only with the hydrodynamic FE volumes instead of the shell elements. The problem arose as gravity acted on all the elements, leading to a collapse of the FE in the regions where particles were not present.

During the course of this research and upon this challenge, the ESI-Group was inquired about possibilities of the VPS solver to address the issue. Subsequently, a new algorithm was implement that puts to zero the acceleration for any node whose current position is outside the limits of the TPBC. A new routine is called each cycle when both volume elements and particles are present. If a periodic boundary is defined, the instant coordinate values for the SPH region is evaluated. Next, for all the solid elements of the parts assigned (the water FE volumes in this case), the position of the nodes is retrieved and compared with the limits of the region. If located outside, the accelerations and velocities are set to zero (the FE are "frozen"). If located inside, the elements are activated and contact is computed through a tied connection between both interfaces. This implementation is illustrated in Figure 5.10.

![Figure 5.10: Translating Periodic Boundary Conditions with FE water model.](image)

The algorithm was made available in a development version of the VPS solver and the technique is analyzed in Figure 5.11. 1000 ms of impact are again simulated, and results from a 100 m long FE-SPH full pool (SPH domain 95m long) are compared against the TPBC approach with an SPH domain of length 37 m, moving along with the center of gravity of the aircraft. The CPU times are the following:

- Reference, full length SFE-SPH pool: 16 h
- TPBC’s with FE U-section: 7.3 h (≈ -54%)
Both tested cases have a total cross-section of $H = 5m$ and $W = 10m$. Results show that the combined FE-TPBC option is, in fact, a viable alternative to the standard full-length approach to simulate the analyzed ditching event. CPU time is reduced by over 50%, and acceleration time histories are very similar. This option was subsequently incorporated in the developed pre-processing tool.

### 5.1.3 Active Box via Nodes

**Fixed-size active box coupled with FE volumes**

The algorithm developed to couple FE volumes and SPH particles with TPBC is also available for the active box via nodes option. Here, particles are placed throughout the full length of the pool, but only a small portion is activated at each time step (see §4.3.3). As in the TPBC case, a U-section of hydrodynamic FE volumes is placed around the SPH domain. FE in the vicinity of the active particles are also activated upon passage, and elements outside this region are set to zero velocity and acceleration.

To first assess the consistency of the active box option, a first case is tested with an active SPH region of constant size (37 m length). The acceleration time history is compared with the same classic full-length FE-SPH pool (100 m long) used for reference in the investigations with the TPBC above. Results are shown in Figure 5.12. CPU times were the following (reference is repeated for clarity):

- Reference, full length FE-SPH pool: 16 h
- Active box via nodes, constant length, with FE U-section: 7.5 h ($\approx -53\%$)

Computed accelerations are very similar, proving that the active box produces very consistent results with the full-length approach, while significantly reducing computational effort. CPU time savings are once again over 50%, as observed with the FE-TPBC technique. It is interesting to notice the slight (1%) increase in CPU time from the TPBC to the active box scheme. This is due to the fact that even though
particles are deactivated outside the defined region in the latter option, the solver still has to analyze all node positions for inspection and deactivation of particles. The main advantage to the active box is the ability to vary the length of the active region over the simulation, which is investigated in the next paragraph.

Adaptive active box coupled with FE volumes

In the previous simulations using TPBC and active box features, the length of the SPH domain is kept constant throughout the entire period. As observed, two separate impacts occur during the simulation. Since in the second impact the total bottom of the fuselage is in contact with the water, the length of the active domain (or TPBC) had to be fixed at around the fuselage length, 37 m. However, during the first milliseconds (first impact), only a small region in the rear part of the fuselage is in contact with the water domain, and the remaining water length is not of interest for the simulation. This induces “wasted” CPU effort. For improvement in terms time efficiency, an active box with variable length is tested. The length is linearly increased from 10 m to 37 m over the 1000 ms of simulation, maintaining constant cross-section dimensions. The simulation is illustrated in Figure 5.13.

The acceleration time history is compared with the reference full-length results in Figure 5.14. CPU times are described below (reference is repeated for clarity):

- Reference, full length SFE-SPH pool: 16 h
- Active box via nodes, constant length, with FE U-section: 5.7 h (≈ -64%)

The acceleration plot shows that this option provides almost the same results as the full length approach. Furthermore, this is the configuration that allows for highest time savings so far, at almost 65% less than the reference case. It is therefore concluded that the particle adaptive box option, in conjugation a U-section of FE volumes on the exterior, is the most adequate for such problems. It should be mentioned that there are still some further improvements that can be tested by taking advantage of this adaptive
Figure 5.13: Adaptive Active Box over simulation (from left to right, $t = 0 \text{ ms}$, $t = 450 \text{ ms}$ and $t = 900 \text{ ms}$). In the water pool: hydrodynamic FE volumes (light blue), activated particles (dark blue), deactivated particles (green).

Figure 5.14: Comparison of COG acceleration histories for full-length pool and adaptive active box options (CFC180).

feature. For example, in between both impacts, the aircraft is not touching the water domain, and as so no active fluid region is necessary during this period. This could be addressed by implementing a different function of variable length, as oppose to the linear increase previously tested. Another alternative is to link the node responsible for length growth (see §4.3.3) to one of the nodes of the aircraft model in some way, instead of directly prescribing growth rate.

As this remains the most efficient approach to the modeling of the water domain and results are consistent with the full-length reference simulation, the active box option will be used in the remaining simulations of the present research. Length will be varied or kept constant according to the simulation investigated (constant length if only the first impact is analyzed, adaptive length if full 1000 ms are simulated).

5.1.4 Particle Refinement

In addition to recently implemented features, several different particle sizes have been tested in order to quantify its influence on the simulated accelerations at the COG of the aircraft model. To this purpose,
the first 300 ms of impact are simulated using a 30 m long pool (active box with constant 10 m length). The ratio between particle spacing and individual finite element length is kept at 1:2. Four different SPH spacings are investigated, corresponding to the following mass and volumes for each particle:

- 600 mm particle spacing corresponding to an individual volume of $2.7 \times 10^7 \text{mm}^3$ (27 kg). Total of 6800 particles.
- 100 mm particle spacing corresponding to an individual volume of $1 \times 10^6 \text{mm}^3$ (1 kg). Total of 150000 particles.
- 70 mm particle spacing corresponding to an individual volume of $3.43 \times 10^5 \text{mm}^3$ (0.343 kg). Total of 463320 particles.
- 50 mm particle spacing corresponding to an individual volume of $1.25 \times 10^5 \text{mm}^3$ (0.125 kg). Total of 1200000 particles.

The accelerations are plotted in Figure 5.15, and CPU times are displayed in Table 5.5. Two main observations are retrieved from the collected data. First, the acceleration results were expected to essentially show less oscillations and to converge when using smaller particles, and such is not perceived. In fact, the two coarser meshes present relatively similar results even with a decrease of 1/27 individual particle weight, but the two finer meshes seem to diverge. The second observation is once again the exponential increase in CPU requirements when increasing the number of particles. The simulation with larger particles requires less than 2 minutes, while the finer mesh takes almost 10 hours to complete.

![Figure 5.15: Particle refinement investigations. Different particle spacings (CFC180).](image)

<table>
<thead>
<tr>
<th>dx</th>
<th>Computational time [h]</th>
</tr>
</thead>
<tbody>
<tr>
<td>300 mm</td>
<td>0.03 (1.8 min)</td>
</tr>
<tr>
<td>100 mm</td>
<td>0.6</td>
</tr>
<tr>
<td>70 mm</td>
<td>2.6</td>
</tr>
<tr>
<td>50 mm</td>
<td>9.8</td>
</tr>
</tbody>
</table>

As no experimental data is available, it is impossible to assess if the smaller particles produce in fact the most accurate results. However, the lack of convergence is of concern. As even smaller particles would
lead to unfeasible CPU times, no further spacings were tested. Trying to justify such divergence in the results, different options were considered. The contact between structure and water domain is performed through a master-slave penalty algorithm (see §3.1.3 and §4.4) which depends on the mass properties of both master and slave. For this reason, it was though that maybe the shell elements representing the rigid fuselage were too thin, and therefore too light when compared to the water particles, to portray realistic contact. To this aim, larger thicknesses for these elements were tested against different particle sizes. However, no significant differences were observed. A finer mesh was also tested for the structure to see if contact was altered, but again no interesting results were perceived. The final factor to be considered was the different critical time steps arising from the particle size. Similarly to classic finite elements, the critical time step for the SPH particles depends directly on their size. It was thought that maybe for larger SPH elements this critical time would become too large to accurately portray the contact between structure and water. To this aim, the time step was artificially constrained to a maximum value of $\Delta t_s = 0.006 \text{ms}$, both for 100 mm and 70 mm spacing, and accelerations were compared once again. The time step chosen is approximately one order of magnitude below the critical time steps initially registered. The results for these investigations are compared to the corresponding SPH sizes without time step constraint, and plotted in Figure 5.16.

![Figure 5.16: Particle refinement investigations. Different particle spacings with and without time step constraint (CFC180).](image)

### Table 5.6: Particle refinement: computational time with different particle spacings and time step constraint.

<table>
<thead>
<tr>
<th>dx=100 mm</th>
<th>dx=70 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>No constraint</td>
<td>With constraint</td>
</tr>
<tr>
<td>Computational time [h]</td>
<td>0.6</td>
</tr>
</tbody>
</table>

The original critical time steps were $\Delta t_s = 0.06 \text{ms}$ and $\Delta t_s = 0.04 \text{ms}$ for the 100 mm and 70 mm particle spacings, respectively. The accelerations show that when restraining the critical time step to the same maximum value, the results are much closer between simulations. Other than this, the results now
show 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. Other than this, other factors not considered in this research may weigh in the results.

The consequent penalty in overall CPU time should also be noted in Table 5.6. By constraining time step, computational time is significantly increased. The difference in time now arises from the computational effort required to compute the contact in between SPH particles, which are in higher number for the case with lower particle spacing. As with this time step constraint the results are very similar for both the particle sizes displayed, and due to the fact that the 70 mm particles require approximately 3 times the CPU effort, 100 mm spacing will be used in the following simulations. The time step will be constrained to a maximum value of $\Delta t = 0.006 \text{ ms}$. 
5.2 Investigations Concerning Aircraft Modeling Approaches

The following investigations aim to evaluate the capabilities and limitations of aircraft modeling approaches. The different options allowed by the developed pre-processing tool are tested. Similar to the path followed to the water investigations in the previous sections, the different models are first checked for mass and inertial properties in order to ensure realistic results.

As referred, the mass distribution is made by the tool by taking in the given mass point data. The rigid body with rigid wing scheme, where the total mass and inertia of the aircraft are computed by the tool and assigned to the COG of the aircraft, has been portrayed before in Table 5.1. The data for the remaining aircraft modeling approaches is shown and compared to the latter, considered as a reference case, in Table 5.7. For the rigid body option with flexible wing and empennage (RBFW), mass and inertia of the fuselage are first computed and attributed to the fuselage’s COG. The flexible wing implicates that mass is distributed through the beam nodes of the wing and inertia properties of the full aircraft arise from the coupling of the two parts. For the DMM, mass is distributed through the beam elements along the fuselage. Finally, for the more detailed GFEM/DFEM options, the resulting structural mass is first computed, followed by the addition of the remaining mass through the seat shell elements. Since all the models involve different techniques for mass distribution, general properties are first computed in the solver to ensure consistency. The coordinate system is the same as the one portrayed in Figure 5.1.

Table 5.7: Aircraft modeling investigations: mass, COG and inertia comparison (percentage is given in comparison with reference values shown in Table 5.1).

<table>
<thead>
<tr>
<th>Position of center of gravity</th>
<th>RBFW</th>
<th>DMM</th>
<th>GFEM/DFEM</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-position [mm]</td>
<td>16511 (+0.12%)</td>
<td>16718 (+1.4%)</td>
<td>16910 (+2.5%)</td>
</tr>
<tr>
<td>Y-position [mm]</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Z-position [mm]</td>
<td>-484 (-17.97%)</td>
<td>-372 (-36.95%)</td>
<td>-363 (-38.47%)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Principal moments of inertia</th>
<th>RBFW</th>
<th>DMM</th>
<th>GFEM/DFEM</th>
</tr>
</thead>
<tbody>
<tr>
<td>I1 [kg.mm²]</td>
<td>4.29E+12 (+0.23%)</td>
<td>4.32E+12 (+0.94%)</td>
<td>4.17E+12 (-2.57%)</td>
</tr>
<tr>
<td>I2 [kg.mm²]</td>
<td>3.22E+12 (+0.02%)</td>
<td>3.24E+12 (+0.62%)</td>
<td>3.06E+12 (-4.97%)</td>
</tr>
<tr>
<td>I3 [kg.mm²]</td>
<td>1.14E+12 (+0.89%)</td>
<td>1.14E+12 (+0.89%)</td>
<td>1.20E+12 (+6.19%)</td>
</tr>
</tbody>
</table>

The highest discrepancies in comparison with the reference case are observed in the z-position of center of gravity. However, the highest difference (around 38%) verified for the GFEM/DFEM approaches corresponds to an absolute value of less than 23 cm, which is considered not to be of high significance given the overall dimensions of the aircraft model (23 cm corresponds to 5% of the maximum fuselage height). For the remaining COG positions, the difference remains under 3%. For the principal moments of inertia, a maximum discrepancy of around 6% is observed. It is therefore considered that acceptable consistency is ensured. Further details on model characteristics are presented in the next subsections.

All the following simulations will be performed with an FE-SPH water pool, with the active box option and FE U-section surrounding the SPH domain. The active domain is kept with constant size throughout each simulation, as only the first impact will be analyzed due to CPU time limitations (first 400 ms). The total cross-section is constant with \( H = 5m \) and \( W = 10m \), and the SPH domain is kept at \( H = 1m \) and \( W = 5m \). Pool length is fixed at 38 m (total FE-SPH water domain). 100 mm particle spacing is maintained, and the time step will be constrained to a maximum value of \( \Delta t = 0.006 \text{ ms} \).
5.2.1 Rigid Body Model

The effect of modeling wing and empennage flexibility is first assessed for the rigid fuselage option. The objective is to assess if the deformation of the wing is reflected in overall kinematics. The distribution of stiffness properties for the wing and empennage is obtained from data relative to a generic transportation aircraft and assigned to the representative beam elements. Contact force histories in the Z-direction are compared to the rigid wing case (the same model used for water investigations) in Figure 5.17, and both CPU times displayed in Table 5.8.

![Figure 5.17: Contact force time histories in Z-direction of rigid body fuselage with rigid and flexible wing/empennage (CFC180).](image)

<table>
<thead>
<tr>
<th></th>
<th>Rigid Wing</th>
<th>Flexible Wing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computational time [h]</td>
<td>6.8</td>
<td>18.4</td>
</tr>
</tbody>
</table>

The force histories show a slight decrease in peak contact force and a discrepancy of around 30 ms (observe the last 50 ms of simulation in the plot) regarding the moment where the aircraft leaves the water domain (when contact force is zero). The CPU cost increases around 170% when modeling wing flexibility. The large increase in computational time arises from the small elastic beam elements utilized for representation of the wing, which induce a low critical time step for the simulation. It is important to notice that for the fully rigid case the smallest time step still arises from the SPH particles, and it is correspondent to the imposed value of $\Delta t_s = 0.006 \text{ ms}$. For the flexible wing and empennage study, the smallest time step is imposed by the structure and comes in at $\Delta t_s = 0.002 \text{ ms}$, a third of the latter case. In previous investigations (see Subsection 5.1.4), it was found that critical time step may influence the accuracy of representation of the contact between water and structure. Therefore, results in these investigations may be affected not only by the effect of elasticity of the wings in the fuselage, but also by the difference in time step in the two simulations. The latter will require further clarification; yet the influence of flexibility of the wing on global loads seems to be negligible for the case studied.
5.2.2 Dynamic Master Model

The next benchmark section addresses the capabilities of the DMM approach. The process is performed by the pre-processing tool which cuts the fuselage geometry into several sections length-wise (see Subsection 4.2.3). The model is sectioned in 87 parts in locations coincident with frame positions. Stiffness properties are automatically computed from the more detailed model (in §5.2.3) and assigned to the referred beams. The final stiffness distribution computed is shown in Appendix B (see Figure B.1).

![Dynamic master model](image)

**Figure 5.18:** Dynamic master model used in aircraft model investigations, with 87 length-wise sections.

The main objective of this investigation is to portray the effect of global structural flexibility in overall aircraft kinematics. All the shell rings attached to the elastic "skeleton" of the model are rigid, and so no skin deformation is allowed. To this aim, the DMM approach is compared to the rigid body option. Both models are tested with elastic wing and empennage elements (the DMM is tested with rigid wings later in Figure 5.20). Contact force histories in Figure 5.19 show that there is no significant difference between the results. In fact, although the computed forces from the DMM display higher oscillations, analysis show no significant difference between peak results or duration of the period that the aircraft remains in contact with the water basin.

![Contact force time histories](image)

**Figure 5.19:** Contact force time histories in Z-direction of DMM and RB models (CFC180).

**Table 5.9:** Dynamic master model investigations: computational times of RB and DMM simulations.

<table>
<thead>
<tr>
<th></th>
<th>Rigid Body</th>
<th>Dynamic Master Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computational time [h]</td>
<td>18.4</td>
<td>18.6</td>
</tr>
</tbody>
</table>
Table 5.9 confirms that almost the same CPU effort is required for both simulations. The critical time step of the elastic beam elements of wing structure and those of the beams of the fuselage are very similar, and as such when comparing both approaches with flexible wing models the required time is of the same order. Even though contact force histories do not show significant differences for this particular model, it is possible to visualize structural deformation throughout the fuselage. Figure 5.20 displays the model after 120 ms of simulation. Due to its small magnitude, deformation is amplified by a factor of 200 in the referred image. It can 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 5.20:** Deformation of DMM at $t = 120$ ms. Structural deformation is amplified in all directions by a factor of 200. Beam elements and water basin are not displayed for clarity purposes.

The collected data indicates that although a small deformation is present, in this particular aircraft model the structure provides a sufficiently high stiffness magnitude across the fuselage preventing larger deformations or changes to the contact force. However, it is thought that for a more flexible fuselage model, or for other (harder) impact conditions, this factor may be of importance. The DMM approach should therefore be studied for other cases in order to further validate its applicability in violent impact problematics such as aircraft ditching.

The following investigation aims to clarify if the wing elasticity affects the contact force in the globally flexible structure. For this, both rigid and flexible wing approaches are compared in Figure 5.21 and Table 5.10.

**Figure 5.21:** Contact force time histories in Z-direction of DMM with rigid and flexible wing (CFC180).
<table>
<thead>
<tr>
<th></th>
<th>Rigid Wing</th>
<th>Flexible Wing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computational time [h]</td>
<td>18.5</td>
<td>18.6</td>
</tr>
</tbody>
</table>

Contact force histories show that the effects of wing elasticity are not significant, similarly to what had been observed in the rigid body investigations (see §5.2.1). However, it had been noted that in the latter case the significant difference in simulation time steps could have influenced the mentioned results. For the DMM, both flexible and rigid wing approaches present similar time steps (as proven by the analogous CPU requirements), and as such this factor is thought not to be of relevance. The presented results for the DMM therefore confirm that, for the present model studied, wing and empennage elasticity does not have an expressive effect in aircraft kinematics during first impact. For this reason, the following simulations will be performed with models containing only rigid elements in these structures, accounting only for their mass and inertia contribution to the global aircraft model.

### 5.2.3 Global Finite Element Model

In previous approaches, all skin elements were rigid and as such no skin deformation was portrayed. In order to investigate the influence of skin deformation in kinematics and the possibility for skin rupture and consequent water ingress, the next benchmarking simulations are performed considering a global FE modeling approach. As meshing is automated by the pre-processing tool and in order to assess the impact of mesh refinement in the structure, two different element sizes are considered for the skin. In the coarser model the length of the skin elements is adjusted to the spacing in between frame positions in X-direction. In the finer model, the element size is reduced to half the length of the coarser elements.

![Figure 5.22: GFEM fuselage with coarser and finer meshes for aircraft modeling investigations (frames highlighted in red).](image)

As previously explained in Subsection 4.2.1, this modeling approach is far more detailed than the RBM and DMM models shown before, allowing for representation of interior structural elements and more meticulous skin representation. The present model consists of the following structural components: fuselage skin, stringers, frames, crossbeams, longitudinal floor beams and seats (including passenger masses). Variable skin thickness ranges between 1 mm and 3 mm for most of the fuselage surface, result of a static sizing process previously performed by an in-house pre-design tool. Skin material corresponds to an aluminum alloy. In the coarser mesh option, 7229 elastic shell elements represent the fuselage skin, and in the finer mesh 11144 shells are used. Frames and stringers are modeled as elastic beams. Further internal elements as seat rails and seat panels are not modified throughout both simulations. The total numbers of entities for the coarser (finer) aircraft model, including all elements, are: 126595 (134395) nodes, 23887 (28570) beam and bar elements, and 85004 (88919) shell elements.
It is important to refer that other than the required CPU time for the simulation, the automated tool for mesh generation takes around 45 minutes to generate each of the models. These are tested in the same water domain as the one used for the RB and DMM cases, over a total run time of 400 ms. In the coarser case, significant deformation on the impact region is observed, but no rupture of the skin occurs. This deformation is displayed in Figure 5.23 for $t = 390$ ms. The different displacements in the apparently non-deformed regions are due to the rotation of the aircraft, as displacement is shown in global coordinates.

The deformation presents one concave-like shape on the area that is in contact with the water basin. It is thought that rupture behavior is not accurately portrayed as all skin shell elements are attached by the sides to the fuselage frames. These stiffer elements are allowed to deform but not to rupture, as only simple elastic beams are used in the corresponding modeling. With the finer mesh, fissure is observed in the skin, as seen in Figure 5.24\(^1\). The deformation pattern is significantly different, as two concave regions are now detected.

\[\text{Figure 5.23: GFEM with coarser mesh: displacement in Z-direction | Zoom on impact region for } t = 390 \text{ ms.}\]

\[\text{Figure 5.24: GFEM with coarser mesh: displacement in Z-direction | Zoom on impact region for } t = 390 \text{ ms.}\]

\(^1\)In Figure 5.24, the elements are automatically deleted by the solver upon rupture, and as such the excessively deformed beam elements in the rear area (in light blue, extending to the left) are purely visual.
In both figures the beam elements of the wing and empennage are not displayed for clearer visualization of fuselage skin. However, it is important to notice that the horizontal tail model (empennage) is rigidly attached to the fuselage on the frame located the closest to the root of the horizontal tail. As this attachment is made in the rear part of the fuselage, close to the impact region, the artificial stiffening of the corresponding frame may have an influence on the results.

The contact force histories of the two models are computed and plotted in Figure 5.25. Computational time for the simulations is presented in Table 5.11. It can first be noted the computational effort required to run the models. The average CPU time between both comes at almost 3 days, around 2.7 times more as the DMM approach. On the other hand, the increase in time between finer and coarser meshes is lower than expected (around 1%). This can is explained by the critical time steps arising from the internal structures of the aircraft model, which are the same for both cases. The contact force results, although filtered with a CFC180 filter, still present considerable oscillations.

![Figure 5.25: Contact force time histories in Z-direction of GFEM with coarser and finer meshing (CFC180).](image)

<table>
<thead>
<tr>
<th></th>
<th>Coarser mesh</th>
<th>Finer mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computational time [h]</td>
<td>65.6</td>
<td>66.3</td>
</tr>
</tbody>
</table>

During the first 100 ms of simulation, the force histories present a similar behavior as the one observed with previous models. However, maximum values and tendency in the following 300 ms is considerably different. Considering mesh refinement, the contact force peak value presents a decrease in around 10% from the coarser to the finer mesh. Yet, both meshes are considered too coarse to accurately portray skin deformation and rupture. It can be seen that the violent impact results in considerable dislocation of the elements, and as such a much finer mesh is required. This is particularly true for the so-called impact region, since the remaining areas of the fuselage skin and the interior structural elements do not present significant deformation. For this reason, comparison between RBM, DMM and the latter more detailed approach will only be made in the next section.
5.2.4 Detailed Finite Element Model

Previous investigations with flexible elements for the fuselage skin have highlighted the importance of mesh refinement in the area first impacting with the water. For this reason the DFEM approach is considered in this section. The mesh is refined in the skin area of interest and local structural entities are modeled with shell elements as oppose to beam elements, benefiting from their enhanced structural description. In the region outside this refinement area the aircraft is modeled with a coarser mesh and beam elements (see §4.2.1). In the present study, a global mesh size is defined as with the same dimensions as the coarser mesh used in the investigations of Subsection 5.2.3. Three transition zones are defined from larger to smaller shell elements. The finer elements have an average edge length of 20 mm.

The same 400 ms of impact are simulated through the 38 m long pool. Displacement in Z-direction in the impact area for \( t = 390 \text{ms} \) is shown in Figure 5.26. Two views are displayed for analysis of local deformation and relative position of the refinement area. The contact force history is also computed in Figure 5.27. It is remarkable that this simulation requires almost close to 3.5 days to be completed.

Figure 5.26: Displacement in Z-direction for the DFEM approach after \( t = 390 \text{ ms} \).

Figure 5.27: Contact force time histories in Z-direction for DFEM approach (CFC180).
Rupture of elements in the rear section of the impact area is again observed, but in smaller scale than in the GFEM approach previously studied. The meshing parameters should be further calibrated through an iterative process to find optimum dimensions of the refinement area, as it is observed that the refined region is not large enough to accommodate the full area of higher distortion and possible failure. The contact force history shows less oscillations when compared to the GFEM approach. Forces increase up to a value of 150 kN but seem to reach a plateau of constant average force after that. Mesh refinement seems to have a considerable impact in the portrayal of skin deformation for this impact event, and therefore special care should be given to this factor in the pre-processing phase.

In order to compare the results achieved with the different aircraft modeling options in the last sections, the contact force histories of the rigid body, dynamic master and detailed finite element models are given in Figure 5.28. The plotted approaches are those with rigid wing and empennage beam models. For analysis of the effect on overall aircraft kinematics, the pitch angle time history is also plotted in Figure 5.29. Overall CPU expense of the three is compared in Table 5.12.

![Figure 5.28: Contact force time histories in Z-directions for RBM, DMM and DFEM models (CFC180).](image)

<table>
<thead>
<tr>
<th></th>
<th>RBM</th>
<th>DMM</th>
<th>DFEM</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Computational time [h]</strong></td>
<td>6.8 (100%)</td>
<td>18.5 (272%)</td>
<td>82.9 (1220%)</td>
</tr>
</tbody>
</table>

The results seem to largely differ between the first two less detailed approaches and the DFEM. Time increase is also remarkable: the latter approach takes approximately 12 times the necessary CPU effort of the rigid body model. In terms of the consistency of the results, several observations could be made. First, previous investigations have shown 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 is ruled by the maximum imposed limit of \( \Delta t_s = 0.006 \text{ ms} \), but the two other approaches have considerably lower time steps due to element size and elasticity (in the DFEM, \( \Delta t_s \) is \( 4\times10^{-4} \text{ms} \), approximately one order of magnitude lower). It was mentioned that this response should be further investigated.
This effect was further proven when comparing RB and DMM with flexible wings in §5.2.2. In this case, the time step is very similar (due to the elastic beams on the wing structure), and the results proved to be almost identical between both models. It is therefore considered that the significant decrease in time step, visible when comparing the three approaches with rigid wing in Figure 5.28 may be one reason for the disparity of the results. One option to verify this effect would be to artificially constrain the time step of the RBM and DMM to the value of the DFEM, to investigate any effect on the results.

Another factor for the different results between modeling approaches is the considerable deformation and consequent failure of the skin elements portrayed by the DFEM. Even though one can 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. The difference in rotation at $t = 390$ ms averages 28% between less and more detailed models. It should be noted that the data referring to the pitch angle time history of the DFEM was recorded from a structural node close to the COG location, since no direct access to the exact COG node was possible in the code (as oppose to the DMM and RBM approaches). This could lead to excessive oscillations in the results.

It is therefore advised to weigh in accuracy and efficiency costs, as the 3.5 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 portrayal 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 methodology followed in previous sections is valid for any model, and the automated pre-processing tool facilitates the testing of all options in coping with the best approach for the ditching analysis in question.
5.3 Investigations Concerning Separation Stress Feature

In the previous chapters of this research it was mentioned that the suction forces acting on the wetted region of the fuselage may have a critical influence on overall kinematics, specially on pitch behavior. This is because the suction forces act 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 full-scale ditching, investigations in [48] are taken as reference. In the latter research, a standard transportation aircraft geometry (of similar shape as to the one used in the present investigations) is tested with the separation stress feature available in VPS. The rigid body model used in the simulations was scaled-down by a ratio of 1:8 applying Froude scaling rules. Separation stress SPSTR and separation thickness factor SPTHK (see Section 4.4) were investigated until suction was realistically portrayed. Main model characteristics and results of this research are shown in Table 5.13.

Table 5.13: Data concerning separation stress investigations in [48].

<table>
<thead>
<tr>
<th>Model mass [kg]</th>
<th>Particle spacing [mm]</th>
<th>Optimum SPTHK</th>
<th>Optimum SPSTR [GPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>159.9</td>
<td>40</td>
<td>1.75</td>
<td>40</td>
</tr>
</tbody>
</table>

As models are similar, Froude scaling rules are employed to the input parameters of the separation stress in order to apply them to the present research. The separation thickness factor is a non-dimensional ratio between search radius for separation and regular contact height, and as so no scaling is applied. Scaling rules for stress state \( \sigma_{FS} = \left( \frac{L_{FS}}{L_{SS}} \cdot \sigma_{SS} \right) \) (where \( \sigma \) is stress, \( L \) length and subscripts FS and SS correspond to full-scale and sub-scale respectively). As so, the separation stress at full-scale is SPSTR=320 GPa. The rigid body model is chosen for consistency purposes. Particle length is kept at 100 mm.

In the reference investigation, the aircraft model remains attached to the water surface during the first 350 ms of impact. Similar results were expected for the full-scale case now tested. However, Figure 5.30 confirms that the behavior of the aircraft is not as predicted. At \( t = 300 \text{ ms} \), the aircraft is 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 the layer of particles that remain attached to the structure, contrary to the objective of simulating suction forces.

Figure 5.30: Separation stress investigations: simulation with SPSTR=320 GPa and SPTHK=1.75.

The results show that maybe due to the fact that particle size was not scaled to match the scaling of both aircraft and separation parameters, an adjustment to the latter factors is needed in the simulation. To this aim, the separation stress is first increased to SPSTR=1E3 GPa. The separation thickness factor seems to be of critical importance, in order to affect a larger mass of particles so that its total mass can keep the aircraft attached to the water, as oppose to the particle attachment observed in Figure 5.30.
Consequently, SPTHK’s of 2.5 and 5.0 were tested maintaining the 100 mm particle spacing. However, the results still show the same tendency, although a slight difference is observed in pitch behavior.

Since the results were still not as expected, a simulation with 300 mm particle spacing was performed (the hydrodynamic FE volumes are also increase to 600 mm length). SPRTHK factor was kept at 1.75 and SPSTR=1E3 GPa. The results are compared to the same configuration without separation stress feature. The pool was shortened to total 35 m length and only the first 300 ms were simulated.

The difference in pitch behavior is portrayed in Figure 5.31. It can be observed that in this case, the application of the separation stress feature results 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 that the suction effect could be portrayed for full-scale aircraft impacting on water. Despite initial difficulties in parameter scaling, it is thought that the results prove that the objective proposed was achieved. However, conclusions are made with caution as no experimental comparison is available, and therefore it is not safe to ensure that this behavior is, in fact, more realistic for the present model being investigated.

5.4 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 scheme has been used, running the full simulation in one node over eight cores. However, computational configuration may have an even larger effect in CPU requirements. For this reason, the present section aims to explore the capabilities of the multi-model coupling option.
MMC allows for the simulation domain to be separated in aircraft and water sub-models, each of which may run on its own time step. Contact is then achieved over super-cycles, where data from both sub-models 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. However, to ensure accuracy in terms of numerical stability (namely in terms of contact computation), time step ratio (TSR) between sub-models should be kept reasonable. Another feature to optimize hardware utilization is the possibility of assigning a different number of cores for each part (see Section §4.5).

The referred parameters are studied in [68] for the vertical impact of a fuselage section on water, using a computational cluster. These investigations conclude that optimum results are achieved when attributing the same number of cores to both structure and water, and that the TSR (equal to $\Delta t_{\text{water}}/\Delta t_{\text{struct}}$) should be kept below 30 to ensure correct contact computation. Taking these regards as a reference, the Multi Model Coupling option was tested for the present model with the GFEM approach (coarser mesh). One important notice is that the MMC option was only available for the commercial version of the VPS solver at the time of this research. 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, cannot be combined. To this effect, the GFEM simulation was repeated using a full length FE-SPH pool over 400 ms. The same simulation was then repeated using the MMC scheme. One node of the computing cluster is used and 4 cores are attributed to each sub-model.

The methodology followed for the MMC simulations is similar to the one performed in [68]. First, no time step ratio is artificially imposed, to verify the natural TSR of the full model. In this case, the average TSR is given at $\Delta t_{\text{water}}/\Delta t_{\text{struct}} = 73.8$. In a second step, TSR is artificially constrained by imposing the corresponding value in the water section of the simulation. In the present case, $\Delta t_{\text{water}}/\Delta t_{\text{struct}} = 30$ and a more conservative $\Delta t_{\text{water}}/\Delta t_{\text{struct}} = 5$ were tested. Finally, after the simulations were completed, the contact force histories were compared to ensure consistency. This data is plotted in Figure 5.32.

![Figure 5.32: MMC investigations: GFEM model with time step ratios TSR=30 and TSR=5.](image)
The contact force time histories show that the MMC results are not exactly consistent with the ones of the shared memory processing scheme previously used. This is particularly evident in the peak value at around $t = 150 \text{ ms}$, and in the behavior between $t = 250 \text{ ms}$ and $t = 300 \text{ ms}$. Furthermore, decreasing TSR does not seem to lead to more consistent results within the tested range. It might be possible that time step ratio has to be further lowered, as TSR=1 should exactly match the results where the MMC feature is not used. However, one should analyze the consequent CPU time savings displayed in Table 5.14.

Table 5.14: Computational times for SMP (No MMC) and MMC computational schemes with TSR=30 and TSR=5.

<table>
<thead>
<tr>
<th></th>
<th>SMP</th>
<th>MMC, TSR=30</th>
<th>MMC, TSR=5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computational time [h]</td>
<td>132.5</td>
<td>24.4 (-81.5%)</td>
<td>49.8 (-62.4%)</td>
</tr>
</tbody>
</table>

In the latter simulations only one node of the computing cluster with 8 cores is used, but the MMC potential can be extended to larger cluster assemblies. Further investigations are advised on the number of cores attributed to each part, as for the current case only one configuration was tested. Nonetheless, 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 for the model investigated, the remarkable decrease in computational expense should be taken into consideration. As computer technology evolves, memory processing schemes increase their capabilities, which could lead to a significant reduction in time spent analyzing more detailed aircraft models.
6 Conclusions

The main aim of this work was to explore advanced numerical methods for simulating the impact of full-scale aircraft on water, in conditions similar to a ditching event. The emphasis was put on the Finite Element and the Smooth Particle Hydrodynamics methods, and in particular in the state-of-the-art SPH features recently implemented in VPS within the European project SMAES. The challenge was first to fully automate model generation including these new features and different aircraft structural modeling approaches, and second to validate their applicability to the problematic of fixed-wing aircraft ditching involving high forward velocity, using full-scale models.

6.1 Achievements

The relevance of water impact for the aeronautical sector was first highlighted. To begin with, a brief review of airworthiness regulations concerning fixed-wing aircraft ditching was given. The particularities of the event were explained in terms of the typical forces involved and the complex fluid-structure interaction phenomena which arise from the high forward velocity of the aircraft upon impact. Finally, the current certification and analysis procedures used to investigate planned water landing were described, with emphasis on several numerical methods used in previous publications. It is concluded that finite elements are a viable option to model the aircraft structure, as this efficient method is broadly used by researchers to investigate other crash conditions. The problem is presented in the modeling of the water domain, which leads to prohibitive mesh deformation in conventional grid-based Lagrangian methods. To this aim, the SPH method is presented as a viable alternative as it is of mesh-free nature, and the fact that both structure and water can be modeled with Lagrangian formulations facilitates contact and coupling of both methods within the same explicit solver.

After summarizing SPH fundamentals, novel modeling techniques were introduced. The state of the art on coupled FE-SPH simulations of impact on water is given. During the course of the SMAES project, investigations were conducted to improve the simulation aircraft ditching. Within this research, the hybrid FE-SPH module of the VPS explicit solver, from ESI-Group, was extended by various features for improvement of water modeling and reproduction of FSI phenomena. The main objective of this thesis was defined as 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. These improvements have been validated in an experimental campaign of guided ditching tests [20], where representative plates are used to investigate the fluid-structure interaction under ditching conditions. It is proposed to contribute to the state of the art by testing some of the recently implemented improvements to the problematic of full-scale aircraft models impacting on water. To this aim, several water modeling options were considered, as well as different aircraft modeling techniques concerning complexity of the models.
One key element highlighted to allow for consistent benchmark studies and for incorporation of the ditching load case in bigger structural optimization tools was the necessary automation of the pre-processing phase of the analysis. With this objective in mind, a fully automated pre-processing script allowing for different models to be generated with reduced effort for the analyst was developed. In a second stage, the tool was used to generate models of a standard transportation aircraft geometry and testing the several options available.

The development of the pre-processing script was achieved using Python language and ANSYS software. Former existing tools developed in the DLR were included in the automated chain. As newly improved features are constantly being released by the ESI-Group, the code is organized in a fully parametrized matter to allow for new modules to be easily included. 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. First, four different aircraft models could be automatically generated based on the given data. The first two were meshed with recurse to the external tool AC-CRASH [56]: complete FE representation including flexible skin elements with uniform meshing, or with the modeling of refinement areas in the regions were violent impact is expected. The two other options consisted of simplifying the latter FE models by deleting internal structural entities and using rigid shell elements for the skin. For the dynamic master model option, stiffness is computed with recurse to another external tool and attributed to representative elastic beams along the fuselage. The same beams are coupled with rigid shell "rings", for representation of fluid-structure interaction. Another option is the rigid body model, where mass and inertias are automatically computed by the code and attributed to the center of gravity of the fuselage structure. As all model generation was automated, the user is now free to chose in between the different options without additional effort. Water generation follows and both modules, structure and water, are automatically combined by the pre-processing tool into one final ditching model.

Concerning water features, one can mention the application of translating periodic boundaries, which allowed moving the SPH domain with a velocity linked to the COG of the aircraft, 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. Coupling between FE and SPH elements for the water domain and in between structure and water was also automated, and the separation stress feature could be activated to simulate the suction effect. Finally, different memory processing schemes were incorporated, such as shared memory processing or multi-model coupling. The latter options require a specific organization of the generated input files for the simulation, and such was achieved by the developed tool. After pre-processing is concluded, the user can choose to directly start the referred simulation in the solver, and what is left to do at this point of the development is the post-processing of the results.

The different modeling options referred above were tested in several numerical studies. For this, a generic transport aircraft (approximately 37m long and weighing 72.5 ton) geometry was fed to the pre-processing tool and different models were generated. Water, aircraft, contact and computational approaches were studied. Concerning water investigations, the rigid body model is used for all simulations for time efficiency. The classic approach using a full pool with FE and SPH elements was used as a reference, and the influence of different parameters was studied. The main conclusions concerning these three approaches are described below.

- For the full length option, cross-section of the water domain should be large enough as to provide sufficient impulse and to remove boundary interference; FE volumes prove to be an efficient option to expande the cross-section of the fluid domain.
• 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 revealed to be efficient in producing consistent results and achieved a considerable reduction in CPU time (close to 55% compared to the classic full-length option).

• 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 size of the active domain over the period of simulation brought further improvements in terms of computational time (close to 65% compared to the classic full-length option), and results are practically the same as those of the full-length approach. This option was chosen to be used in the subsequent simulations.

Another important observation was the effect of particle refinement in the acceleration history of the aircraft. It was observed that a big discrepancy arises between larger and smaller particle modeling. The effect of time step reduction in contact computation was considered. It was shown that by artificially reducing critical time step, results may be more consistent. This was thought to indicate that due to the violent impact of the structure in the water, a small time step should be used in order for the code to correctly compute the contact between fluid and structural domains. However, several other factors may affect the conclusions mentioned before.

Investigation concerning aircraft modeling followed. It was proven that even though the developed pre-processing tool used different methods for mass distribution in each of the models, global properties were successfully consistent. Modeling approaches were tested to simpler to more detailed: first the RBM, followed by the DMM and GFEM schemes, and finally the DFEM. It was shown that for the aircraft model used in the simulations choosing to use flexible beams for wing and empennage structures does not have a significant influence in kinematics while decreasing computational efficiency. The dynamic master model correctly portrayed deformation along the fuselage structure, but for the tested case global flexibility does not seem to significantly influence acceleration time histories. The GFEM approach is tested with coarser and finer meshes, and it was observed that a fine mesh is necessary in the impact region to accurately portray skin deformation and eventual failure. To this aim, the DFEM option was considered. Overall conclusion is the following: skin deformation appears 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. However, time step influence in contact computation may be a factor. For the analyst, if global pitch kinematics is to be evaluated, the RBM approach gave 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 to be analyzed or accurate portrayal 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 methodology followed in the present research is valid for any model, and the automated pre-processing tool facilitates the testing of all options in coping with the best approach for the ditching analysis in question.
The separation stress feature was also tested, proving that modeling of suction forces in the rear part of the fuselage is possible. Finally, the Multi-Model Coupling was tested for the expensive GFEM option. Although accuracy is affected, a very significant time reduction is achieved with this computational scheme (over 80% for higher TSR). In conclusion, the latter option should be used but TSR should be verified with reference to conventional model.

Summarizing, the main contributions 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.

6.2 Future Work

The pre-processing tool was shown to be both practical and efficient for the investigations performed. Nonetheless, there is still a lot of room for improvement in the developed script. It can be referred the expansion of the available wing and empennage models to include skin panels and engine pylons, as contact between these structures and the water domain may be of significance. Other than this, new features for the VPS explicit solver are currently being released and some of the existing options improved, hence the tool should be updated regularly to accommodate for the generation of new input cards and thus take full profit of the algorithms available. Some scripts have been formulated for the post-processing of data after the simulation is completed, and their implementation in the current tool would for easier analysis of acceleration, pressure or forces.

Furthermore, the study of additional phenomena like cavitation and eventually ventilation should be performed. A cavitation model is already available in VPS, but this feature is not included in the present research. In addition, for more realistic results, a more complex lift model may be of interest as oppose to the linear function used in this research.

In recent investigations, non-uniform particle distributions have been considered. The implementation is complex due to instability issues arising in the SPH formulation, but results in [50, 69, 70] seem promising. This will allow in the future to refine particle size in the areas closer to impact, while still maintaining acceptable computation times.

Finally, the multi-model coupling option showed significant time improvements, as so did advanced options such as translating periodic boundaries or active box for the SPH domain. In the time of this research, both approaches could not be combined due to the present software architecture. Yet these features are now under development to be implemented in conjugation with distributed memory processing schemes, meaning that in the future this combination could lead to even more reduced computational times for detailed ditching models.
Bibliography


A Appendix: CPACS Data Format

Since 2005 the Common Parametric Aircraft Configuration Schema (CPACS) is developed by DLR for the exchange of information on the level of preliminary design. The system is in operational use at all aeronautical institutes of DLR and has been extended for civil and military aircraft, rotorcraft, jet engines and entire air transportation systems. Exemplary scheme is presented in A.1.

Figure A.1: Example of CPACS file in XML editor.

The data-format is based on XML technology. CPACS holds precise definitions for geometric elements like aircraft, wings, sections, elements, profiles, points, and transformations. The geometry description is parametric to ease changes in the geometry and hence optimization loops.
Appendix: Stiffness Distribution for Dynamic Master Model

Stiffness properties in Figure B.1 are automatically computed from the more detailed model (in §5.2.3) and assigned to the referred beams. Namely, bending stiffness in y- and z- direction ($EI_{yy}$ and $EI_{zz}$), and the crossed inertia $EI_{xz}$ are shown as a function of fuselage x-position.

![Graph of computed stiffness](image)

**Figure B.1:** Computed stiffness of GFEM approach, transferred to DMM model.