Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

UNCORRECTED PROOF

Development of a smoothed particle hydrodynamics method and its application to aircraft ditching simulations

Tianhang Xiao [a,b], Ning Qin [b], Zhaoyan Lu [a], Xuan Sun [a], Mingbo Tong [a], Zhengzhong Wang [c]

[a] *College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, 29 Yudao Street, Nanjing, 210016, PR China*
[b] *Department of Mechanical Engineering, The University of Sheffield, Mappin Street, Sheffield, S1 3JD, UK*
[c] *Science and Technology on Rotorcraft Aeromechanics Laboratory, Helicopter Research and Development Institute, 16 Hangkong Road, Jingdezhen, 333001, PR China*

ARTICLE INFO

Article history:
Received 2 November 2015
Received in revised form 9 February 2017
Accepted 24 February 2017
Available online xxxx

Keywords:
Smoothed particle hydrodynamics
Aircraft ditching
Water entry
Particle neighbor search
Dummy particle boundary condition
Weakly compressible SPH

ABSTRACT

The present study addresses the development and validation of a smoothed particle hydrodynamics (SPH) method, particularly to examine its feasibility and capability in hydrodynamics and dynamics of aircraft during ditching. The developed method solves the weakly compressible Navier–Stokes equations coupled with six-degree of freedom dynamics to achieve an accurate prediction of the interaction between the aircraft and the fluid. In this SPH method, a dummy particle wall-boundary condition is automatically implemented to meet the requirement of application on geometrically complex engineering problems. An efficient particle search strategy merging the ideal of Cell-linked list with Vertlet list is proposed to speed up the neighbor particles search process. The present SPH method uses an OpenMP memory-shared parallelization in conjunction with Z-curve reordering to accelerate the computation. Validations have been performed on several classic hydrodynamic problems, where good agreements were achieved via comparing with documented experimental results. The developed SPH method is applied to predict the ditching event of a complex helicopter model. Results demonstrate the ditching process, indicating that the method can be potentially used in aircraft ditching applications.

© 2017 Elsevier Masson SAS. All rights reserved.

1. Introduction

DITCHING is one of the most extreme emergency circumstances that ends with an intentional impact of the aircraft with water. Aircraft ditching on water is a very complex physical problem, which involves a wide range of disciplines such as kinematics, stability, hydrodynamics and structural engineering, thus makes the analysis of such problems a multi-disciplinary task. Additionally, the dynamic responses applied to the structure during an impact on water are significantly different from those happening during an impact on a rigid ground. The aforementioned facts eventually lead to the necessity of performing experimental and numerical investigations in order to improve the survivability and structure tolerance when aircraft ditching occurs.

Experimental model drop test particularly using the full-scale aircraft [1,2] is regarded as one of the most straightforward ways to predict the ditching events. However, due to the economical and practical reasons, scaled-model test with model launched in a water tank is more feasible especially when various ditching scenarios are considered. An example is the ditching test performed by Climent et al. [3] for the sake of investigating the ditching behavior of a military aircraft CN-235-300M where a 1/8 scaled model was used. More recently, an extensive test campaign of guided ditching has been conducted by CNR-INSEAN [4] within the working package of the FP7 research project SMAES aiming to provide available experimental database for validation. However, experimental tests are generally expensive, and thus only a narrow number of geometries can be tested during the aircraft design stage. There are also other limitations related to experimental testing, such as repeatability issues arose from the unensured precise ditching motion and the inherent but undesirable scale effects in hydrodynamics.

Numerical ditching analysis ranges from simple analytical method to numerical simulation methods. A variety of analytical/semi-analytical methods are documented to simulate water impact, e.g., von Karman's estimation method based on momentum theory [5,6], the simple analytical method introduced by Farhad [7] derived from the linear potential flow theory and the very recent semi-analytical method for a plate impacting on water problem [8]. In view of the fact that for highly non-linear hydrodynamic behav-

---

E-mail address: xthang@nuaa.edu.cn (T. Xiao).

http://dx.doi.org/10.1016/j.ast.2017.02.022
1270-9638/© 2017 Elsevier Masson SAS. All rights reserved.

Please cite this article in press as: T. Xiao et al., Development of a smoothed particle hydrodynamics method and its application to aircraft ditching simulations, Aerosp. Sci. Technol. (2017), http://dx.doi.org/10.1016/j.ast.2017.02.022

UNCORRECTED PROOF

ior of complex geometry with potential failure of realistic structures, analytical/semi-analytical approaches are inappropriate to solve this fluid–structure interaction problem. Numerical methods provide alternative solutions to predict the effects of aircraft ditching due to their abilities in solving this type of non-linear hydrodynamics.

During the past decades, the mesh-based Lagrangian methods, e.g. Finite Element Method (FEM), were widely applied to water impact. Vignjevic and Meo [9], Pentecote and Vigliotti [2], and Ortiz et al. [10] used the FEM to numerically simulate the vertical impact on water of the conventional helicopter structure for crashworthiness assessment purposes. However, the violent nature of the considered aircraft ditching problems causes the mesh-based Lagrangian methods to fail in accurately describing the fluid flow because they cannot handle significantly large mesh distortion. Arbitrary Lagrangian–Eulerian (ALE) methods were later considered as an alternative way to overcome the limitation of Lagrangian FEM due to its inherent continuous rezoning capability. With the freedom in moving the computational mesh offered by the ALE description, greater distortions of the continuum can be handled than that would be allowed by a purely Lagrangian method. This also provides more resolution than that offered by a purely Eulerian scheme. Jackson and Fuchs [11] simulated the water impact using ALE approach in the LS-DYNA software, which is a non-linear, explicit transient dynamic finite element model. Their results showed that the ALE method can provide reasonable pre-test predictions of the floor level acceleration responses. Hua et al. [12] proposed a three-dimensional dynamic structural model with the real geometry of an aircraft and an ALE fluid field model to simulate the fluid–solid interactions caused by low speed ditching using the LS-DYNA with experimental validation. Similar ALE model has been created by Hu et al. [13] and further applied to a full-scaled shape of Boeing 777–200 model. However, as a mesh-based method, the ALE approach relies heavily on high quality mesh and may inherently suffer from the poor mesh caused by large deformation and complex free surface geometries. Moreover, the ALE methods face some other potential problems, such as accuracy in free surface tracking, severe fluctuation of pressure field due to variation of mesh density and leakage modeling of surface-to-surface interaction.

Another mesh-based approach applied to water ditching of aircraft is the Finite Volume Method (FVM) coupled with free surface tracking method, e.g. the Volume of Fluid (VOF) method, where the aerodynamic and hydrodynamic forces can be derived. Wick [14] performed simulations to study the splashdown of an unmanned air vehicle (UAV) taking nose dive into seawater from various heights with a range of impact velocities. They used a time-accurate FVM based on the unsteady compressible ensemble averaged Navier–Stokes equations for the air and the unsteady incompressible ensemble averaged Navier–Stokes equations for the seawater. Guo et al. [15] used the Unsteady Reynolds-averaged Navier–Stokes (URANS) equations to numerically investigate the effect of pitch angle on the impact characteristics of water ditching. The transformation of the air–water interface is treated by the VOF model. Further study carried out by Qu et al. [16] imposed the global-moving-mesh method to deal with the relative motion between the water and the object. Results illustrated that the global-moving-mesh method can avoid the high computation expense and low-quality mesh in the conventional ALE mesh-deforming methods. One of the big issues of FVM for water impact problems is that a free surface tracking method is additionally needed and the accuracy of FVM is mostly dependent on the accuracy of free surface tracking. Similar to the mesh-based Lagrangian or ALE methods, the FVM method also suffers from the aspect of dynamic mesh deformation which not only increases additional computational cost but torments or even aborts computation if the mesh is not treated very carefully.

The meshless Lagrangian Smoothed Particle Hydrodynamics (SPH) method [17] is identified as well-suited for solving water impact problems by the violent nature of a ditching event which involves high deformation, nonlinear phenomena and complex free surface shapes. The SPH method divides the continuous medium into a set of particles and integrates the fluid governing equations on each particle in Lagrangian formula. The physical quantity of any particle is computed by an interpolation of the values of the nearest neighbor particles, and then particles move according to these quantities. For fluid flow, the pressure field is obtained by a weakly compressible state equation [18] or by additionally solving a pressure Poisson equation [19–21]. Without a mesh, the SPH method makes itself more convenient to describe the violent fluid deformation and could principally avoid the problem with distortion of the mesh. In addition, the Lagrangian frame will leave out the spatial discretization of the convection term in the governing equations, preventing the consequent diffusion. Recently, a few softwares or open-source SPH codes, e.g. SPHysics [22], are being developed and has been successfully applied on environmental, oceanic and coastal engineering problems [23–25]. Besides, the SPH method was also extended to be a potential useful tool in the aeronautical applications. Groenenboom et al. [25,26] implemented the SPH method into an explicit finite element program and one of their applications is to analysis ditching and floating behavior of a helicopter with external flotation system. The results demonstrated the capabilities of the SPH method to solve complex aircraft ditching problems. Later, within the European SMAES project, efforts have been made by Groenenboom [27,28], Benítez et al. [29], to extend the SPH module within the hybrid FE-SPH code VSP with various innovative feathers. Further works [30–32] that attempted to couple the SPH with the structural finite element model provided new possibilities to investigate the structure–fluid interactions with structure deformation taken into consideration during an aircraft ditching event.

The main work of this paper is to develop a SPH-based numerical method in order to investigate the specific hydrodynamics and dynamics of aircrafts during ditching events. The numerical framework of the in-house developed SPH code is introduced in Section 2, where a weakly compressible SPH method coupled with six-degree-freedom dynamics is presented. Also, an automatic dummy particle wall boundary treatment is developed for three-dimensional complex geometries to meet the requirements of engineering applications. Section 3 subsequently presents the solution strategies of SPH, high efficiency particles searching method and parallel computing issue, in which, the implementation strategies of efficient neighbor search and memory-shared OpenMP parallelization in conjunction with Z-curve reordering are highlighted. To assess the capabilities of the SPH method, several engineering applications are validated in terms of numerical accuracy and computational efficiency in Section 4.

2. SPH methods

2.1. Governing equations

The governing equations for the motion of an iso-thermal fluid in a Lagrangian frame are the continuity equation,

$$\frac{d\rho}{dt} = -\rho \vec{\nabla} \cdot \vec{v}, \tag{1}$$

and the momentum equation,

$$\frac{d\vec{v}}{dt} = \frac{1}{\rho}\vec{\nabla}\underline{\underline{\sigma}} + \vec{S}, \tag{2}$$

Please cite this article in press as: T. Xiao et al., Development of a smoothed particle hydrodynamics method and its application to aircraft ditching simulations, Aerosp. Sci. Technol. (2017), http://dx.doi.org/10.1016/j.ast.2017.02.022

Download English Version:

https://daneshyari.com/en/article/5472853

Download Persian Version:

https://daneshyari.com/article/5472853

Daneshyari.com