Commun. Comput. Phys.
doi: 10.4208/cicp.060709.060110a

Vol. 8, No. 2, pp. 265-288
August 2010

A Level Set Immersed Boundary Method for Water Entry and Exit

Yali Zhang[1,*], Qingping Zou[1], Deborah Greaves[1], Dominic Reeve[1], Alison Hunt-Raby[1], David Graham[2], Phil James[2] and Xin Lv[1]

[1] *School of Marine Science and Engineering, University of Plymouth, United Kingdom, PL4 8AA.*
[2] *School of Mathematics and Statistics, University of Plymouth, United Kingdom, PL4 8AA.*

Received 6 July 2009; Accepted (in revised version) 6 January 2010

Communicated by Boo Cheong Khoo

Available online 12 March 2010

Abstract. The interaction between free surface flow and structure is investigated using a new level set immersed boundary method. The incorporation of an improved immersed boundary method with a free surface capture scheme implemented in a Navier-Stokes solver allows the interaction between fluid flow with free surface and moving body/bodies of almost arbitrary shape to be modelled. A new algorithm is proposed to locate exact forcing points near solid boundaries, which provides an accurate numerical solution. The discretized linear system of the Poisson pressure equation is solved using the Generalized Minimum Residual (GMRES) method with incomplete LU preconditioning. Uniform flow past a cylinder at Reynolds number Re=100 is modelled using the present model and results agree well with the experiment and numerical data in the literature. Water exit and entry of a cylinder at the prescribed velocity is also investigated. The predicted slamming coefficient is in good agreement with experimental data and previous numerical simulations using a ComFlow model. The vertical slamming force and pressure distribution for the free falling wedge is also studied by the present model and comparisons with available theoretical solutions and experimental data are made.

AMS subject classifications: 6D05, 76T10, 65B99, 65E05

Key words: Level set method, immersed boundary method, slamming coefficient, water entry and exit, free surface, fluid-structure interaction.

---

*Corresponding author. *Email addresses:* yali.zhang@plymouth.ac.uk (Y. Zhang), qingping.zou@plymouth.ac.uk (Q. Zou), deborah.greaves@plymouth.ac.uk (D. Greaves), dominic.reeve@plymouth.ac.uk (D. Reeve), alison.hunt-raby@plymouth.ac.uk (A. Hunt-Raby), david.graham@plymouth.ac.uk (D. Graham), phil.james@plymouth.ac.uk (P. James), xin.lv@plymouth.ac.uk (X. Lv)

1 Introduction

Investigation of fluid structure interaction at the free surface is a classical hydrodynamic problem and has a wide range of applications particularly in the fields of naval architecture, civil and ocean engineering and physical oceanography. A flow singularity occurs when a body impacts the free surface, which gives rise to a high pressure peak localized at the spray root and makes water entry and exit problems difficult. In grid based numerical methods, there are two main strategies to handle a moving or deforming boundary problem with topological change, namely body conforming moving grids (Baum et al. 1996; Yan et al. 2007) and embedded fixed grids (Yang et al. 1997; Ye et al. 1999; Tucker et al. 2000; Fadlun et al. 2000; Tseng et al. 2003; Balaras et al. 2004; Yang et al. 2006; Lv et al., 2006). For the former method, the grid can be efficiently deformed in an arbitrary Lagrangean-Eulerian (ALE) frame of reference to minimize distortion if the geometric variation is quite modest. Boundary conditions can be applied at the exact location of the rigid boundary. However, if the change of topology is complex, it will be very difficult and time consuming to regenerate the mesh. Also difficulties arise in the form of grid skewness and additional numerical dissipation may be a consequence of the redistribution of the field variables in the vicinity of the boundary.

An alternative to body conforming moving grids is embedded fixed grids where the governing equations are usually discretized on fixed Cartesian grids. The method can also be divided into two major classes based on the specific treatment of the boundary cells; (1) Cartesian cut cell methods (Yang et al. 1997; Ye et al. 1999; Tucker et al. 2000) and (2) Immersed boundary methods (Fadlun et al. 2000; Tseng et al. 2003; Balaras et al. 2004; Yang et al. 2006; Lv et al., 2006). Although the Cartesian cut cell method was originally developed for potential flow, it has been applied and extended to the Euler equations, shallow water equations, Navier-Stokes equations to simulate low speed incompressible flows and flows with moving interfaces. It has the potential to significantly simplify and automate the difficulty of mesh generation. There are also a number of disadvantages inherent in the use of this method. It cuts the solid body out of a background Cartesian mesh, which can generate sharp corners and a variety of different cut cell types. Thus, extending this method to three-dimensions is not a trivial task. In addition, arbitrarily small cells arising near solid boundaries due to the Cartesian mesh intersecting a solid body can restrict the stability of the Cartesian solvers.

In the immersed boundary method the momentum forcing which is introduced to enforce the boundary condition of the body in the fluid can be prescribed on a fixed mesh so that the accuracy and efficiency of the solution procedure on simple grids is maintained. Bodies of almost arbitrary shape can be dealt with and flows with multiple bodies or islands can be computed at reasonable computational cost (Fadlun et al. 2000). The immersed boundary method has the advantage of simplified grid generation and inherent simplicity which allows the study of moving bodies (Mittal et al. 2005) on fixed Cartesian grids. Furthermore, the appropriate treatment in the immersed boundary method leads to a convenient method for computing forces acting on a body, namely lift and