Accepted Manuscript

Parallel embedded boundary methods for fluid and rigid-body interaction

C. Samaniego, G. Houzeaux, E. Samaniego, M. Vázquez

PII: S0045-7825(15)00110-3
DOI: http://dx.doi.org/10.1016/j.cma.2015.03.008
Reference: CMA 10582

To appear in: *Comput. Methods Appl. Mech. Engrg.*

Received date: 29 September 2014
Revised date: 16 March 2015
Accepted date: 18 March 2015

Please cite this article as: C. Samaniego, G. Houzeaux, E. Samaniego, M. Vázquez, Parallel embedded boundary methods for fluid and rigid-body interaction, *Comput. Methods Appl. Mech. Engrg.* (2015), http://dx.doi.org/10.1016/j.cma.2015.03.008

Parallel embedded boundary methods for fluid and rigid-body interaction

C. Samaniego[a,*], G. Houzeaux[a], E. Samaniego[b], M. Vázquez[a]

[a]Barcelona Supercomputing Center (BSC-CNS), Edificio NEXUS I, Campus Nord UPC Gran Capitn 2-4, 08034 Barcelona, Spain

[b]School of Engineering and Departamento de Recursos Hídricos y Ciencias Ambientales, Universidad de Cuenca, Av. 12 de Abril s/n., Cuenca, Ecuador

Abstract

The implementations of an updated body-fitted and a non body-fitted method to deal with the interaction of a fluid and a rigid body are described. The physics of the fluid is modeled by the incompressible Navier-Stokes equations. A parallel fluid solver based on the VMS (Variational Multiscale) Finite Element method serves as the basis for the implementation. For the rigid body movement, the Newton-Euler equations are solved numerically. To account for the interaction, the force that the fluid exerts on the rigid body is determined, on the one hand. On the other hand, the velocity of the rigid body is imposed as a Dirichlet boundary condition on the fluid. A fixed Eulerian mesh discretizes the fluid domain, except for nodes in the vicinity of the rigid body boundary for the case of the updated body-fitted approach. The wet boundary of the rigid body is embedded in the fluid mesh and tracked by a moving surface mesh. It is a distinctive characteristic of the updated body-fitted strategy that, in order to impose velocities on the interface, some of the nodes near the body surface are moved by using a local r-adaptivity algorithm to conform with this surface. By contrast, the non body-fitted approach uses kriging interpolation for velocity prescription over the fluid on the interface. Given that fluid nodes can become solid nodes and vice versa due to the rigid body movement, we have adopted the FMALE approach, a variation of the ALE method to keep the fluid mesh fixed. Algorithms to ensure high performance, like skd-trees to determine if a given spatial point is currently inside the solid, are also used. All these ingredients constitute two approaches that are both computationally efficient and accurate. Numerical experiments are presented to assess their performance comparatively.

Keywords: fluid and rigid-body interaction; embedded boundary method; Navier-Stokes; finite element method; parallelization; FMALE

1. Introduction

The detailed modeling of the interaction of a rigid solid with a fluid has been the object of intensive research [1, 2, 3, 4]. However, this is still a challenging subject that entails several difficulties. The problem can become even harder when a high performance computing implementation is sought. We tackle the problem by means of two strategies that aim at being both accurate and computationally efficient: an updated body-fitted approach and a non body-fitted one. We will refer to them as UBF and NBF, respectively, from now on.

The physical behavior of the fluid is mathematically modeled by the incompressible Navier-Stokes equations. The incompressible Navier-Stokes solver we use is a parallel solver based on a master-worker strategy, which can run on thousands of processors. It is implemented inside the Alya System [5]. Alya is a computational mechanics (CM) code with two main features. Firstly, it is specially designed for running with

*Corresponding author. Tel: +34934017295.
Email address: cristobal.samaniego@bsc.es (C. Samaniego)