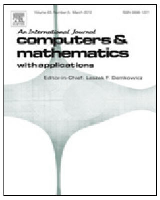
Computers and Mathematics with Applications 72 (2016) 1055–1065

Contents lists available at ScienceDirect

Computers and Mathematics with Applications

journal homepage: www.elsevier.com/locate/camwa



Modeling and parallel computation of the non-linear interaction of rigid bodies with incompressible multi-phase flow

Amir Malvandi [a,*], Amirmahdi Ghasemi [b], Rasoul Nikbakhti [c], Amirreza Ghasemi [d], Faraz Hedayati [e]

[a] *Department of Mechanical Engineering, Neyshabur Branch, Islamic Azad University, Neyshabur, Iran*
[b] *Mechanical Engineering Department, Worcester Polytechnic Institute, Worcester, MA, United States*
[c] *Department of Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad, Iran*
[d] *Department of Civil Engineering, Sharif University of Technology, Tehran, Iran*
[e] *Department of Engineering Technology and Construction Management, University of North Carolina at Charlotte, Charlotte, NC, United States*

ARTICLE INFO

Article history:
Received 14 November 2015
Received in revised form 14 June 2016
Accepted 16 June 2016
Available online 8 July 2016

Keywords:
Fluid–solid interaction
Level set
Immersed boundary
CFD
Free surface

ABSTRACT

A computational tool is developed to capture the interaction of solid object with two-phase flow. The full two-dimensional Navier–Stokes equations are solved on a regular structured grid to resolve the flow field. The level set and the immersed boundary methods are used to capture the free surface of a fluid and a solid object, respectively. A two-step projection method along with Multi-Processing (OpenMP) is employed to solve the flow equations. The computational tool is verified based on numerical and experimental data with three scenarios: a cylinder falling into a rectangular domain due to gravity, transient vertical oscillation of a cylinder by releasing above its equilibrium position, and a dam breaking in the presence of a fixed obstacle. In the first two validation simulations, the accuracy of the immersed boundary method is verified. However the accuracy of the level set method while the computational tool can model the high density ratio is confirmed in the dam breaking simulation. The results obtained from the current method are in good agreement with experimental data and other numerical studies. The applicability of the current computational tool for the interaction of a buoy in a water wave tank with two types of waves; symmetrical and asymmetrical waves; has also been studied.

1. Introduction

Fluid–solid interaction has been paid a lot of attention where one or more solid objects interact with a surrounding or internal fluid flow [1–4]. Since, analytical solutions to the Navier–Stokes equations are hard to obtain, numerical modeling has been employed to reach the fundamental physics involved in the interaction between solid and fluid.

Many models have been proposed for the interaction of fluid and solid which can be categorized into (a) approximation and (b) direct numerical simulation. In approximation models, either the viscosity or the inertial effect is neglected to simplify the governing equations. Such flows can be classified as potential flow and Stokes flow, respectively. In potential

---

[*] Corresponding author.
E-mail addresses: amirmalvandi@aut.ac.ir, a.malvandi@staff.um.ac.ir (A. Malvandi).

http://dx.doi.org/10.1016/j.camwa.2016.06.018

Nomenclature

ϵ	Controller of the thickness of the transition zone
μ	Dynamic viscosity (kg/ms)
ϕ	Distance function
ρ	Density of fluid (kg/m^3)
ρ_{f_1}	Density of fluid 1 (kg/m^3)
ρ_{f_2}	Density of fluid 2 (kg/m^3)
ρ_s	Density of solid (kg/m^3)
τ	Stress tensor (Pa)
υ	Volume of grid cell (m^3)
$\vec{\omega}_s$	Rotational solid velocity (rad/s)
$\vec{F}_b$	Body forces (N)
$\vec{F}_s$	Fluid–solid interaction force (N)
$\vec{r}$	Position vector relative to the center of mass of the solid object (m)
$\vec{V}$	Velocity vector (m/s)
$\vec{V}^*$	Intermediate velocity (m/s)
$\vec{V}_{solid}$	Solid velocity vector (m/s)
$\vec{V}_s$	Translational solid velocity (m/s)
$\vec{x}$	Location vector (m)
I_s	Moment of inertia of solid object (kg m^2)
Ib	Marker function of the solid object
M_s	Mass of solid object (kg)
n	Current step
$n + 1$	Next step
p	Pressure (Pa)
t	Time (s)
y	Local grid position vector (m)
y_t	Solid object interface location vector (m)

flow [5], separation does not occur and the pressure loss is underestimated, while in Stokes flow because the fluid inertial forces are neglected, the nonphysical motion of the solid object can be predictable [6].

In more advanced models which solve the full Navier–Stokes equations, no simplification is made. Numerical methods falling in this category are called direct numerical simulation (DNS). In DNS, the governing equations are coupled with the boundary conditions e.g. no-slip and no-penetration along the solid surface. Direct numerical simulation to capture the fluid–solid interaction can be further categorized into body conformal, immersed boundary, fictitious domain, lattice Boltzmann, and distributed Lagrangian multiplier methods [7,8]. Eulerian and Lagrangian approaches are used to solve the body conformal mesh method but in both methods the computational domain should be remeshed which is computationally intensive. Three subcategories of the body conformal mesh method are presented as (a) Deforming-Spatial Domain/Stabilized Space–Time (DSD/SST) [9], (b) Arbitrary Lagrangian Eulerian (ALE) [10] and (c) Fictitious Boundary Method (FBM) [11]. The aim of the above methods is to mitigate the issues associated with the body conforming unstructured grid methods, however, they cannot eliminate them completely.

In contrast of body conformal mesh method, the immersed boundary and fictitious domain (FD) methods are the most popular methods in tracking moving solid objects in fluid flow. The immersed boundary method uses a distribution function to interpolate the fluid velocity from an Eulerian grid onto the Lagrangian markers and to spread the forcing term computed at the Lagrangian markers onto the surrounding Eulerian nodes. The solid boundary interacts with the fluid by means of local body forces applied at the position of the solid points to the fluid. This body force imposes the kinematic constraint that the velocity at each of these solid point is coupled to the fluid velocity at that point. The introduction of these body forces has become the basic idea behind several fluid–solid interaction methods [12]. Three steps are typically involved in the immersed boundary method: (1) prescribing a force generating operator near the immersed boundary which affects the stability of the method, (2) spreading of forcing term in the fluid domain such that the integral represents total force. Discretizing the delta Dirac function in the spreading operator requires special treatments [13]. (3) solving equation of motion for the boundary which implies that the boundary is moving with the same velocity as the fluid at that location. In applications to rigid objects, forcing function can either be modeled as direct [14] or discrete forcing method [15]. In the direct forcing methods with high Reynold flows, Lagrangian points are utilized inside the solid domain which increases the computational cost. In the discrete forcing approach, the Navier–Stokes equations are first discretized without the forcing term. The discrete forcing method can be divided into three classes: discrete direct forcing, ghost cell, and cut cell methods, detailed descriptions of which can be found in [16,17].