Computers and Fluids 148 (2017) 56–68

Contents lists available at ScienceDirect

Computers and Fluids

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

An adaptive numerical method for free surface flows passing rigidly mounted obstacles[☆]

Kirill D. Nikitin[a], Maxim A. Olshanskii[b,*], Kirill M. Terekhov[c], Yuri V. Vassilevski[a], Ruslan M. Yanbarisov[d]

[a] Institute of Numerical Mathematics of the Russian Academy of Sciences, Moscow, Russia and Keldysh Institute of Applied Mathematics of the Russian Academy of Sciences, Moscow, Russia
[b] Department of Mathematics, University of Houston, Houston, TX, USA
[c] School of Earth, Energy & Environmental Sciences, Stanford University, Stanford, CA, USA
[d] Moscow Institute of Physics and Technology, Dolgoprudny, Russia

ARTICLE INFO

Article history:
Received 23 November 2016
Revised 7 February 2017
Accepted 9 February 2017
Available online 11 February 2017

Keywords:
Free surface
Incompressible flow
Mesh adaptation
Navier–Stokes
Octree meshes
Curvilinear boundaries
Sloshing container
3D Cylinder of circular cross-section
Flow around oil platform

ABSTRACT

The paper develops a method for the numerical simulation of a free-surface flow of incompressible viscous fluid around a streamlined body. The body is a rigid stationary construction partially submerged in the fluid. The application we are interested in the paper is a flow around a surface mounted offshore oil platform. The numerical method builds on a hybrid finite volume / finite difference discretization using adaptive octree cubic meshes. The mesh is dynamically refined towards the free surface and the construction. Special care is taken to devise a discretization for the case of curvilinear boundaries and interfaces immersed in the octree Cartesian background computational mesh. To demonstrate the accuracy of the method, we show the results for two benchmark problems: the sloshing 3D container and the channel laminar flow passing the 3D cylinder of circular cross-section. Further, we simulate numerically a flow with surface waves around an offshore oil platform for the realistic set of geophysical data.

1. Introduction

Free surface flows passing partially submerged objects are common in nature and engineering applications. The examples include water flows around bridge piers, ship bodies, water plants, or costal constructions. A mathematical model of such phenomena includes fluid dynamics equations and an evolution equation for the free surface. These equations can be posed in a domain of complex geometry. Handling the equations and the geometry numerically in an efficient and accurate way constitutes the major challenge for a CFD method applied to simulate free surface flows passing submerged obstacles. Depending on the applications, the fluid and free surface equations can be coupled to other mathematical models of transport, elasticity, etc. Thus, a reliable fast and accurate solver is desirable.

The previous studies of free surface flows passing submerged bodies include the simulation of Euler flows around hydrofoils [17], a boundary element method with the Lagrangian treatment of free surface evolution [21], a non-body conformal grid finite difference method for compressible flows [15], a stabilized finite element method for fluid equations in ALE form [34], and other FEM-based ALE techniques for fluid-structure interaction described in [2]. The variants of the immersed boundary method [30,37] for the free surface flows were discussed in [26,50]. Analytical and semi-analytical solutions of the free surface flows around specific submerged bodies were studied in [7,47].

The method developed in this paper is based on a hybrid discretization using octree Cartesian background meshes. Octree meshes enjoy a growing reliance in scientific computing community due to the simple Cartesian structure and embedded hierarchy, which makes mesh adaptation, reconstruction and data access fast and easy. In particular, octree meshes can be dynamically adapted towards the free surface. The adaptation can be also based on various error indicators. Fast remeshing with octree grids makes them a natural choice for the simulation of moving interfaces and free surface flows, see, e.g., [14,27,28,32,40,44], as well as more general non-Newtonian and high-speed Newtonian flows, see, e.g., [4,6,20,35,39,51]. The Cartesian structure of octree meshes requires,

☆ Supported by Russian Science Foundation through the grant 14-11-00434.
* Corresponding author.
E-mail address: molshan@math.uh.edu (M.A. Olshanskii).

http://dx.doi.org/10.1016/j.compfluid.2017.02.007
0045-7930/

however, a special technique for handling curvilinear boundaries and interfaces, since the mesh itself provides only the first order geometric accuracy in this case.

Using octree grids for the simulation of flows over partially submerged bodies gives the advantage of better local resolution of the free surface and fluid interaction with the body. For the more accurate treatment of the equations near the curvilinear boundary of the construction, we immerse the rigid object in the background mesh and construct the second order approximation of the fluid and free surface equations in the cut cells. The level-set method is used to recover the evolution of the free surface. Other important ingredients of our approach are the semi-Lagrangian characteristic method for the level-set equations on the dynamic octree meshes from [46], and the splitting method for the fluid equations on the octree meshes from [35] with filtering. In that paper, the method was studied for enclosed incompressible viscous flows in cavities and over bluff bodies.

Compared to well-studied higher-order finite volume and finite difference discretizations on uniform grids, the schemes that exploit adaptivity properties of octree meshes often pay the price of lower accuracy and higher numerical dissipation. This happens due to the presence of hanging nodes on irregular interfaces and non-uniform mesh size, which require interpolation of unknowns and make impossible certain cancellations of discretization errors. Such error cancellations take place for uniform grid due to the stencil symmetry. To overcome this loss of accuracy, we operate with a suitable sets of nodes and least-square minimizing interpolants. Further, we validate our approach by performing a series of numerical experiments. First, we compute a channel flow past a 3D circular cylinder. Second, we simulate the sloshing of water in a 3D tank subject to periodic horizontal excitation. The critical statistics, which are drag, lift coefficients for the first test and water levels for the second test, are compared against reference data found in the literature. The success of the numerical method for both benchmark problems demonstrates its ability to accurately simulate incompressible viscous free-surface flows and flows passing streamlined bodies with curvilinear boundaries. Therefore, we apply the method to simulate the water flow with surface waves around an offshore oil platform rigidly mounted in the Kara sea offshore. The platform is a reconstruction of a currently operating unit. The sea waves runup reproduces the realistic weather scenario in the region of the Kara sea offshore. The statistics of interest are water levels at the platform and forces experienced by the construction.

The rest of the paper is organized as follows. Section 2 reviews the mathematical model. Section 3.1 presents the splitting method for the numerical time integration. Section 3.2 discusses the details of the discretization on the gradely refined octree meshes. In Section 3.3 we devise the numerical treatment of the curvilinear boundaries embedded in the background mesh. Section 4 collects the results of numerical experiments.

2. Mathematical model

Consider a Newtonian incompressible fluid flow in a bounded time-dependent domain $\Omega(t) \in \mathbb{R}^3$ for $t \in (0, T]$. The fluid dynamics is governed by the incompressible Navier–Stokes equations

$$\begin{cases} \rho\left(\dfrac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}\cdot\nabla)\mathbf{u}\right) - \operatorname{div}\boldsymbol{\sigma}(\mathbf{u}, p) = \mathbf{g} \\ \nabla\cdot\mathbf{u} = 0 \end{cases} \quad \text{in } \Omega(t),\ t \in (0, T], \tag{1}$$

where $\boldsymbol{\sigma}(\mathbf{u}, p) = \nu[\nabla\mathbf{u} + (\nabla\mathbf{u})^T] - p\mathbf{I}$ is the stress tensor of the fluid, $\mathbf{u}$ is the velocity vector field, p is the kinematic pressure, $\mathbf{g}$ is the external force (e.g., gravity), ρ is the density, and ν is the kinematic viscosity. At the initial time $t = 0$ the domain and the velocity field are known:

$$\Omega(0) = \Omega_0, \quad \mathbf{u}|_{t=0} = \mathbf{u}_0, \quad \nabla\cdot\mathbf{u}_0 = 0. \tag{2}$$

We assume that $\overline{\partial\Omega(t)} = \overline{\Gamma_D} \cup \overline{\Gamma(t)} \cup \overline{\Gamma}_{\text{out}} \cup \overline{\Gamma}_{\text{in}}$, where Γ_D is the static boundary(walls), $\Gamma(t)$ is the free surface of fluid, Γ_{in}, Γ_{out} are inflow and outflow parts of the boundary, respectively. Note, that Γ_D, Γ_{in}, Γ_{out} may vary in time, in general. We assume the free surface $\Gamma(t)$ passively evolves with the normal velocity of fluid, i.e., the following kinematic condition is valid

$$v_\Gamma = \mathbf{u}\cdot\mathbf{n} \quad \text{on } \Gamma(t), \tag{3}$$

where $\mathbf{n}$ is the normal vector for $\Gamma(t)$ and v_Γ is the normal velocity of $\Gamma(t)$. Since the free surface flows we interested in this paper have large Weber numbers, we ignore the capillary forces and the boundary condition on $\Gamma(t)$ reads

$$\boldsymbol{\sigma}(\mathbf{u}, p)\mathbf{n} = \mathbf{0} \quad \text{on } \Gamma(t). \tag{4}$$

On the static part of the flow boundary, we assume the velocity field satisfies either no-slip boundary condition

$$\mathbf{u} = \mathbf{0} \quad \text{on } \Gamma_D, \tag{5}$$

or no-penetration and free-slip boundary conditions:

$$\mathbf{u}\cdot\mathbf{n} = \mathbf{0} \quad \text{and} \quad \frac{\partial(\mathbf{u}\cdot\mathbf{t}_i)}{\partial\mathbf{n}} = 0,\ i = 1, 2, \quad \text{on } \Gamma_D, \tag{6}$$

where $\mathbf{t}_i$ and $\mathbf{n}$ are tangential and normal vectors on Γ_D. We shall use the generic notation $\mathcal{B}\mathbf{u}|_{\Gamma_D}$ to denote boundary conditions (5) or (6) on Γ_D. We assume that $\mathbf{u}$ is given on Γ_{in} and $\boldsymbol{\sigma}(\mathbf{u}, p)\mathbf{n} = \mathbf{0}$ on Γ_{out}.

For computational purposes, we shall employ the implicit definition of the free surface evolution with the help of an indicator function. Let $\Gamma(t)$ be given as the zero level of a globally defined Lipschitz continuous *level set* function $\varphi(t, \mathbf{x})$ such that

$$\varphi(t, \mathbf{x}) = \begin{cases} < 0 & \text{if } \mathbf{x} \in \Omega(t) \\ > 0 & \text{if } \mathbf{x} \in \mathbb{R}^3 \setminus \overline{\Omega(t)} \\ = 0 & \text{if } \mathbf{x} \in \Gamma(t) \end{cases} \qquad \text{for all } t \in [0, T].$$

The initial condition (2) defines $\varphi(0, \mathbf{x})$. The kinematic condition (3) implies that for $t > 0$ the level set function can be found as the solution to the transport equation [36]:

$$\frac{\partial\varphi}{\partial t} + \tilde{\mathbf{u}}\cdot\nabla\varphi = 0 \quad \text{in } \mathbb{R}^3 \times (0, T], \tag{7}$$

where $\tilde{\mathbf{u}}$ is any (divergence-free) smooth velocity field such that $\tilde{\mathbf{u}} = \mathbf{u}$ on $\Gamma(t)$.

A numerical method studied in this paper solves the system of equations, boundary and initial conditions (1)–(7). The implicit definition of $\Gamma(t)$ as zero level of a globally defined function φ leads to numerical algorithms which can easily handle complex topological changes of the free surface. The level set function provides an easy access to useful geometric characteristics of $\Gamma(t)$. For instance, the unit outward normal to $\Gamma(t)$ is $\mathbf{n} = \nabla\varphi/|\nabla\varphi|$, and the surface curvature is $\kappa = \nabla\cdot\mathbf{n}$. From the numerical point of view, it is often beneficial if the level set function possesses the signed distance property, i.e. it satisfies the Eikonal equation

$$|\nabla\varphi| = 1. \tag{8}$$

3. Numerical method

The section describes the key ingredients of our numerical approach.