Computers and Fluids 153 (2017) 118–129

Contents lists available at ScienceDirect

Computers and Fluids

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

An incompressible immersed boundary solver for moving body flows using a cut cell discontinuous Galerkin method

Dennis Krause [a,b,*], Florian Kummer [a]

[a] Chair of Fluid Dynamics, TU Darmstadt, Otto-Berndt-Str. 2, Darmstadt D-64287, Germany
[b] Graduate School Computational Engineering, Dolivostr. 15, Darmstadt D-64293, Germany

ARTICLE INFO

Article history:
Received 21 November 2016
Revised 3 May 2017
Accepted 12 May 2017
Available online 15 May 2017

Keywords:
Discontinuous Galerkin
Immersed boundary method
Moving body flow
Extended/unfitted DG
Incompressible Navier-Stokes equations
Fluid-structure interaction

ABSTRACT

We present a higher-order immersed boundary method (IBM) for moving body flows, based on a discontinuous Galerkin (DG) discretization. In our method bodies are represented by a sharp interface approach using cut cells. The position of the bodies and their velocity is prescribed by Newtons equation of motion. Furthermore, we introduce a splitting type approach of coupling fluid and rigid body solver and point out the explicit calculation of hydrodynamic forces using hierarchical moment fitting. To verify our method we compare our new approach to a range of numerical benchmarks taken from the literature. At first we compare our method with calculations using a body fitted grid. Thereafter we will focus on representing boundaries with fixed motion to point out the accuracy and flexibility of our IBM. Afterwards we show results of full coupled test cases in which the explicit coupling approach and the calculation of forces mentioned above come into play. It can be shown, that with our method we can get similar results by decreasing the number of total unknowns significantly, which is the main advantage of the proposed scheme.

1. Introduction

Particulate flows with lots of moving bodies are of high scientific and technological interest with numerous applications. The understanding of such flows is important in chemical engineering (separation), life science (blood flow) and basic understanding of nature (sedimentation in ocean or river beds). On both, the macro- and microscopic scale, the phenomena of particulate flows are not fully understood and are still under ongoing research in research facilities and industry. Furthermore, there is still a lack of accurate and efficient solvers to tackle these type of flows numerically. Even with the computer power nowadays available on big research and industry clusters, the efficiency of such solvers is still a bottle-neck especially for industrial applications. This is due to the complexity of such particulate flows, meaning both, particle position and velocity are unknown.

Methods for particulate flows can be separated in two general groups: The first one is the so called Lagrangian approach which uses a mesh fitted to the particle surfaces. As the mesh can move arbitrary in the fluid, those methods are called Arbitrary Lagrangian Eulerian (ALE). The ALE method was used for particulate flows by Hu et al. [1,2] and Maury [3,4]. The second group are immersed boundary methods. The first immersed boundary method was proposed by Peskin [5] in the field of fluid-structure interaction for simulating flow patterns around heart valves. The new feature of this method was, that all calculations were done on a fixed Cartesian grid. It was not needed to remesh and project the solution onto the new grid in every timestep in order to be conform with the geometry. The success of his method was to impose the influence of the immersed boundary on the flow without remeshing. In the following years, various modifications on this method have been proposed and an overview can be extracted from Mittal and Iccarino [6]. Immersed boundary methods are commonly used for fluid-structure interaction (FSI) problems including particulate flows. A broad review on immersed boundary methods (IBM) for FSI can be found in the work of [7].

In context of particulate flows, IBM can be further differentiated by the coupling between fluid and particle interactions. The two variants are implicit and explicit coupling schemes. In the implicit coupling approach the forcing is incorporated into the flow equations before discretization. Important work in this field was done by Glowinski (e.g [8,9].) using body-force distributed lagrange multipliers and Patankar [10] using a stress distributed Lagrange multiplier ansatz to model the coupling forces. In the second approach the forcing is introduced after discretization.

* Corresponding author at: Chair of Fluid Dynamics, TU Darmstadt, Otto-Berndt-Str. 2, Darmstadt D-64287, Germany.
E-mail address: krause@fdy.tu-darmstadt.de (D. Krause).

http://dx.doi.org/10.1016/j.compfluid.2017.05.008
0045-7930/

First, the IBM was extended to Stokes flow around suspended particles [11] and Navier-Stokes flow around fixed cylinders [12]. The number of particles was increased by the scheme proposed by Höfer and Schwarz [13]. Further work has been done by Wang and Turek [14] who proposed a multigrid fictitious domain method. To track the particles a volume based integral function is used, which was first proposed by Duchanoy and Jongen [15]. In this context, an alternative scheme was proposed by Uhlmann [16], who combines Peskins regularized delta function approach [17] with direct forcing in a finite-difference context. The immersed boundary method was also used in a lattice-Boltzmann context, e.g. by Feng and Michaelides [18].

An important issue which should be mentioned in terms of immersed boundary methods with moving domains is the occurrence of spurious pressure oscillations. These oscillations are present in many different immersed boundaries methods [16,19–22]. Seo and Mittal [23] found out that the reason for those oscillations lie in a violation of the conservation law due to the appearance and disappearance of cells at the interface. They suggest to apply a cut cell approach with virtual cell merging to enforce mass conservation. Further attempts to eliminate pressure oscillations have been made by [19–21].

For decades, finite-volume (FV) methods have dominated the computational fluid dynamics community not only for single phase problems but for multi-phase problems like water-air interaction and particulate flows. In contrast, Discontinuous Galerkin (DG) methods, first proposed by Reed and Hill [24], became popular because of their ability to use higher-order ansatz spaces, like finite element (FE) methods, but still preserve conservation properties by definition, like FV. DG methods also have several additional advantages, e.g. it is easy to handle hanging nodes and local refinements because of their discontinuous ansatz spaces at cell boundaries.

The work in this paper is based on the extended discontinuous Galerkin (XDG) approach by Kummer [25]. Here, a sharp-interface representation is used. In order to treat the problem of high condition numbers for arbitrary small cut cells a cell-agglomeration procedure is employed. Using such a sharp-interface representation shifts the problem of accuracy and efficiency to the quadrature on those cut cells. For this, we use the hierarchical moment fitting strategy (HMF) first proposed by Müller et al. [26] and later extended in the work of Kummer [25]. To the best of the authors knowledge, there is no work using cut cell/extended DG methods with hierarchical moment fitting in connection with immersed boundaries to tackle particulate flow problems. However, extensive work in case of extended discretization methods has been done in context of extended FE methods (XFEM), first introduced by Möes et al. [27] and later used for fluid dynamics by Gross and Reusken [28]. Beside other authors working in the field of XFEM, the first actual cut cell DG method was presented by Bastian and Engwer [29].

This paper is divided as follows: In Section 2 the physical models for fluid and rigid bodies are presented as well as the calculation of the hydrodynamic forces. Section 3 is about the numerical discretization of the problem and the quadrature on cut cells. In Section 4 numerical experiments are presented to proof the accuracy of the solver. At first two testcases testing the immersed boundary method itself are introduced. This is followed by two test calculations where the full coupling is taken into account. We sum up in Section 5 with a conclusion and an outlook to ongoing work and possible extensions of our solver.

2. Governing equations

2.1. Incompressible Navier–Stokes equations

For introducing the immersed boundary method we define the following disjoint partitioning of the computational domain $\Omega \subset \mathbb{R}^2$:

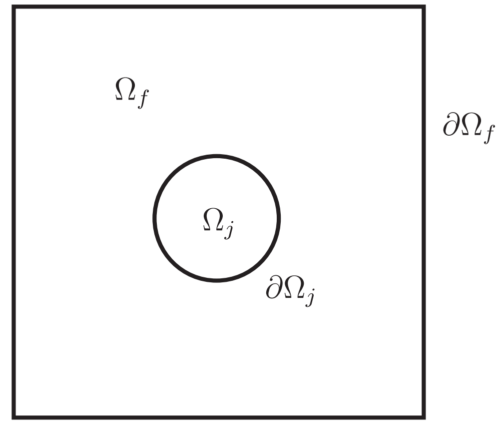

Fig. 1. Computational domain.

$$\Omega = \Omega_f(t) \cup \Omega_j(t) \tag{1}$$

with Dirichlet- and Neumann-type boundaries

$$\Gamma_D \cup \Gamma_N = \partial\Omega_f(t) \backslash \partial\Omega_j(t) \tag{2}$$

and body boundary

$$\Gamma_j(t) = \partial\Omega_j(t). \tag{3}$$

We restrict our setting to be two-dimensional. A schematic figure of the computational domain can be seen in Fig. 1. Therefore physical parameters like densities will be split to ρ_f in Ω_f and ρ_j in Ω_j.

The immersed boundary solver will be used to calculate incompressible flows in interaction with circular shaped bodies. The flow is described by the unsteady Navier–Stokes equations in the fluid region

$$\rho_f\left(\frac{\partial \vec{u}}{\partial t} + \vec{u}\cdot\nabla\vec{u}\right) + \nabla p - \mu_f \Delta\vec{u} = \vec{f} \tag{4a}$$

and the continuity equation

$$\nabla\cdot\vec{u} = 0 \quad \forall\, t \in (0, T) \quad \text{in } \Omega_f(t) \tag{4b}$$

with given initial and boundary conditions

$$\vec{u}(\vec{x}, 0) = \vec{u}_0(\vec{x}) \quad \forall\, \vec{x} \in \Omega_f(0) \quad \text{with} \quad \nabla\cdot\vec{u}_0 = 0 \tag{4c}$$

$$\vec{u} = \vec{u}_D \text{ on } \Gamma_D,\ (\boldsymbol{I}p - \mu_f\nabla\vec{u})\vec{n}_{\Gamma_N} = 0 \text{ on } \Gamma_N \text{ and } \vec{u} = \vec{u}_j \text{ on } \Gamma_j. \tag{4d}$$

In the equations above $\vec{u}$ is the velocity vector, p the pressure and $\vec{u}_j$ the body velocity. The fluid density is denoted by ρ_f, while $\mu_f = \rho_f \cdot \nu_f$ is the dynamic viscosity of the fluid. Furthermore, volume forces acting on the fluid are described by the force vector $\vec{f}$. Boundary conditions like Dirichlet- and Neumann-types have to be imposed on the outer boundaries $\partial\Omega_f(t) = \Gamma_D \cup \Gamma_N$ of the fluid domain and the presence of the body will be represented in the fluid domain by Dirichlet type boundary conditions for velocity on Γ_j.

2.2. Body motion

The rigid bodies are allowed to translate and rotate freely in the fluid domain. In the following we use capital letters for Lagrangian and small ones for Eulerian quantities. The body movement is described by the Newton-Euler Equations:

$$M_j \frac{d\vec{U}_j}{dt} = M_j\vec{f} + \vec{F}_j \tag{5a}$$