Comput. Methods Appl. Mech. Engrg. 267 (2013) 133–149

Contents lists available at ScienceDirect

Comput. Methods Appl. Mech. Engrg.

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

An interface-fitted subspace projection method for finite element simulations of particulate flows

Steffen Basting *, Rodolphe Prignitz

Department of Mathematics, Friedrich-Alexander-University Erlangen-Nuremberg, Cauerstr. 11, 91058 Erlangen, Germany

ARTICLE INFO

Article history:
Received 18 January 2013
Received in revised form 19 August 2013
Accepted 21 August 2013
Available online 30 August 2013

Keywords:
Particulate flow
Navier–Stokes equation
Finite element method
Fictitious domain method
Mesh optimization
Arbitrary Lagrangian–Eulerian formulation

ABSTRACT

A novel finite element method for the direct numerical simulation of particles in a Newtonian carrier fluid is presented. The proposed method is based on a fictitious or one-domain formulation and a subspace projection method to account for the rigid body motion of the particles. Underlying equations are posed in an ALE (arbitrary Lagrangian–Eulerian) formulation, allowing for moving computational meshes. The mesh is adapted to the particles by a novel mesh smoothing approach, guaranteeing both mesh optimality and a sharp representation of the particles' boundaries.

We show that by using quadratic Taylor–Hood finite elements for the discretization of the Navier–Stokes equations and isoparametric elements to represent the geometry, second order convergence with respect to the energy norm can be achieved (as opposed to a sharp $h^{1/2}$ result which holds if no mesh adaptation is performed).

We present numerical examples in 2d confirming the theoretical convergence results. Furthermore, we validate the approach by simulating the sedimentation process of a single particle and show the potential of the method by simulating a lid-driven cavity flow with 100 particles.

1. Introduction

Highly accurate and efficient direct numerical simulation (DNS) of particulate flows is of great interest in many technical applications and processes [9]. Let us just briefly mention a few scenarios.

DNS of particulate flows may serve as the main building block in multiscale models [19,30] or give essential insight into problems exhibiting complex behavior due to the presence of particles, for instance in gas flows loaded with particles [27]. Further application scenarios include complex interaction problems such as fluids containing ionized particles [18] or heat transfer processes [6,13].

In view of various application scenarios and the many numerical methods available, there is still potential to improve upon existing methods in terms of accuracy and efficiency. This article is mainly concerned with addressing the first topic and presents a highly accurate, yet efficient finite element approach.

Over the years, many different methods have been proposed for the simulation of particulate flows. In terms of particle representation, these methods can be divided in two categories: The first category consists of methods based on unstructured computational grids aiming at resolving the particle geometry by the mesh. This can be done either by concentrating the mesh around the particle [32] or by explicitly representing the particle by the mesh [12,17,11]. The latter approach is usually facing the need for remeshing if severe mesh deformation occurs. The second category consists of methods which do not rely on an explicit representation of the geometry and are commonly defined on structured grids. Let us mention

* Corresponding author. Tel.: +49 9131 8567228.
E-mail addresses: basting@math.fau.de (S. Basting), prignitz@math.fau.de (R. Prignitz).

0045-7825/$ - see front matter © 2013 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.cma.2013.08.014

fictitious domain methods (FDM) based on Lagrange multipliers [14,5,4] and immersed boundary methods [20,29]. Other approaches may be based on introducing an artificial interaction force to impose the rigid body motion on the particle, see for instance [26,31]. Due to their efficiency, these methods are often used for the simulation of large scale particulate flow problems.

As far as the error analysis is concerned, many questions are still open. For most of the mentioned methods, sharp error bounds for the spatial discretization are not available.

This article aims at combining the advantages of both categories (accurate representation of the particle and efficient solution techniques) by making use of two methods presented in [23,2]. The first method, the subspace projection method (SPM), is an efficient technique used to enforce rigid body motion inside the discretized particles. The second method addresses the discretization of the particles by aligning a given mesh to the particles' boundaries in an arbitrary Lagrangian–Eulerian (ALE) framework. The proposed alignment strategy guarantees mesh optimality while maintaining mesh connectivity. We give an error estimate for numerical solutions obtained on the particle aligned meshes. By formulating the equations in a moving coordinate system, both methods can be combined to yield an efficient and (provably) accurate algorithm which can be easily implemented in an existing Navier–Stokes solver due to its modularity.

2. Mathematical model

In this section we introduce a model for particulate flows and hereby follow [22]. For the ease of presentation we restrict ourselves to the 2d-case with one particle. The extension to 3d and/or more particles is straightforward, and a detailed description can be found in [23]. We denote by $\Omega(t) \subset \mathbb{R}^2$ the area occupied by a Newtonian fluid with homogeneous Dirichlet boundary condition on its outer boundary Γ_D. $P(t) \subset \mathbb{R}^2$ is the particle of constant density ρ_P and its Fig. 1center of mass is denoted by $\mathbf{X} = \frac{1}{|P(t)|}\int_{P(t)} \mathbf{x} dx$, while $\mathbf{r} = \mathbf{x} - \mathbf{X}$ is its relative coordinate. It should be emphasized that the fluid area and the particle area do not intersect, $\Omega(t) \cap P(t) = \emptyset$. We denote by $\Omega_c(t) = \Omega(t) \cup P(t) \cup \partial P(t)$ the *combined fluid/particle domain*. The unknowns in the fluid area are the velocity $\mathbf{u}$ and the pressure p, which are described by the Navier–Stokes-equations. The motion of the particle, being a rigid body, is described by Newton's law. The describing values are the translatorial and angular velocities $\mathbf{U}$, ω, respectively, the position $\mathbf{X}$ and the orientation in space given by the angle Θ. In order to follow the motion of the particle explicitly, we consider the governing equations in a moving coordinate system (ALE, arbitrary Lagrangian–Eulerian formulation [16,8]). We therefore assume that the combined fluid/particle domain can be expressed in terms of a reference domain $\hat{\Omega}_c$ and a time depending mapping

$$\begin{aligned} &\chi : \mathbb{R} \times \mathbb{R}^2 \to \mathbb{R}^2 \\ &\chi(t, \hat{\Omega}_c) = \Omega_c(t). \end{aligned} \tag{1}$$

We assume χ to be smooth and a homeomorphism for each time instant t. By defining the mesh velocity $\mathbf{y}$ for each t,

$$\begin{aligned} &\mathbf{y}(t, \cdot) : \Omega(t) \to \mathbb{R}^2 \\ &\mathbf{y}(t, \cdot) = \partial_t \chi(t, \chi(t, \cdot)^{-1}), \end{aligned} \tag{2}$$

the system in nondimensional form and ALE formulation reads

$$\partial_t \mathbf{u} + (\mathbf{u} - \mathbf{y}) \cdot \nabla \mathbf{u} + \nabla \cdot \overbrace{\left(p\mathbb{I} - \frac{1}{\mathrm{Re}} \mathrm{D}[\mathbf{u}] \right)}^{\sigma} = f \quad \text{in } \Omega(t), \tag{3}$$

$$\nabla \cdot \mathbf{u} = 0 \quad \text{in } \Omega(t), \tag{4}$$

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

$$\mathbf{u} = \mathbf{U} + \omega \times \mathbf{r} \quad \text{on } \partial P(t), \tag{6}$$

$$M\dot{\mathbf{U}} = \mathbf{F} + \int_{\partial P(t)} \sigma \cdot \mathbf{n} ds, \tag{7}$$

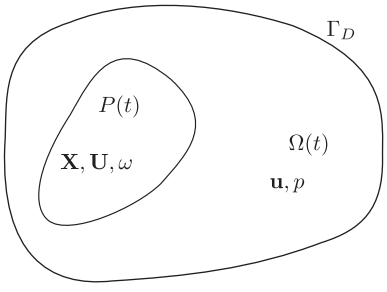

Fig. 1. Fluid domain $\Omega(t)$ and arbitrarily shaped particle $P(t)$.