Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 125 (2017) 443–449

Energy Procedia

www.elsevier.com/locate/procedia

European Geosciences Union General Assembly 2017, EGU
Division Energy, Resources & Environment, ERE

Is Magnus effect relevant for proppant settling in narrow fractures?

Jakub Poła*, Mateusz Bancewicz, Zbigniew Koza

Faculty of Physics and Astronomy, University of Wrocław, pl. M. Borna 9, 50-204 Wrocław, Poland

Abstract

Resolved Computational Fluid Dynamics-Discrete Element Method is an attractive approach for simulating multiphase flows of fluid-solid mixtures, as it includes a 4-way coupling. However, many of its modern implementations neglect the torque coupling between the fractions. Here we augment the CFDEM®coupling toolkit to allow it to capture this interaction and use it to simulate sedimentation of spherical grains in two-dimensional narrow fractures, with and without grain rotation taken into account. While the rotational effects turns out to change the dynamics of individual grains, it does not affect the average settling velocity significantly.

Peer-review under responsibility of the scientific committee of the European Geosciences Union (EGU) General Assembly 2017 – Division Energy, Resources and the Environment (ERE).

Keywords: CFD-DEM; four-way coupling; proppant settling; sedimentation; multiphase flows; fracture simulations

1. Introduction

Motion of proppant grains settling in a narrow fracture filled with a fracturing fluid is a complex phenomenon. To simulate it numerically, one has to take into account that each grain is subject to many forces of different nature, including gravity and buoyancy, hydrodynamical drag and lift, and mechanical interactions with other grains and the fracture walls. Another important effect is the influence of the proppant grains on the fluid velocity field. The problem is further complicated by the fact that the drag and lift are actually surface forces originating from a thin fluid layer around the solid body. To simulate it numerically, one can either use a very fine computational mesh near the solid-liquid interface or use a coarser mesh and add some effective forces that would compensate for the errors related to using a too coarse mesh. The downside of the first approach is that it permits to simulate only systems with a very small number of grains, usually too small to take into account collective effects properly, whereas the disadvantage of the second one is its inability to fully control the truncation errors imposed by the coarse lattice. One strategy that is often used consists therefore in striking the balance between the two approaches: one combines moderate-size meshes

* Corresponding author. Tel.: +48-71-375-9397; fax: +48-71-321-4454.
E-mail address: jakub.pola2@uwr.edu.pl

1876-6102 © 2017 The Authors. Published by Elsevier Ltd.
Peer-review under responsibility of the scientific committee of the European Geosciences Union (EGU) General Assembly 2017 – Division Energy, Resources and the Environment (ERE).
10.1016/j.egypro.2017.08.097

with effective forces that improve the accuracy of the results. Here, however, arises the problem of selecting optimal meshes and effective forces. In particular, one has to determine whether some effective forces acting on a solid body moving in a fluid, like the added mass force [1], Basset force [2], Saffman lift force [3] or Magnus lift force [4] are relevant in a particular physical situation or whether they can be neglected.

As an example of such dilemma one can consider the lift force. On the one hand its value should follow directly from the hydrodynamic pressure exerted by the fluid on the solid body, which in turn follows from the solution of the Navier-Stokes equations for the fluid pressure and velocity fields. On the other hand, the moving boundary conditions to these equations at the surfaces of proppant grains are very cumbersome to handle numerically. This rises the question of whether the solid-fluid interaction could be simplified without a significant harm to the solution. One possible simplification consists in neglecting the possibility of the proppant grain rotation. This corresponds to the assumption that while the boundary conditions to Navier-Stokes equations are moving along with the grains, they cannot rotate. From the physical point of view the problem is whether the change of the lift force acting on proppant grains due to their combined rotation and motion, known as the Magnus effect [4], is relevant for the fluids, proppant and fracture geometry typical of proppant sedimentation occurring during shutdown in hydraulic fracturing. This is the main question we tackle in this report. To answer it, we performed simulations similar to that reported recently in [5], where a two-dimensional model of monodisperse disks settling in narrow fractures filled with a viscous fluid was studied, and compared the results obtained with and without taking into account the possibility of the proppant grains to rotate.

2. Implementation details

In our study we use CFDEM®coupling toolkit [6] (https://cfdem.com), an open-source implementation of the CFD-DEM (Computational Fluid Dynamics-Discrete Element Method) algorithm, a popular method of simulating multiphase flows of solid bodies immersed in a viscous fluid [7]. In its most sophisticated, resolved version [8, 9], CFD-DEM allows for an efficient four-way coupling of particle-particle, particle-wall, particle-on-fluid and fluid-on-particle interactions. It also allows to capture in detail the complex fluid flow patterns formed between nearby particles. This, in turn, allows one to measure the velocity and the forces acting on each particle with high precision. While the CFDEM®coupling software implements all these features, one problem with it is that it does not take into account the hydrodynamic torque acting on the solid phase, therefore deliberately neglecting the Magnus effect. Therefore, we had to augment CFDEM®coupling toolkit with our patches that allowed it to simulate the lift force due to particle rotation.

2.1. CFD-DEM

The system we investigate is composed of two phases, the solid and the fluid one, each controlled be equations essentially different in nature. The dynamics of the fluid phase is governed by the (incompressible) local averaged Navier-Stokes equations, which are nonlinear partial differential equations (PDEs) for the pressure and velocity fields, whereas the evolution of the solid phase is described by Newton's equations, which are ordinary differential equations (ODEs) for the position and velocity of individual proppant grains.

To combine the two worlds of PDEs and ODEs, the CFD-DEM method solves the equations for the fluid and solid phases using some standard Computational Fluid Dynamics (CFD) and Discrete Element Method (DEM) solvers, respectively. The CFDEM®coupling toolkit accomplishes this general idea by coupling the OpenFOAM library [10, 11] for the CFD part with the LIGGGHTS library [12] for the DEM part. OpenFOAM employs the finite volume method (FVM), a mesh-based CFD algorithm that approximates the nonlinear PDEs governing the fluid flow with a system of algebraic equations, which are then linearized and solved. The LIGGHTS library, in turn, is specialized in computing the motion of thousands of solid particles by solving the appropriate Newton's equations. In contrast to OpenFOAM, the LIGGHTS solver is a mesh-free method—it does not need a computational mesh to compute the solution. It is also worth mentioning that both solvers used in any CFD-DEM implementation conceptually work in parallel and have to exchange some data to synchronize their states. The CFD solver sends to DEM the information about hydrodynamic forces and torques acting on each solid particle, but to calculate these parameters it has to know