International Journal of Multiphase Flow 91 (2017) 89–100

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Comparison of the implementation of three common types of coupled CFD-DEM model for simulating soil surface erosion

Yuan Guo, Xiong (Bill) Yu*

Department of Civil Engineering, Case Western Reserve University, Cleveland, OH, United states

ARTICLE INFO

Article history:
Received 2 May 2016
Revised 19 January 2017
Accepted 23 January 2017
Available online 27 January 2017

Keywords:
Computational fluid dynamics (CFD)
Discrete element method (DEM)
CFD-DEM
Soil erosion
Geotechnics of soil erosion
Fluid-solid interaction

ABSTRACT

Soil erosion is a common process studied by soil science, environmental engineering, geotechnical engineering, coastal engineering, and many other fields. In the areas of hydraulic engineering, the geotechnics of soil erosion remains a high priority topic as the bridge scour is a common cause of bridge failures. Accurate predictions of scour depth and soil erosion rate remain challenging, due to the limitations of existing scaled experimental approach in fulfilling the hydrological and hydrodynamic similarity requirements. Computational model offers a promising alternative to further the microscale understanding of soil erosion which can help to develop engineering tools in practice. Computational model that couples Computational Fluid Dynamics (CFD) and Discrete Element Method (DEM) to simulate the behaviors of fluid-solid systems is promising to advance the current tools for soil erosion analyses. Different mathematical forms for laminar fluid flows exist for the coupled CFD-DEM model as documented in published literatures and implemented in commercial and open-source software; each of them is based on certain physics assumptions and corresponding mathematical treatments. There are, however, no direct comparison of the results of CFD-DEM models based on these seemingly different mathematical formulations, which would help researchers to select the proper simulation tool. This study implemented coupled CFD-DEM models based on three most common types of mathematical formats used in the previous modeling work. The results of different CFD-DEM models are firstly validated by comparing the results of simulating the free settling of a particle in fluid. A case study is then designed to compare the models in simulating the surface erosion of cohesionless soil inside a pipe flow using laminar flow equations. Comparison indicates that for a relatively sparse particle-fluid system, the difference of the three models is negligible. For a dense particle-fluid system, simulation with the three different mathematical formats can predict different results (as large as 10% in the fluid velocity and 20% in the particle drag force for the simulation case study analyzed). The results of this case study indicate that the three CFD-DEM models achieved comparable results for simulating soil erosion from an engineering perspective, however, the differences between these models, which originate from their underlying physics assumptions, must be kept in mind in selecting an appropriate simulation model as well as in comparing the results from different models.

1. Introduction

Bridge scour is a common issue in hydraulic and geotechnical engineering. It has also been found to be the number one cause for bridge failures (Briaud et al., 2001). Accurate predictions of the scour depth and erosion rate under turbulent flow remain challenging, especially for cohesive soils. A major obstacle in understanding the erosion process is the complex interactions between water and soil as well as those among soil particles. The erodibility of soil is controlled by its physical properties, geochemical properties, and also affected by the biological activities (Martinez-Mena et al., 1999; Tejada and Gonzalez, 2006; Wynn et al., 2008; Grabowski et al., 2011; Dickhudt et al., 2011). Most of the current soil erosion models are based on the data from scaled laboratory experiments or in-situ observations. A simplified method in describing the erosion process is to consider fluid and soil separately (Sumer, 2007), where the erosion model is consisted of erosion criteria model and erosion rate model. Such simplified method, however, does not consider the particle-level soil-water interaction

* Corresponding author.
E-mail addresses: yxg223@case.edu (Y. Guo), xiong.yu@case.edu, xxy21@case.edu (X. (Bill) Yu).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2017.01.006
0301-9322/

under different flow conditions, i.e. laminar flow versus turbulent flow, which can have a major influence on the soil erosion process.

Mesoscale models considering the fluid-solid interaction have been introduced to simulate the interactions of soils with flowing water and to understand the soil erosion mechanism. Many of these studies are extensions of the models for fluid-solid systems, which are firstly introduced to simulate the behavior of fluidized beds based on the concept of locally averaged variables (Anderson and Jackson, 1967, 1969). The locally averaged variables are advantageous in treating the discontinuous fluid-solid system as a continuous system with equivalent properties, which help to significantly increase the computational efficiency. The interactions between solid and fluid is modeled with the drag force model (e.g., Wen and Yu, 1966). Based on this concept, Two Fluid Model (TFM) has been developed to simulate the fluid-solid interaction systems and is widely used in fields such as chemical and mechanical engineering (i.e., Monahan and Fox 2007; Bravo et al., 2007; Nguyen et al., 2009; Huang et al., 2010; Verma et al., 2013; Wang et al., 2014). This Eulerian-Eulerian model uses similar equation forms for the fluid phase and solid phase respectively based on volume fraction, and exchanges momentum between fluid and solid phases during the time-dependent calculation. The flow velocity and pressure in the Navier-Stokes equations are averaged by the porosity. To explain the influence of turbulence in particle-gas suspension, turbulence models, e.g., the $k-\varepsilon$ turbulence model, have also been implemented into TFM framework (Shah et al., 2013; Patro and Dash, 2014).

For granular materials such as sand, the physical and mechanical properties can be quite different from those of continuous materials. A physics based approach to describe granular materials is the Discrete Element Method (DEM) proposed by Cundall (1979). By using large amount of discrete particles, the DEM model is able to describe particle flows under Newton's Laws of Motion. In the research of fluidized beds, Tsuji (1992, 1993) proposed a coupling Computational Fluid Dynamics (CFD) model and DEM model to describe fluid-solid systems and demonstrated its advantages in predicting the fluid-solid interactions reasonably well. As an extension of TFM, the coupled CFD-DEM model combines the advantages of high efficiency in solving continuous fluid field by way of TFM and the realistic description of motion of discontinuous particle phase by DEM. It is therefore able to explain the behaviors of fluid-solid systems in the mesoscale, which are consistent with what observed by other researchers analyzing different types of applications (Zhu et al., 2007; Chen and Wang, 2014; Patil et al., 2015). This CFD-DEM model has also been used in geotechnical engineering to explain the complicated interactions between soil and fluid, e.g., debris flow, poromechanics, etc. (Zhao and Shan, 2013, Shan and Zhao 2014; Chen et al., 2011; Zhao et al., 2014).

Literature review on the CFD-DEM method has shown that there exist a few different mathematical forms for this Eulerian-Lagrangian model (Bouillard et al., 1989; Feng and Yu, 2004a). The major difference of these mathematical forms lies in the assumption for pressure exchange between fluid phase and solid phase, which is expressed as different pressure terms in the locally averaged Navier-Stokes equations. For the modeling of fluidized beds, it was believed that the variances in results due to different mathematical forms can be high and should not be ignored (Zhu et al., 2007). However, there has been no direct comparison of the results based on implementing different mathematical forms of CFD-DEM models.

This study aims to provide a direct comparison of the simulation results by three most common types of governing equations of the laminar CFD-DEM models. The results of different CFD-DEM model implementations are firstly validated by comparing with the standard experiment of free settling of a single particle in fluid. A testbed comparison case was then designed to compare three different CFD-DEM implementations on simulating soil erosion under pipe flow. It is assumed that sand particles, which is deposited inside a pipe under gravity, is subjected to pipe flow erosion. The results by different CFD-DEM simulation models are compared.

2. Mathematical formulations

The mathematical formulations describing the physics and mechanical basis of the CFD-DEM model are summarized in this section. Overall the fluid phase is described with the Navier-Stokes equations in Eulerian form; the solid phase is solved by Newton's Laws of Motion in Lagrangian form. Interactions between water and soil particles are calculated through a drag force model. During each coupling time step, fluid and particles will exchange data about their current statuses and update accordingly. The equation forms for particle and fluid phases in implementing the CFD-DEM models are described and compared in the following context.

2.1. Governing equations for the fluid phase

The Navier-Stokes equations have been averaged according to the volume fraction of fluid phase in the fluid-solid system. By adopting the concepts of locally averaged velocity and pressure, the fluid field is regarded as continuous in space even inside solid particles (Anderson and Jackson, 1967). The conservation of mass is described in Eq. (1). Three common mathematical formats for momentum conservation of the fluid, which are used in different CFD-DEM implementations, are listed in Eqs. (2a), (b), and (c).

$$\frac{\partial \mathrm{n}\rho}{\partial \mathrm{t}} + (\nabla \cdot \mathrm{n}\rho\mathbf{u}) = 0 \tag{1}$$

$$\frac{\partial \mathrm{n}\rho\mathbf{u}}{\partial \mathrm{t}} + (\nabla \cdot \mathrm{n}\rho\mathbf{uu}) = -\mathrm{n}\nabla \mathrm{p} + \mathrm{n}\nabla \cdot \mathbf{K} + \mathrm{n}\rho\mathbf{g} + \mathbf{f}^{\mathrm{p}} \tag{2a}$$

$$\frac{\partial \mathrm{n}\rho\mathbf{u}}{\partial \mathrm{t}} + (\nabla \cdot \mathrm{n}\rho\mathbf{uu}) = -\mathrm{n}\nabla \mathrm{p} + \nabla \cdot \mathrm{n}\mathbf{K} + \mathrm{n}\rho\mathbf{g} + \mathbf{f}^{\mathrm{p}} \tag{2b}$$

$$\frac{\partial \mathrm{n}\rho\mathbf{u}}{\partial \mathrm{t}} + (\nabla \cdot \mathrm{n}\rho\mathbf{uu}) = -\nabla \mathrm{p} + \nabla \cdot \mathrm{n}\mathbf{K} + \mathrm{n}\rho\mathbf{g} + \mathbf{f}^{\mathrm{p}} \tag{2c}$$

where, n is porosity from the solid phase; t is the time; $\mathbf{u}$ is the fluid velocity vector; ρ is the density of fluid; p is fluid pressure; $\mathbf{K}$ is stress tensor of the fluid field, and $\mathbf{K} = \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^{\mathrm{T}})$ for laminar flow; μ is the fluid dynamic viscosity; $\mathbf{g}$ is gravitational acceleration and g_z equals to $-9.80\,\mathrm{m/s^2}$; $\mathbf{f}^{\mathrm{p}}$ is volumetric fluid-particle interaction force, which equals to total concentrated interaction force (drag force & buoyant force) divided by the volume of the mesh element.

These three mathematical forms are based on different physical assumptions and have been used by different researchers (i.e., Eq. (2a) by Chen et al., 2011; Zhao et al., 2014; Eq. (2b) by Zhao and Shan 2013, Shan and Zhao, 2014; Tomac and Gutierrez, 2014; Guo et al., 2014; Eq. (2c) by Feng and Yu, 2004a; Jovanovic et al., 2014). Besides, different commercial or open-source software use different formats of these equations (i.e., PFC3D Coarse-Grid Fluid Scheme with Eq. (2a); OpenFoam-LIGGGHTS with Eq. (2b)). Computational models based on these different formats are referred as Model A, B, C respectively in this paper, which is also summarized in Table 1.

The major difference of these formats lies in the pressure and stress tensor terms in the momentum equation. In Model A, the momentum equation is normalized by extracting the porosity,