Computers and Fluids 000 (2018) 1–13

Contents lists available at ScienceDirect

Computers and Fluids

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

Benchmark solutions

A semi-Lagrangian splitting method for the numerical simulation of sediment transport with free surface flows

Sébastien Boyaval[a], Alexandre Caboussat[b,*], Arwa Mrad[c], Marco Picasso[c], Gilles Steiner[c,d]

[a] Laboratoire d'hydraulique Saint-Venant, Université Paris-Est (Ecole des Ponts), Chatou Cedex 78401, France
[b] Geneva School of Business Administration, University of Applied Sciences Western Switzerland (HES-SO), Carouge 1227, Switzerland
[c] Mathematics Institute, Ecole polytechnique fédérale de Lausanne, Lausanne 1015, Switzerland
[d] Ycoor Systems SA, Sierre 3960, Switzerland

ARTICLE INFO

Article history:
Received 15 January 2018
Revised 27 March 2018
Accepted 2 April 2018
Available online xxx

MSC:
65M60
65M08
65M55
76D05
76T10

Keywords:
Particle flow
Sediment transport
Mixture model
Free surface flow
Operator splitting

ABSTRACT

We present a numerical model for the simulation of 3D poly-dispersed sediment transport in a Newtonian flow with free surfaces. The physical model is based on a mixture model for multiphase flows. The Navier–Stokes equations are coupled with the transport and deposition of the particle concentrations, and a volume-of-fluid approach to track the free surface between water and air. The numerical algorithm relies on operator-splitting to decouple advection and diffusion phenomena. Two grids are used, based on unstructured finite elements for diffusion and an appropriate combination of the characteristics method with Godunov's method for advection on a structured grid. The numerical model is validated through numerical experiments. Simulation results are compared with experimental results in various situations for mono-disperse and bi-disperse sediments, and the calibration of the model is performed using, in particular, erosion experiments.

© 2018 The Authors. Published by Elsevier Ltd.
This is an open access article under the CC BY license. (http://creativecommons.org/licenses/by/4.0/)

1. Introduction

The modeling of sediment transport in rivers, lakes or shores is particularly relevant in hydraulic engineering to determine the amount and location of granular matters in the liquid. Sediments have indeed an influence on structural damages, operations efficiency and management, but also influence the efficiency of energy production in dam retention lakes. Moreover, the accumulation of river sediments in certain areas of the rivers modifies the natural environment, which might have important consequences for hydraulic energy production [1] or environmental regulations.

The modeling of sediment transport in a flow classically relies on a multiphase model. Two-phase flow models [2–4] use a second liquid field for the dilute sediment phase, with a different momentum equation in addition to that of the first liquid field, and possibly with a different rheology. The other alternatives are to macroscopically model the sediment concentration by an additional concentration field [5,6], which is well-validated at low concentration levels, or via the modeling of individual sediment particles at the microscopic level [7,8].

We focus here on sediments in suspension or accumulated in a Newtonian fluid (typically water). We investigate a macroscopic model for the sediment transport based on a sediment concentration with a single momentum balance for the mixture. The dilute concentration of sediments oscillates between zero and a maximal concentration corresponding to consolidated sediments.

The model proposed here couples the Navier–Stokes equations, with a volume-of-fluid approach for the tracking of the free surfaces between water and air, plus a nonlinear advection equation for the sediments' migration from low to high concentration areas. Since both dilute and undilute sediment concentrations in the liquid need to be described, a model able to describe not only the two phases but also the migration of the sediments from high to low concentration areas, and the resulting density variations, is chosen. This requires a *miscible model*, by opposition with, e.g., immiscible multiphase flow model [9]. As opposed to [9], here the advection equation for the tracers is *nonlinear* and concentrations vary along the Lagrangian trajectories.

* Corresponding author.
E-mail address: alexandre.caboussat@hesge.ch (A. Caboussat).

https://doi.org/10.1016/j.compfluid.2018.04.002
0045-7930/© 2018 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY license. (http://creativecommons.org/licenses/by/4.0/)

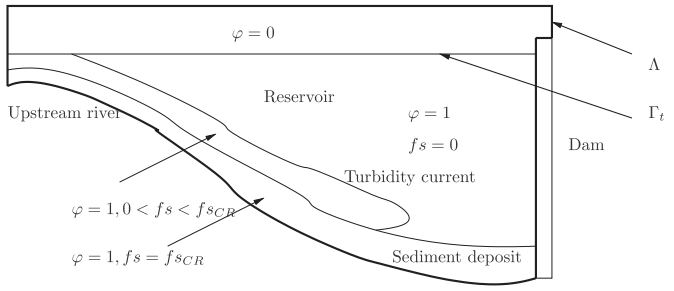

Fig. 1. Dam retention lake including sediment transport and deposition. 2D sketch of the geometrical domain.The cavity Λ is highlighted in bold. At each time $t \in (0, T)$, the liquid domain Ω_t is separated from the ambient air by the water-air interface Γ_t. The water domain is described by its characteristic function φ, while the sediment concentration fs is bounded in the liquid by its maximal value fs_{CR}.

A mathematical model for the simulation of Newtonian fluids with free surfaces, without sediment transport, has been presented and validated in [10–13], and has been applied to hydraulic engineering situations in [14]. It is extended here to include sediment transport. The addition of sediments has a direct effect on the density and viscosity of the flow. Reciprocally, the velocity of the flow is used to transport the sediment concentration, in addition to deposition effects due to the gravity.

An operator splitting approach allows to decouple the diffusion operator, the advection operator (by the mixture velocity) and the nonlinear transport operator for sediment deposition. A two-grids method couples a finite element discretization for the solution of a Stokes problem, with a finer structured grid of small cells for the discretization of advection operators and sediment deposition. While finite elements techniques are used for the approximation of the Stokes problem, a characteristics method and a Godunov method are used for the approximation of the linear and nonlinear transport problems respectively. Several numerical experiments validate the mathematical model presented in this work, starting with benchmark situations in simple geometries to real experiments for erosion problems, in which computational results are benchmarked against experimental results. Mono- and bi-disperse sediments are considered, and sensitivity analyses are performed.

The novelty of the proposed approach lies in the choice of a dedicated numerical method proposed to solve this multiphysics model, which couples sediment transport and free surfaces. The advocated splitting algorithm efficiently decouples the various physical phenomena and addresses each of them with dedicated techniques, involving finite elements, finite volumes and characteristics methods. Furthermore, there is no explicit tracking of the interfaces. The free surface between water and air is modeled by an Eulerian (volume-of-fluid) approach, while a diffuse interface modeling is used for the interface between water and the sediments.

This article is structured as follows. In Section 2, we describe the mathematical model for coupling the evolution of a Newtonian fluid with free surfaces with sediment transport. Sections 3 and 4 detail respectively the time and space discretizations. The results of numerical experiments for various test cases are presented in Section 5.

2. Mathematical model

Let us consider Λ a bounded domain in $\mathbb{R}^3$ with a sufficiently smooth boundary. Typically, we can consider a water reservoir or a dam retention lake, a sketch of which is illustrated in Fig. 1 in two space dimensions.

Let $T > 0$ be the final time of simulation. For any given time $t \in (0, T)$, let $\Omega_t \subset \Lambda$ be the domain occupied by the fluid (mixture including sediments), so that the remaining part of the domain Λ is occupied by the ambient air. Let Γ_t be the free surface between the liquid and the ambient air; it is defined by $\Gamma_t := \partial\Omega_t \backslash \partial\Lambda$ (namely the boundary of the liquid domain that is not in contact with the boundary of the whole cavity).

The mathematical model reads as follows. First let us describe the set of unknowns. Let Q_T denote the space-time domain containing the liquid, that is $Q_T = \{(\mathbf{x}, t) : \mathbf{x} \in \Omega_t, 0 < t < T\}$. The liquid domain is described by its characteristic function φ: $\Lambda \times (0, T) \to \{0, 1\}$, which implies that $Q_T = \{(\mathbf{x}, t) \in \Lambda \times (0, T) : \varphi(\mathbf{x}, t) = 1\}$. Assuming that the liquid domain Q_T is known and sufficiently regular, in the liquid region, the velocity field $\mathbf{v} : Q_T \to \mathbb{R}^3$ and the pressure field $p : Q_T \to \mathbb{R}$ are assumed to satisfy time-dependent, incompressible Navier–Stokes equations, with variable density and viscosity coefficients, and an additional Darcy-like reaction term modeling the porous solid matrix [15,16]. Finally, for the various classes of sediments, the sediment concentrations are defined in the liquid domain as $fs_i : Q_T \to [0, fs_{CR}]$, where fs_{CR} is the maximal sediment concentration. The set of corresponding equations read as follows. The evolution of the mixture (water and sediments) domain $\Omega_t \subset \Lambda$ is modeled by means of a volume-of-fluid method. Let $\varphi : \Lambda \times (0, T) \to \mathbb{R}$ be the characteristic function of the liquid domain Q_T. The function φ equals one at the point $(\mathbf{x}, t)$ if the liquid is present, zero if it is not. In order to describe the kinematics of the free surface, φ must satisfy (in a weak sense):

$$\frac{\partial \varphi}{\partial t} + \mathbf{v} \cdot \nabla \varphi = 0 \qquad \text{in } \Lambda \times (0, T), \tag{2.1}$$

where $\mathbf{v}$ outside Q_T is a regular extension of $\mathbf{v}$ inside Q_T (see, *e.g.*, [17]). More precisely, $\mathbf{v}(\mathbf{X}(t), t) = \mathbf{v}(\mathbf{X}(0), 0)$, where $\mathbf{X}(t)$ is the trajectory of a fluid particle which is at position $\mathbf{X}(0)$ at time $t = 0$, thus $\mathbf{X}'(t) = \mathbf{v}(\mathbf{X}(t), t)$.

The characteristic function of the liquid domain φ is given at initial time, which is equivalent to defining the initial liquid region $\Omega_0 = \{\mathbf{x} \in \Lambda : \varphi(\mathbf{x}, 0) = 1\}$. The initial velocity field $\mathbf{v}$ is prescribed in Ω_0 (see below), and boundary conditions are given on the inlet part of $\partial\Omega$.

Together, we consider a poly-dispersed model for the miscible sediment in the liquid. Assuming M populations of sediments (differing by size and/or density and/or shape), the presence rate of a sediment population is denoted by the solid fractions $fs_i : Q_T \to [0, 1]$ for $i = 1, \ldots, M$. This presence rate is a percentage of solid sediment in a given volume. The total amount of sediment

$$fs = \sum_{i=1}^{M} fs_i$$

is actually limited by a critical maximum value $fs_{CR} < 1$ that essentially depends on the shape of the sediment particles. In practise, if we consider a mono-disperse model with solid spherical particles and without consolidation, this value is approximately equal to 0.63.

We assume the liquid mixture velocity and pressure $\mathbf{v} : Q_T \to \mathbb{R}^3$ and $p : Q_T \to \mathbb{R}$ satisfy, in Q_T:

$$\rho(fs)\frac{\partial \mathbf{v}}{\partial t} + \rho(fs)(\mathbf{v} \cdot \nabla)\mathbf{v} - 2\nabla \cdot (\mu(fs)\mathbf{D}(\mathbf{v})) + \alpha(fs)\mathbf{v} + \nabla p = \rho(fs)\mathbf{g}, \tag{2.2}$$

$$\nabla \cdot \mathbf{v} = 0. \tag{2.3}$$

Here $\mathbf{D}(\mathbf{v}) = 1/2(\nabla \mathbf{v} + \nabla \mathbf{v}^T)$ is the symmetric deformation tensor, $\mathbf{g}$ denotes the gravity field, and $\rho(fs)$ (resp. $\mu(fs)$) is the density (resp. viscosity) of the fluid-sediment mixture. The coefficient $\alpha(fs)$ is a Darcy-like penalization term. All physical coefficients depend on the sediment concentrations fs_i, $i = 1, \ldots, M$. More precisely,

Please cite this article as: S. Boyaval et al., A semi-Lagrangian splitting method for the numerical simulation of sediment transport with free surface flows, Computers and Fluids (2018), https://doi.org/10.1016/j.compfluid.2018.04.002

Download English Version:

https://daneshyari.com/en/article/7155836

Download Persian Version:

https://daneshyari.com/article/7155836

Daneshyari.com