Applied Mathematical Modelling 37 (2013) 3478–3494

Contents lists available at SciVerse ScienceDirect

Applied Mathematical Modelling

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

CFD methodology for sedimentation tanks: The effect of secondary phase on fluid phase using DPM coupled calculations

Roza Tarpagkou *, Asterios Pantokratoras

Laboratory of Hydraulics and Hydraulic Structures, Democritus University of Thrace, Department of Civil Engineering, V. Sofias 12, GR-67 100 Xanthi, Greece

ARTICLE INFO

Article history:
Received 30 January 2012
Received in revised form 18 July 2012
Accepted 17 August 2012
Available online 29 August 2012

Keywords:
Computational Fluid Dynamics
Sedimentation
Multiphase flow
Momentum transfer
Turbulence

ABSTRACT

Computational Fluid Dynamics (CFD) methods are employed in order to simulate the 3D hydrodynamics and flow behaviour in a sedimentation tank. Unlike most of the previous numerical investigations, in the present paper the momentum exchange between the primary and the secondary phase is taken into account, using a Lagrangian method (discrete phase model) with two-way coupled calculations. By computing particle trajectories the proposed numerical model can track the momentum gained or lost by the particle stream that follows that trajectory and these quantities can be incorporated in the subsequent continuous phase calculations. Thus, while the continuous phase always impacts the discrete phase, the effect of the discrete phase trajectories on the continuum can be incorporated. This interchange affects fluid velocity, especially in the case of large particles sizes, which have a greater relaxation time in relation to the characteristic time of the tank. The present investigation compares a series of numerical simulations for a sedimentation tank with varying particle diameters and volume fractions, in order to identify the influence of the secondary phase to the primary phase and vice-versa and the way that this influence affects the efficiency of the tank.

1. Introduction

Sedimentation tanks constitute one of the most common types of water-treatment processing units. They are used in water-treatment facilities in order to remove the majority of the suspended solids, by the mechanism of gravitational settling. There are many important considerations that directly affect the design of the sedimentation process such as the local climatic conditions, the variations in the plant flow rate, the types of the tank inlet and outlet, the applied method for the sludge removal, the cost and geometric configuration of the tank and finally the suspended particles settling velocity [1].

Hydrodynamics and particle settling are the most important parameters in order to estimate the sedimentation process efficiency due to particle–fluid interaction. For this purpose, in the present paper Computational Fluid Dynamics (CFD) methods are used, in order to simulate and study the settling processes that take place within a sedimentation tank.

Many researchers have focused on the study of sedimentation tanks for wastewater treatment. Larsen [2] was probably the first who applied a CFD model to several secondary clarifiers. Shamber and Larock [3] used a finite volume method in order to solve the Navier–Stokes equations, the k–ε turbulence model equations and a transport equation for solids concentration where the settling velocity is incorporated in order to model secondary clarifiers. McCorquodale et al. [4] developed a model using a combination of finite element methods (for the stream function) and finite difference methods (for the boundaries). McCorquodale and Zhou [5] investigated the effect of various solids and hydraulic loads on circular clarifier

* Corresponding author. Tel.: +30 25413 00365.
E-mail addresses: rozatarpagou@hotmail.com, rtarpag@civil.duth.gr (R. Tarpagkou).

http://dx.doi.org/10.1016/j.apm.2012.08.011

performance, whereas Zhou et al. [6] linked the energy equation with the Navier–Stokes equations to simulate the effect of neutral density and warm water into a clarifier model.

In a primary sedimentation tank, where the solids concentration is limited and discrete settling prevails Imam et al. [7] applied a fixed settling velocity and used an averaged particle velocity. Stamou et al. [8] simulated the flow in a primary sedimentation tank using a 2D model in which the momentum and solid concentration equations were solved. Adams and Rodi [9] used the same model and did extensive investigations on the inlet arrangements and the flow through curves. An even more advanced work can be found in the paper by Lyn et al. [10] that accounts for flocculation where six different size classes with their respective velocities were considered.

A similar investigation but for potable water treatment exists in the literature only in the work of Goula et al. [11], who modeled a sedimentation tank for potable water and examine the influence of a feed flow control baffle (a) and the effect of influent temperature variations (b), but without accounting the interaction between primary and secondary phase. Additionally Wang et al. [12] simulated the flow field and SS (Suspended Solid) concentration in a rectangular sedimentation tank.

In addition, sedimentation procedure is used in other fields such as in the industrial area. Kahane et al. [13] modelled the thickener operation at Worsley Alumina refinery in order to reduce the operating costs. White et al. [14] investigated the fluid flow in feedwell thickener tanks. Farrow et al. [15] studied flocculation in order to improve thickener performance.

The particle–fluid interaction has been studied by some researchers, such as Righetti and Romano [16], Sbrizzai et al. [17], Hetsroni [18], Li et al. [19], Reeks [20], but for other applications different than sedimentation tanks.

So far, many researchers have used CFD simulations to study water flow and solids removal in settling tanks for sewage water treatment. However, there are not many works in the literature in CFD modelling of sedimentation tanks for potable water treatment. Moreover, according to the authors best knowledge, the interaction between primary and secondary phase (between water and particles) in a sedimentation tank, and the way that this momentum exchange can influence the water velocity and hence the process efficiency, has never been investigated previously.

The main aim of the present paper is to investigate the 3D hydrodynamics and flow behaviour of a sedimentation tank taking into account the momentum exchange between the primary (water) and the secondary phase (solid particles), using a Lagrangian multiphase CFD method (discrete phase model) with two-way couple calculations that is offered by the commercial CFD code ANSYS FLUENT. An experimental installation of a sedimentation tank that was used in experiments conducted by the Laboratory of Hydraulic Construction of the Swiss Federal Institute of Technology, is adopted for the numerical simulations of the present paper.

The structure of the present paper is the following: In the present section a brief introduction on sedimentation tanks and a brief literature review in previous works have just been conducted. Section 2 presents the numerical model that is used in the simulations of the present investigation, which is based on a Lagrangian approach where the fluid phase is treated as a continuum by solving the Navier–Stokes equations, while the dispersed phase is solved by tracking a large number of particles that exchange momentum with the fluid phase. For turbulence closure the Renormalization-group (RNG) k–ε model is applied, which is an enhanced version of the widely used standard k–ε model. Also it is described the physical problem, the materials and methods that were used in simulations and the validation of the proposed numerical model comparing the numerical results with the corresponding experimental data of Kantoush et al. [21] and Dewals et al. [22]. Section 3 and Section 4 presents the results and reports the main conclusions drawn from the present study, respectively.

2. Material and methods

2.1. Mathematical model

2.1.1. General information

The hydrodynamic characteristics of a sedimentation tank can be studied as a multiphase flow using either an Euler–Euler or an Euler–Lagrange approach. In the literature, Eulerian applications are used for almost all diffusion dominated problems, but without calculating individually the particle trajectories along the flow field. Due to their versatile capabilities, approaches based on the Lagrangian method have been applied extensively for many two-phase flow problems. The Lagrangian approach provides a more detailed and realistic modelling of particle deposition because the equation that describes the particle motion is solved for each particle moving through the field of random fluid eddies. In these approaches, the fluid is treated as a continuum and the discrete (particle) phase is treated in a natural Lagrangian manner, which may or may not have any coupling effect with the carrying fluid momentum (in the proposed model with coupling effect). A fundamental assumption made in this model is that the dispersed secondary phase occupies a low volume fraction, even though high mass loading is acceptable. De Clercq et al. [23] mentioned that the Lagrangian model should not be applied whenever the particle volume fraction exceeds 10–12%. The particle trajectories are computed individually at specified intervals during the fluid phase calculation.

In the case of turbulent flows, the conservation equations are solved to obtain time-averaged information. Since the time-averaged equations contain additional terms, which represent the transport of mass and momentum by turbulence, turbulence models that are based on a combination of empiricism and theoretical considerations are introduced to calculate these quantities from details of the mean flow.

In the simulations of the current work the RNG k–ε model is applied for turbulence closure. This model was derived using a rigorous statistical technique, the renormalization group theory. The basic form of the RNG k–ε model is similar to the