Powder Technology 268 (2014) 139–149

Contents lists available at ScienceDirect

Powder Technology

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

The influence of lamellar settler in sedimentation tanks for potable water treatment — A computational fluid dynamic study

Roza Tarpagkou *, Asterios Pantokratoras

Democritus University of Thrace, Department of Civil Engineering, V. Sofias 12, GR-67100 Xanthi, Greece

ARTICLE INFO

Article history:
Received 6 May 2014
Received in revised form 6 August 2014
Accepted 14 August 2014
Available online 23 August 2014

Keywords:
Computational fluid dynamics
Sedimentation
Lamella settling tanks
Multiphase flow
Particle fluid interaction

ABSTRACT

Lamella gravity settlers are often used in water treatment to make sedimentation tanks more cost-effective. Effective surface area for settlement is increased by the inclined plates giving such systems a smaller footprint than the conventional tanks. In this paper a numerical model was used to simulate the dynamics and flow structure of a rectangular sedimentation tank for potable water through a multiphase approach, using computational fluid dynamic (CFD) methods. Two configurations have been examined, one with a system of inclined parallel plates (lamellar settlers) and another with a conventional design, in order to evaluate the influence of lamellar settlers in the process efficiency. Unlike most of the previous numerical investigations which studied the lamellar settlers separately, the present numerical approach studies the whole sedimentation tank with a full scale system of inclined parallel plates. The momentum exchange between the primary and the secondary phase (particles) is taken into account, using a Lagrangian method (discrete phase model) with two-way coupled calculations. Contours of stream function, velocity and concentration are presented, as well as the velocity and concentration profiles for the inclined plates. The results show that the lamellar settlers influence the flow field and increase the sedimentation efficiency by 20% in comparison with the convectional design.

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 settle-able solids, by the mechanism of gravitational settling. Lamella gravity settlers are often used in water treatment to make rectangular tanks more cost-effective. As in conventional clarifiers, water flows from the bottom of the clarifier and out through the top while particulate settles by gravity to the bottom of the clarifier. The difference between conventional clarifiers and enhanced gravity separators is that the flow in the settling zone is directed through inclined parallel plates or tubes. This virtually eliminates unstable flow patterns and mixing currents, which greatly inhibit the settling of solids. In addition to enhancing laminar flow conditions, the inclined surfaces of the plates reduce the distance that particles need to travel before settling. The particles begin to agglomerate as soon as they hit the surface of the plate, as shown in Fig. 1. The newly created agglomerated particles settle much more rapidly than the fine particles and slide along the settling surface, moving toward the bottom of the clarifier for collection and discharge. Moreover, the efficiency of discrete particle settling in horizontal liquid flow, depends on the area available for settling. The effective surface area for particle settling in enhanced gravity separators is increased by the inclined plates, giving such systems a smaller footprint than the conventional tanks.

Inclined parallel plates are a classical subject with a long history, and Boycott [1] was the first who observed that the settling rate of suspension is better "if the tube is inclined than when it is vertical". This is also known as the Boycott effect. The settling behavior in inclined vessel was modeled firstly by Ponder [2] and latter by Nakamura and Kuroda [3], and for this reason is also known as the PNK theory. According to this theory, the quality of the clarified fluid is a function of the vertical settling velocity of particles and the horizontal projection area of the settler.

In general many researchers have been focused on the study of sedimentation tanks for wastewater treatment. Larsen [4] was probably the first who applied a CFD model to several secondary clarifiers. McCorquodale and Zhou [5] investigated the effect of various solids and hydraulic loads on circular clarifier performance. Patziger et al. [6] investigated the effects of sludge return and the inlet geometry on settling tanks performance.

In a primary sedimentation tank, where the discrete settling prevails, Imam et al. [7] applied a fixed settling velocity and used an averaged particle velocity. Liu et al. [8] conducted measurements and simulations to optimize design of a settling tank.

For potable water treatment Goula et al. [9] modeled a sedimentation tank for potable water and examined the influence of a feed flow control baffle. Tarpagkou and Pantokratoras [10] simulated and examined the effect of particles on fluid phase due to momentum exchange and how this

* 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.powtec.2014.08.030
0032-5910/

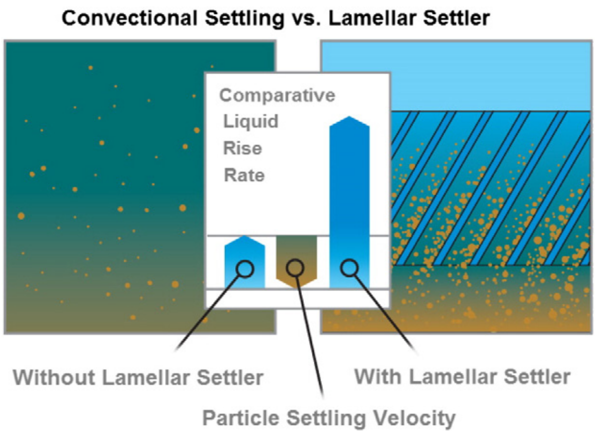

Fig. 1. Particle behavior in conventional settling in comparison with enhanced gravity separators (http://www.brentwoodindustries.com).

affects the fluid velocity. Wang et al. [11] simulated the flow field and SS concentration in a rectangular sedimentation tank.

There are very few theoretical studies about lamellar settlers. Demir [12] used experimental results obtained under different surface loadings and plate angles to evaluate settling tanks. Lekang et al. [13] presented an evaluation of lamellar settling tank using experimental study for measuring inlet and outlet water values of the sedimentation basin. Kowalski and Mięso [14] presented the design of sedimentation tanks and installations utilizing the Boycott's effect. Doroodchi et al. [15] investigated the influence of inclined plates on the expansion behavior of solids suspensions in liquid fluidized beds. Sarkar et al. [16] studied the performance of the inclined plate settlers for aquaculture waste. Rubescu et al. [17] presented a theoretical study concerning the design of lamella secondary settling tank. Galvin et al. [18] examined gravity separation using closely spaced inclined channels. Callen et al. [19] study the particle elutriation from a fluidized bed incorporating parallel inclined plates. Laskovski et al. [20] examine the tendency for particle re-suspension in inclined channels. Lavine [21] examine the mixed convection between inclined parallel plates with a uniform heat flux boundary condition. Okoth et al. [22] summarized the factors which have effect on the separation efficiency of inclined parallel plates, and they modeled the suspension-sediment interaction phenomenologically.

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 modeling of sedimentation tanks for potable water treatment. Moreover, according to the authors' best knowledge, the study of the whole sedimentation tank with a full scale system of inclined parallel settlers has never been investigated previously. Unlike most of the previous CFD investigations which studied the lamellar plates separate and alone, the authors believe that this is the first work in the literature which modeled the whole system of sedimentation tank with lamellar settlers for potable water.

The main aim of the present paper is to investigate the 2D hydrodynamics and flow behavior of a rectangular sedimentation tank with two configurations: one with a system of inclined parallel plates (lamellar settlers) and another with a conventional design, in order to evaluate the influence of lamellar settlers in the process efficiency. This was achieved using the computational fluid dynamic (CFD) methods offered by the commercial software ANSYS CFD. The present work fully considers the interaction between the liquid and solid phase (and vice-versa).

The structure of the present paper is the following: in the present section a brief introduction on sedimentation tanks and a brief literature review of previous works have just been conducted. Section 2 presents the mathematical model that is used in the simulations of the present investigation. Section 3 presents the physical problem and the materials and methods that were used in the simulations. It also describes the validation of the proposed numerical model comparing the numerical results with the corresponding experimental data. Section 4 and Section 5 present the results and report the main conclusions drawn from the present study, respectively.

2. Mathematical model

2.1. 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 modeling 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 such an approach, 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/two-way coupled calculations). In this paper we used the Euler–Lagrange multiphase approach with two-way coupled calculations which takes into account the interaction between the particles and the fluid (and vice-versa), and calculated the rate of momentum exchanged and transferred from the water to particles (and vice-versa). The main advantage of this approach is that 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 with the discrete phase, in this model we incorporated the effect of the discrete phase trajectories on the continuum. This interchange affects fluid velocity, especially in the case of large particle sizes, which have a greater relaxation time in relation to the characteristic time of the tank [10].

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 standard k–ε model, but it includes a number of refinements, rendering it more appropriate for the numerical simulations of the present investigation, as it is more accurate for swirling flows and rapidly strained flows and also accounts for low Reynolds number effects. Moreover, it provides an analytical formula for the calculation of the turbulent Prandtl numbers. At this point it should be mentioned that the RNG k–ε model is also modified accordingly in order to simultaneously account for the primary (continuous) phase and the secondary (dispersed) phase of the simulated flows.

2.2. Governing equations

2.2.1. Fluid phase

Fluid phase is treated as a continuum by solving the Navier–Stokes equations so the equations of conservation of mass (Eq. (1)) and