Chemical Engineering Science 158 (2017) 21–29

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Numerical simulation of separation process for enhancing fine particle removal in tertiary sedimentation tank mounting adjustable baffle

Hong Guo [a], Seo Jin Ki [a], Seungjae Oh [b], Young Mo Kim [a], Semyung Wang [b], Joon Ha Kim [a],*

[a] *School of Environmental Science and Engineering, Gwangju Institute of Science and Technology (GIST), Gwangju 500-712, Republic of Korea*
[b] *School of Mechatronics, GIST, Gwangju 500-712, Republic of Korea*

HIGHLIGHTS

- The hydraulic behavior of a sedimentation tank was studied using the finite element method.
- The sedimentation tank was equipped with adjustable baffles to better analyze particle settling.
- The simulation results matched well with the experimental data.
- The particle removal efficiency varied considerably by varying the angles of the adjustable baffle.
- The results can be used to design a compact sedimentation tank for aquaculture farming.

ARTICLE INFO

Article history:
Received 3 June 2016
Received in revised form
18 August 2016
Accepted 16 September 2016
Available online 19 September 2016

Keywords:
Separation process
Finite element method
Fine particle removal
Adjustable baffle
Sedimentation tank
Hydraulic behavior

ABSTRACT

The presence of flow control devices and/or obstacles in the fluid flow such as baffles and spacers complicates the solid-fluid mixing pattern, which remains difficult to describe by classical analytical solutions. In this study, the removal of fine particles in a tertiary sedimentation tank mounting an adjustable baffle was investigated using the computational fluid dynamics code-COMSOL. The solid-fluid motion was solved by consecutively applying the equations of the continuity and momentum using the finite element method. The experiment was conducted by the sedimentation tank with the adjustable baffle inclined at 30° in a pilot scale plant. It's used as the reference data set for numerical simulations that were run on a 2-dimensional domain by modifying the configuration settings of angles for an adjustable baffle (i.e., 30°, 45°, and 60°) and without one. Results showed that the simulation results matched well with the experimental data for an adjustable baffle at 30° (NSE = 0.97). The sedimentation tank with the adjustable baffle at different angles had a lower overflow rate (in the area of flow rebound) and mixing intensity (in the area of flow curve) than without one, eventually leading to enhanced particle removal efficiency. This tendency became more pronounced as the particle motion stabilized over time. The sedimentation tank mounting the adjustable baffle at 30° provided the best settling efficiency among the four different flow patterns. However, the conventional index that represents the mixing properties did not correctly address their relative efficiency for fine particle removal. Therefore, a numerical simulation tailored to a given geometry should be conducted to fully elucidate the fluid dynamics in the sedimentation tank with complex devices or obstacles.

1. Introduction

Fluid dynamics in water treatment systems plays an important role in facilitating the particle removal process and ensuring effective system design in affordable budget (Brouckaert and Buckley, 1999). A sedimentation tank, a popular unit in advanced water treatment processes, is traditionally used to remove suspended solids from turbid water by settling via gravity. The performance of the sedimentation tank is affected by two universal factors, flow patterns and particle properties (Goula et al., 2008b). More precisely, the flow motion is determined by the flow rate or velocity (that determines the turbulence), temperature, and mixing characteristics. The intrinsic particle properties include their size, density, shape, and charge, which affect the ways they interact with water through drag and buoyancy forces. In addition, the geometry of the sedimentation tank is tightly bound to these two dominant factors, which further complicates the description of the hydraulic behavior in the sedimentation tank, specifically in the

---

* Corresponding author.
E-mail address: joonkim@gist.ac.kr (J.H. Kim).

http://dx.doi.org/10.1016/j.ces.2016.09.022
0009-2509/

Nomenclature

c_d	particle drag coefficient (dimensionless)
d	diameter (m)
F	volume force (N/m^3)
g	gravity vector (m/s^2)
m	mass (kg)
n	normal vector
p	pressure (Pa)
u'	turbulent fluctuation (m/s)
u	velocity vector (m/s)
U	mean velocity of fluid flow (m/s)
u_{slip}	relative velocity vector between two phases (m/s)
ζ	random number normally distributed with zero mean and unit standard deviation
μ	viscosity (Pa s)
ρ	density (kg/m^3)
τ_{Gm}	sum of viscous and turbulent stresses (kg/m s^2)
τ_p	particle velocity response time (s)
ϕ	volume fraction (dimensionless)
D_{md}	turbulent dispersion coefficient (m^2/s)
m_{dc}	mass transfer rate from dispersed to the continuous phase (kg/(m^3 s)
Re_p	particle Reynolds number (dimensionless)
k	turbulent kinetic energy (m^2/s^3)
ε	dissipation rate of kinetic energy (m^2/s^3)
P_k	production term of turbulence (dimensionless)
F	additional volume force (N/m^3)

Subscripts

c	continuous phase
d	dispersed phase
f	fluid (of water)
p	particle

presence of fluid obstacles or assisted devices (Patziger et al., 2012). Therefore, a physically-based model that can accurately explain these factors is required to effectively design the sedimentation tank in compact size while enhancing the settling of suspended solids.

A mathematical modeling study expedites the design of the sedimentation tank by avoiding trial and error experiments in the lab or via a pilot system. Larsen (1977) developed the original mathematical model for simulating the movement of water only in a rectangular settling tank, and investigated 2-dimensional velocity profiles (i.e., without particles) in a fluid field. Lyn et al. (1992) expanded this original model by including a gravity-density term in particle transport and accounted for the settling of particles of different sizes with respect to the flow velocity. The expanded model, however, did not address the interactions between the liquid and solid phases occurred in a solid-fluid field. Subsequently, Manninen et al. (1996) improved the expanded model using a single governing equation that combined the continuous (for fluids) and dispersed phases (for particles) as two interpenetrating continua, simultaneously solving the continuity and momentum equations of the mixture. This model has since been widely used as a parent for studying the motion of particles or gas bubbles in fluid fields such as membrane bioreactors, gas dispersion reactors, and settling tanks.

Such mathematical models are, however, difficult to analytically solve as geometric complexity of the system (that affects the hydraulic behaviors) is increased progressively. DeVantier and Larock (1986, 1987) applied the finite element method (FEM) to the sedimentation tank to simulate a 2-dimensional turbulent flow in terms of density effects, obtaining a fairly good prediction accuracy while consuming large memory resources. On the other hand, reasonable prediction accuracy was achieved in secondary clarifiers using the finite volume method (FVM), which used less memory than the FEM during the simulation (Szalai et al.,1994). Also McCorquodale et al. (1991), combined the FEM and finite different method (FDM) to examine the effects of particle size on the flow motion in primary clarifiers, but did not find any remarkable advantage over the FEM alone. Out of these methods, the FVM has been preferred in the area of fluid dynamics due to less memory requirements and acceptable accuracy. Combining the FVM with the Euler-Lagrange approach has recently enabled researchers to successfully describe the hydraulic behaviors of flow and particles in the sedimentation tank under the influence of turbulent flow, sludge return, and water temperature (Al-Sammarraee et al., 2009; Goula et al., 2008a; Karama et al., 1999; McCorquodale et al., 1991).

In fact, the transport of the particles, including their interaction with the surrounding fluid, can be numerically elucidated using either the Euler-Euler or Euler-Lagrange approaches (Goula et al., 2008a; Manninen et al., 1996; Tarpagkou and Pantokratoras, 2013; Wang et al., 2013). Two methods were similar in that the fluid was considered as a continuum, but were dissimilar in the particle phase simulation. For instance, the Euler-Euler approach assumed the particles as a piece of a continuous medium (i.e., one of volume fractions), whereas the particle phase was treated separately from the fluid phase (as the Lagrangian discrete field) in the Euler-Lagrange approach (Adamczyk et al., 2014). In a practical aspect, the Euler-Euler method was more easily applied to multiphase flow systems at the field scale than the Euler-Lagrange method due to convenience in experimental methods measuring the parameters (required in the simulation) such as the particle density and diameter. Another advantage of the Euler-Euler approach was that with low computational demand, the particle size, volume fraction, and relative velocity distribution were estimated directly at the end of the simulation. In contrast, the Euler-Lagrange approach provided accurate information on individual particles such as the particle pathway and particle dispersion patterns, allowing the simulation of agglomeration or deformation of the particles in the fluid systems (Zhu et al., 2015). One notable disadvantage was, however, that it required large computational resources for the particles at high concentration, and thus, was rarely used for the simulation of the dense fluidized beds. Also, the absence of a module regarding the collision between the particles, which was normally adopted in the Euler-Euler approach, was another minor drawback in the Euler-Lagrange approach (Loha et al., 2014).

As compared to previous studies, this study explores the effect of geometric complexity on the settling of fine particles in the compact sedimentation unit using the FEM with the Euler-Euler approach This is because the FEM, which can be now implemented affordably due to advances in computational resources, provides a higher prediction accuracy than the FVM for the system with complex geometry (Gohil et al., 2011). For this study, a pilot scale experiment in the sedimentation tank equipped with an adjustable baffle installed at an inlet channel of influent was conducted to investigate the dynamic flow patterns and particle motion, which were then used as a basis for numerical analysis. Using the computation fluid dynamics package COMSOL, this study specifically aims: 1) to identify how the adjustable baffle