Applied Mathematical Modelling 35 (2011) 3012–3020

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

# Numerical modeling on inlet aperture effects on flow pattern in primary settling tanks

Fatemeh Rostami, Mahdi Shahrokhi, Md Azlin Md Said *, Rozi Abdullah, Syafalni

*School of Civil Engineering, Universiti Sains Malaysia, 14300 Nibong Tebal, Seberang Perai Selatan, P. Penang, Malaysia*

ARTICLE INFO

*Article history:*
Received 25 April 2010
Received in revised form 2 December 2010
Accepted 9 December 2010
Available online 21 December 2010

*Keywords:*
Settling tanks
Inlet structure
Kinetic energy
Numerical simulations

ABSTRACT

Inlets should be designed to dissipate the kinetic energy or velocity head of the mixed liquor and to prevent short-circuiting, mitigate the effects of density currents, and minimize blanket disturbances. Flow in primary settling tank is simulated by means of computational fluid dynamics. The fluid is assumed incompressible and non-buoyant. A two-dimensional computational and one phase fluid dynamics model was built to simulate the flow properties in the settling tank including the velocity profiles, the flow separation area and kinetic energy. In this study, the RNG turbulent model was solved with the Navier–Stokes equations. In order to evaluate hydraulic influences on the velocity profile, separation length and kinetic energy, three different of opening positions and two and three aperture in inlets were simulated. The flow model uses to apply a fixed-grid of cells that are all rectangular faces; the fluid moves through the grid and free surfaces are tracked with the volume-of-fluid (VOF) technique. Effects of numbers and locations of inlet apertures on the flow field are presented and the results show the positions of inlet apertures are affected on the flow pattern in the settling basin and increasing the numbers of slots can reduce kinetic energy in the inlet zone and produce uniform flow.



## 1. Introduction

The process of removing suspended particles from water by gravity is known as sedimentation. This method, which has been used for over 1000 years, is an integral part of any water and wastewater treatment plant. Settling tanks are among the main parts of a treatment plant, especially those involved in the purification of turbid flows. In these tanks, turbid water flows through the length of the tanks at a low speed, giving enough time for suspended particles to settle. As such, finding new and useful methods to increase hydraulic efficiency is the objective of many theoretical, experimental, and numerical studies.

Settling tanks can be rectangular with horizontal flow or circular with radial flow patterns. Energy dissipation is the main objective of designing a primary clarifier inlet. In rectangular tanks, the influent enters the basin at the inlet, the energy of which must be dissipated at the inlet zone by selecting the best position and configuring the inlet or by using the baffles in the inlet zone [1].

Because the rate and extent of the flocculation reaction is dependent on the concentration of the particles to be flocculated, among others, the design engineer must ensure optimum conditions for flocculation at the inlet where the concentration of solids is highest. While much has been done to improve inlet designs in secondary clarifiers to promote flocculation, there has been little improvement on primary clarifiers [1].

* Corresponding author. Tel.: +60 45995834x6202; fax: +60 45941009.
*E-mail addresses:* fa.rostami@gmail.com (F. Rostami), mhd.shahrokhi@gmail.com (M. Shahrokhi), azlin@eng.usm.my (M.A. Md Said), cesrozi@eng.usm.my (R. Abdullah), cesyafalni@eng.usm.my (Syafalni).

0307-904X/$ - see front matter 
doi:10.1016/j.apm.2010.12.007

Enlarging the size of the inlet zone and using inlet energy for flocculation can improve the removal of suspended solids. Impinging flow streams against one another is an effective way of promoting flocculation [2]. Imam et al. [3], simulated primary settling tanks and found a relation between inlet wall submergence and inlet eddy length. Larger eddies comprise smaller inlet apertures, thus decreasing removal efficiency.

Density effects and potential energy may decrease due to the low position of the inlet, as proposed by some studies [1]. Bretscher et al. [4] confirmed these theoretical considerations through prototype measurements in rectangular tanks. In their study, the inlet aperture was not quantified and flocculation or energy dissipation was not taken into account. Tamayol et al. [5] suggested that the best position for the inlet is somewhere in the middle depth of the tank, and inlets at the bottom of the tank are believed to be better than inlets at its surface. Flocculation and avoiding floc breakup were the main objectives of Larsen's [6] inlet design, which was developed in pure-water model tests by evaluating velocity profiles in place of flow through curves (FTCs).

Goula et al. [7] studied the effect of inlet baffle heights on flow patterns and particle trajectories throughout or at the outlet of tanks and found that the baffle affects the inlet section and the area near the bottom of the tank. It seems that the extended baffle provides better influent mixing and isolation between the tank influent and effluent than the short baffle in the original tank design, thus significantly enhancing sedimentation. The extended baffle increases the kinetic energy and the dissipation rate in the inlet baffle in the region and, consequently, weakens the current in the area.

## 2. Governing equation

### 2.1. Time averaged flow equations

The governing equations that determine flow are the general mass continuity and momentum expressions. The turbulence model is also used to calculate the Reynolds stresses. The mass continuity equation for fluid is simple: as the flow pattern is assumed to be two dimensional (2D), two momentum equations in the $x$ and $z$ directions respectively represent the length and height of the tank to be solved. The general mass continuity equation is as follows [8,9]:

$$V_f \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + \frac{\partial}{\partial z}(\rho w A_z) = 0, \tag{1}$$

where $V_f$ is the fractional volume of the flow in the calculation cell and $\rho$ is the fluid density. The variables $(u, w)$ are the velocity components in the length and height $(x, z)$ directions. The momentum equations for the fluid velocity components in the $(x, z)$ directions are described by the Navier–Stokes equations [10]:

$$\begin{aligned} \frac{\partial u}{\partial t} + \frac{1}{V_f}\left\{uA_x\frac{\partial u}{\partial x} + wA_z\frac{\partial u}{\partial z}\right\} &= -\frac{1}{\rho}\frac{\partial P}{\partial x} + G_x + f_x \\ \frac{\partial w}{\partial t} + \frac{1}{V_f}\left\{uA_x\frac{\partial w}{\partial x} + wA_z\frac{\partial w}{\partial z}\right\} &= -\frac{1}{\rho}\frac{\partial P}{\partial z} + G_z + f_z. \end{aligned} \tag{2}$$

where $G_x, G_z$ are the body accelerations and $f_x, f_z$ are the viscous accelerations for a variable dynamic viscosity $\mu$:

$$\rho V_f f_x = wsx - \left\{\frac{\partial}{\partial x}(A_x\tau_{xx}) + \frac{\partial}{\partial z}(A_z\tau_{xz})\right\}, \quad \rho V_f f_z = wsz - \left\{\frac{\partial}{\partial x}(A_x\tau_{xz}) + \frac{\partial}{\partial z}(A_z\tau_{zz})\right\}$$

where:

$$\tau_{xx} = -2\mu\frac{\partial u}{\partial x}, \quad \tau_{zz} = -2\mu\frac{\partial w}{\partial z}, \quad \tau_{xz} = -\mu\left\{\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right\}$$

In the above expressions, the terms *wsx* and *wsz* are wall shear stresses. If these terms are omitted, there is no wall shear stress because the remaining terms contain the fractional flow areas $(A_x, A_z)$ which vanish at walls. The wall stresses are modeled by assuming a zero tangential velocity on the portion of any area closed to flow. Mesh boundaries are an exception because they can be assigned non-zero tangential velocities. For turbulent flows, a law-of-the-wall velocity profile is assumed near the wall, which modifies the wall shear stress magnitude [10].

When a free surface flows simulated in Flow-3D, the volume of fluid (VOF) method is active automatically to calculate the free surface configuration in surface cells. Therefore, it is impossible to consider the free surface as a rigid boundary. In VOF method, grid cells are classified as empty, full, or partially filled with fluid. Cells are allocated with fluid fractions varying from zero to one, depending on the quantity of fluid. Thus, fluid exists in $F = 1$, and $F = 0$ corresponds to void regions. This function illustrates the VOF per unit volume and satisfies the following equation [8]:

$$\frac{\partial F}{\partial t} + \frac{1}{V_F}\left\{\frac{\partial}{\partial x}(FA_x u) + \frac{\partial}{\partial z}(FA_z w)\right\} = 0. \tag{3}$$

The free surface slope of a partially filled cell is calculated by the free surface angle and the location of the neighborhood cells. It is determined by a series of connected lines in 2D or by connected surfaces in 3D simulations. These fractions are derived into all terms of the Reynolds-averaged Navier–Stokes (RANS) equations.