INVESTIGATION OF THE HYDRAULIC CHARACTERISTICS OF THE RECTANGULAR SEDIMENTATION TANK BY USING SST k- ω TURBULENCE MODEL

Hasim A. M. Heikal
Prof. Dr. in Mech. Power Depart., Faculty of Engineering (Mattaria), Helwan University

Aida A. El-Hafiz
Prof. Dr. in Mech. Power Depart., Faculty of Engineering (Mattaria), Helwan University

Ahmed R. El Baz
Prof. Dr. Mech. Power Depart., Faculty of Engineering, Ain Shames University

Sherif M. Farghaly
Ph.D. in Mech. Power Eng., Faculty of Engineering (Mattaria), Helwan University

ABSTRACT

For the proper design of sedimentation tanks, the detailed study of the hydraulic characteristics of the sedimentation tanks should be performed. Then, Steady state incompressible flow conditions with viscous effect are generally considered in hydraulic numerical modelling. Besides, the SST k- ω turbulence model was used in the numerical calculations. In order to validate the mentioned method, a comparison was performed between the numerical results and the previous experimental data which carried out by other authors.

Key Words: Rectangular Sedimentation Tank, Computational Fluid Dynamics, Settling Tank, Numerical Modelling and Turbulence Model.

1. INTRODUCTION

Solids removal is probably the main water purification method in water treatment plants. The most significant phase of this process is to separate sludge and suspended particles from water by means of gravity. In these basins, the turbid water flows into the basin at one end and the cleaner water is taken out at the other end by decanting. Obviously, the water must flow in the tank long enough for the appropriate particle deposition. Sedimentation by gravity is a usual and important process in settling tanks to remove inorganic settleable solids from water and waste water in refinery plants [Swamee and Tyagi 1990].

Numerous studies show that in order to remove suspended solids with minimum cost; they should be removed as quickly and efficiently as possible from the water. In fact, if the removed solid concentrations from settling tanks are increased in order to increase treatment efficiency, the size of water treatment facilities, which are located downstream of the clarifiers, can be reduced (Cripps et al, 2000). According to the investigations of Camp (1946) and Swamee and Tyagi (1990), the investment costs of settling facilities contribute to a large portion (typically one-fourth to one-third) of the total cost of treatment plant construction. For that reason, significant savings in both capital and operational costs at various stages of treatment can be expected by increasing solid removal efficiency (Cripps et al, 2000). As a result, increasing removal efficiency is important.

Many factors can influence removal efficiency, including tank hydraulics, which are of great significance (De Clercq et al, 2003). It is noticeable that the ability of a sedimentation tank to remove suspended solids depends on its flow field. Therefore, investigating the structure of the flow field is of great importance (Campbell and Empie, 2006).

Settling tanks are divided into two main categories: primary and secondary (final) sedimentation tanks. It is worth making a distinction between these types of tank. Primary sedimentation tanks are designed to reduce the particulate flow velocity and provide for the settling of organic solids (AWWA, 1990). The sludge in these tanks is not activated [AWWA, 1990], hence particle concentration is low and there is not a large difference between particle sizes. As a result, the flow is not much influenced by concentration distribution (Tamayol et al, 2010). But in a final sedimentation tank, the particle concentration in settled sludge is relatively high resulting in significant density effects. Also, a wide range of particles with various sizes can be found. In a prototype test, Anderson compared the flow fields in primary and final settling tanks of similar geometries and hydraulic loadings. In the primary clarifiers, flow was observed to be along the surface from inlet to outlet (Anderson, 1945). On the contrary, sludge concentration in the secondary-clarifier inlet resulted in a density current along the bottom, causing reverse flow at the surface. Furthermore, in a secondary clarifier, particles flocculate, which produces larger particles, causing an increase in particle settling velocity (Overcamp, 2006).

The sedimentation performance depends on the characteristics of the suspended solid and flow field in the tank. Given that there is low concentration in the primary settling tanks, flow-field is not influenced by particles; further, the flow pattern and the track taken by suspended solid through the tank are closely linked to each other and the settling tank efficiency. The flow field in the sedimentation tanks is turbulent, and such turbulence affects particle concentration and deposition; thus, if the turbulence is not predicted correctly, it may cause re-suspension of particles that have already settled. Recent numerical models have shown fractional success in predicting the velocity field and the concentration distribution of suspended solids in sedimentation tanks. On the other hand, several researchers have used the two-equation $k - \varepsilon$ turbulence model . Kahane et al. [2002] mentioned that the transport of suspended particles from the inlet to various points in the settling zone is governed by the hydrodynamics and turbulent of the flow.

Lowe (1990), Studied the behaviour of sedimentation basins by using both numerical and experimental model simulations. Two specific effects to be considered were the effect of wind action, and the effect of density stratification. The results indicated that, as wind speed increased, sediment removal efficiency decreased, with the counter-current case slightly worse than the co-current case. The overall comparison of the numerical and experimental values was good, indicating that the numerical model can give reasonable estimates of basin performance even with the presence of wind and/or stratification.

Van der Walt (2002) reviewed the several investigations carried out by using CFD techniques for the rectangular sedimentation tanks and can be summarised in table 1.0.

Table (1) Secondary clarifier and sedimentation tank CFD models - Van der Walt (2002).

Sr. No.	Author	Tank Type SC-Secondary clarifier, ST -Sedimentation tank	Solids transport model PTP-Pseudo Two Phase, TPU-Uncoupled Two Phase, EL-Euler-Lagrange	Turbulence model	Discretisation method FD - Finite Difference, FE-Finite Element, FV-Finite Volume
1	Larsen (1977)	SC	PTP	$\mu t = k$	FD
2	Shamber and Larock (1981)	SC	No	k-ε model	FV
3	Imam and McCorquodale (1983)	SC	No	$\mu t = k$	FD, grid refinements
4	Shamber and Larock (1983)	SC	TPU	k-ε model	FV
5	Larock, Chun and Shamber (1983)	SC	TPU	k-ε model	FV
6	Imam, McCorquodale and Bewtra (1983)	SC	TPU	$\mu t = k$	FD
7	Celik, Rodi and Stamou (1985)	SC	TPU (Dye)	k-ε model	
8	DeVantier and Larock (1986)	SC	PTP	k-ε model	FV
9	DeVantier and Larock (1986)	SC	PTP	k-ε model	FV
10	Stamou, Adams and Rodi (1989)	SC	TPU	k-ε model	FV
11	Adams and Rodi (1990)	SC	TPU	k-ε model	FV
12	Casonato and Gallerano (1990)	SC	No	k-ε model	FD, self adaptive
13	McCorquodale et al. (1991)	SC	PTP	k-ε model	FV
14	Krebs (1991)	SC	PTP	$\mu t = k$	FV, Phoenics
15	Stamou (1991)	SC	TPU	k-ε model	FV, TEACH
16	Samstag et al. (1992)	SC	PTP	k-ε model	FV
17	Lyn, Stamou and Rodi (1992)	SC	PTP	k-ε model	FV
18	Zhou et al. (1992)	SC	PTP	k-ε model	FV
19	Zhou and McCorquodale (1992)	SC	PTP	k-ε model	FV
20	Samstag, McCorquodale and Zhou (1992)	SC	PTP	k-ε model	FV
21	McCorquodale and Zhou (1993)	SC	PTP	k-ε model	FV
22	Frey et al. (1993)	ST	Euler-Lagrange, TPU	k-ε model	FV
23	Olsen and Skoglund (1994)	Sand Trap	TPU	k-ε model	FV
24	Dahl, Larsen and Petersen (1994)	SC	TPU	k-ε model	FV, Phoenics
25	Zhou, McCorquodale and Godo (1994)	SC	TPU (Temperature)	k-ε (algebraic) $\mu t = 100 \mu$	FV
26	Krebs, Vischer and Gujer (1995)	SC	TPU		FV, Phoenics
27	Van der Walt (1994, 1996))	SC	PTP	k-ε model	FV, Flo++
28	Ekama et al. (1997)	SC	Various approaches		
29	Deiniger, Holzhausen and Wilderer (1998)	SC	TPU	k-ε model	FE
30	Van der Walt (1998b)	ST	PTP	k-ε model	FV, Flo++
31	Brouckaert et al. (1998)	SC	Single phase	Not reported	FV, Phoenics
32	Marais Ekama & De Haas (2000)	SC	PTP	k-ε model	FV
33	van der Walt (2000c)	ST	PTP	k-ε model	FV, Flo++
34	van der Walt (2000d).	ST	PTP	k-ε model	FV, Flo++

McCorquodale et al (2006) studied the hydrodynamics of different configuration under similar hydraulic loading and differ in fundamental aspects that determine what types of geometric modifications are likely to improve the clarifier performance. The best combination was an inlet skirt and an extended inboard launder with a perforated baffle.

Krish et all (2007), assessed the current performance and evaluate alternatives to further improve the efficiency of the FSTs at the higher loads expected in New York wastewater treatment plant. As a result, a three-dimensional CFD model has been developed that is being used to consider different inlet and baffle arrangements, altering the method of sludge withdrawal and considering loading alternatives to the FSTs.

Razmi et al (2009) stated that the circulation regions always exist in settling tanks. These regions would result in short-circuiting enlargement of the dead zone and high flow mixing problems and avoid optimal particle sedimentation.

Tamayol et al (2008) studied the performance of settling tanks using an Eulerian-Lagrangian method. The inlet position would also affect the size and location of the recirculation region. Using a proper baffle configuration could substantially increase the performance of the settling tanks. The best position for the inlet was near the bottom (at middepth of the tank) and existence of a reflection entrance baffle near the free surface and near the circulation zone of settling tanks can increase the performance of primary settling tanks.

Jamshidnia et al (2010), studied experimentally the effect of baffle on the flow field of the primary sedimentation tank. The provision of a baffle as a geometrical modification of a tank may influence the flow field for better sedimentation. It was quantitatively found that the intermediate baffle not only influences the flow field in its downstream, but also affects the flow pattern in its upstream. It was found that the baffle setting and its position relative to the inlet and outlet influences the flow field and the development of flow. Baffled flow may provide better conditions for sedimentation by influencing velocity profiles. Positioning of a baffle in the middle of the channel ($x_b/L = 0.5$) may improve the flow field at its downstream by modifying the velocity gradient near the channel bed.

Firoozabadi et al (2010) predicated the hydraulic efficiency of the primary rectangular settling tanks. The non-linear k-ε turbulence model is used for predicting the length of the reattachment point in the separated flow of a Karlsruhe tank (university of Karlsruhe, Germany). Then, the recirculation bubble size, which is out of the capability of standard turbulence models, is determined. Also, the effect of the separation zone size on the tank's hydraulic efficiency is investigated.

Tamayol et al (2010) Stated that the flow in settling tanks is stratified, but the effect of buoyancy force on the flow field depends on the inlet concentration of particles and flow bulk velocity. The results show that neither Reynolds nor Froude numbers are sufficient to be considered alone. Results show that in high Reynolds numbers, the flow field and baffle position are not affected by the inlet Froude number.

Naser et al (2010) studied the hydrodynamics of a rectangular sedimentation basin under turbulent conditions by creating a steady, two-dimensional numerical model. a two-equation turbulence model i.e., k-ε model was used. The model successfully predicted the velocity distribution, the concentration of the suspended load, and both the kinetic energy and its dissipation rate.

It is cleared from the above that most of the previous studies were carried out by using k-ε turbulence model for the investigation of the sedimentation tank performance.

The main objective of the present work is to validate/calibrate the CFD model in order to investigate the hydraulic characteristics of the rectangular sedimentation tank by using SST k- ω turbulence model.

The shear-stress transport (SST) k-ω model was developed by Menter [1994] to effectively blend the robust and accurate formulation of the k-ω model (Wilcox, 1988) in the near-wall region with the free-stream independence of the k- ε model (Rodi,

XVI
World Water Congress
International Water Resources Association (IWRA)
Cancun, Quintana Roo. Mexico. 29 May- 3 June, 2017

1980)) in the far field. To achieve this, the k- ε model is converted into a k-ω formulation. The SST k-ω model is similar to the standard k-ω model, but includes the following refinements:

- The standard k-ω model and the transformed k- ε model are both multiplied by a blending function and both models are added together. The blending function is designed to be one in the near wall region, which activates the standard k-ω model, and zero away from the surface, which activates the transformed k- ε model.
- The SST model incorporates a damped cross-diffusion derivative term in the ω equation.
- The definition of the turbulent viscosity is modified to account for the transport of the turbulent shear stress.
- The modeling constants are different.

These above features make the SST k-ω model more accurate and reliable for a wider class of flows than the standard k-ω model.

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 flows occurring in a rectangular sedimentation tank, the governing equations for two-dimensional mean flow are as follows:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

Momentum Equations:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}(-\rho\overline{u_i' u_j'}) \tag{2}$$

They have the same general form as the instantaneous Navier-Stokes equations, with the velocities and other solution variables now representing ensemble-averaged (or time averaged) values. Additional terms now appear that represent the effects of turbulence.

These Reynolds stresses, $-\rho\overline{u_i' u_j'}$ must be modelled in order to close Equation (2).

The Reynolds-averaged approach to turbulence modelling requires that the Reynolds stresses in Equation (2) are appropriately modelled. A common method

employs the Boussinesq hypothesis to relate the Reynolds stresses to the mean velocity gradients:

$$-\rho\overline{u_i'u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t\frac{\partial u_k}{\partial x_k}\right)\delta_{ij}$$

……………………………………(3)

Where k is the turbulent kinetic energy, and μ_t is the turbulent or eddy viscosity. Whereas the dynamic viscosity μ is a fluid property, the eddy viscosity strongly depends on the state of turbulence.

2.2 The SST k-ω Turbulence model

The transport equations for the SST k-ω Turbulence model as follows (Fluent's 14.5 User Guide):

Kinematic Eddy Viscosity

$$\nu_T = \frac{a_1 k}{\max(a_1\omega, SF_2)}$$

………………………………………………......(4)

Turbulence Kinetic Energy

$$\frac{\partial k}{\partial t} + U_j\frac{\partial k}{\partial x_j} = P_k - \beta^* k\omega + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_k\nu_T)\frac{\partial k}{\partial x_j}\right]$$

………………………….(5)

Specific Dissipation Rate

$$\frac{\partial\omega}{\partial t} + U_j\frac{\partial\omega}{\partial x_j} = \alpha S^2 - \beta\omega^2 + \frac{\partial}{\partial x_j}\left[(\nu + \sigma_\omega\nu_T)\frac{\partial\omega}{\partial x_j}\right] + 2(1 - F_1)\sigma_{\omega2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial\omega}{\partial x_i}$$

…………………………………………………………………………………………….(6)

Closure Coefficients and Auxilary Relations

$$F_2 = \tanh\left[\left[\max\left(\frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right)\right]^2\right]$$

$$P_k = \min\left(\tau_{ij}\frac{\partial U_i}{\partial x_j}, 10\beta^* k\omega\right)$$

$$F_1 = \tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y}, \frac{500\nu}{y^2\omega}\right), \frac{4\sigma_{\omega2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\}$$

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial\omega}{\partial x_i}, 10^{-10}\right)$$

$$\phi = \phi_1 F_1 + \phi_2(1 - F_1)$$

$$\alpha_1 = \frac{5}{9}, \alpha_2 = 0.44$$

$$\beta_1 = \frac{3}{40}, \beta_2 = 0.0828$$

XVI
World Water Congress
International Water Resources Association (IWRA)
Cancun, Quintana Roo. Mexico. 29 May- 3 June, 2017

$$\beta^* = \frac{9}{100}$$

$$\sigma_{k1} = 0.85, \sigma_{k2} = 1$$

$$\sigma_{\omega1} = 0.5, \sigma_{\omega2} = 0.856$$

3. NUMERICAL MODEL

Steady-state incompressible flow conditions with viscose effects are generally considered in hydraulic numerical modelling, and the Navier–Stokes equation has been well modified to solve the governing equation. The Navier–Stokes equation is an incompressible form of the conservation of mass and momentum equations, and is comprised of non-linear advection, rate of change, diffusion, and source terms in the partial differential equation. The mass and momentum equations joined through velocity can be used to obtain an equation for the pressure term. When the flow field is turbulent, the computation becomes more complex. Hence, the RANS equations, which are modified forms of the Navier–Stokes equation including the Reynolds stress term, which approximates the random turbulent fluctuations by statistics, are prevalently used.

In this study, the available computational fluid dynamics (CFD) program Fluent 14.5, developed by Ansys, was used for the numerical simulation. Fluent solves the RANS equations by the finite volume formulation obtained from a rectangular finite difference grid. For each cell, the average values of the flow parameters, such as pressure and velocity, are computed at discrete times. The new velocity in each cell is calculated from the coupled momentum and continuity equations using previous time step values in each center of the face of cells. The pressure term is obtained and adjusted using the estimated velocity to satisfy the continuity equation. With the computed velocity and pressure for later time, the remaining variables, including turbulent transport, density advection and diffusion, and wall function evaluation, are estimated.

3.1 Validation Test

To validate the software simulation of settling tanks, the experimental conditions used by Razmi et al (2009) were considered.

Razmi et al (2009, established an open channel loop at the Fluid Mechanics Laboratory of the Mechanical Engineering Department of Sharif University of Technology (Iran), the open channel was originally designed to demonstrate the hydraulic characteristics and performance of an open rectangular sedimentation channel was utilized for the experiments. A schematic sketch of the channel is shown in Figure (1.0).
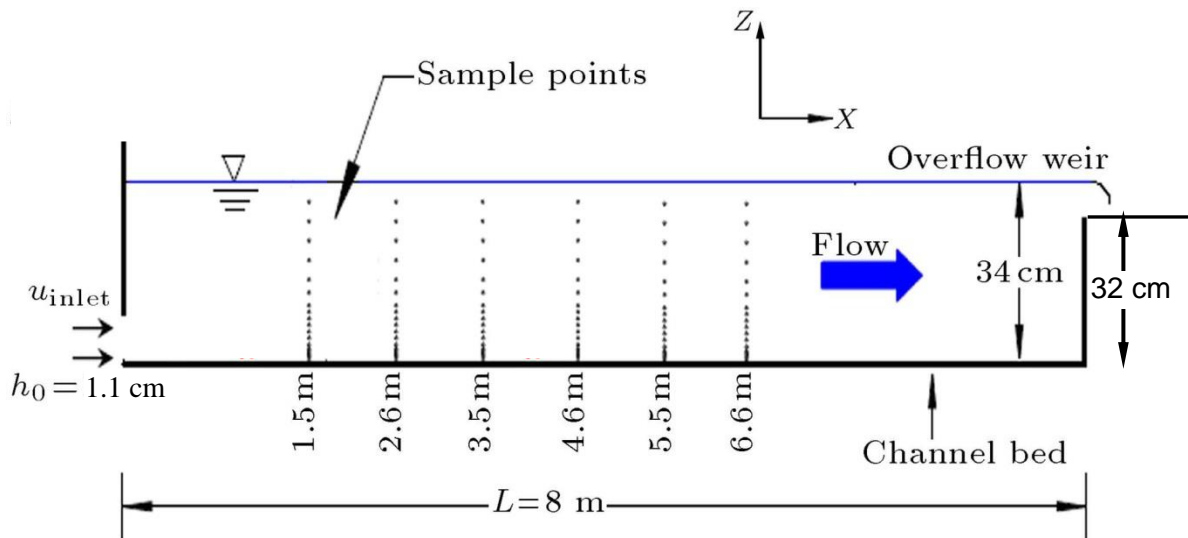
Figure 1.0 Schematic of Open Channel and Measurement Sections - Razmi et al (2009).

Experiments were conducted in a rectangular open channel 8 m x 0.2 m x 0.4 m in length, width and height (x, y and z with x = 0 at the upstream end, y = 0 at the center of the channel and z = 0 at the bed), respectively, with a smooth bottom. A rectangular bottom feed slot with a height of ho=0.011 m extending throughout the full width of the channel provided the inlet gate. The depth of water was controlled by a sharp-edged weir of a height of 32 cm located at the downstream end of the channel.

This tank was selected because performance data are available for model calibration, and because it represents a marginal performance case. After calibration, the model may be used to evaluate different internal configurations.

Experiments were performed to investigate the effect of a baffle on the flow structure at different flow rates in a rectangular sedimentation open channel by measuring flow velocities. The experiments were conducted in with the velocity of the pure water flow was measured at the flow rate of Q = 42 l/min. This flow rate was the maximum established flow rate that could be achieved and, thus, was termed as full flow rate. Table 3.1 summarizes flow conditions in the current experiments. The value of important non-dimensional quantities, such as Reynolds (Re) and Froude (Fr) numbers, at the inlet and in the channel has been illustrated for the test flow rates. The Reynolds and Froude numbers are depend by the following relationships:

$$Re = \frac{U_{ch}.L_{ch}}{v}$$ ……………………………………………………….………...(1)

$$Fr = \frac{U_{ch}}{(g.L_{ch})^{0.5}}$$ ……………………………………………………… (2)

In which Lch and Uch are characteristic length and velocity, respectively. At the inlet, Re and Fr numbers are depend in terms of inlet height (ho) and inlet bulk velocity. v is the kinematic viscosity of the water at T = 20°C. The Reynolds number inside the channel is calculated based on the average bulk velocity inside the channel and the hydraulic diameter of the channel.

Table (2) Experimental Conditions- Razmi et al (2009)

Item	Unit	Value
Inlet Flow (Q)	l/min	42
Re (In the Channel)	-	3016.93
Re (Inlet)	-	3486.06
Fr (In the Channel)	-	0.0056
Fr (Inlet)	-	0.0353

Although the inlet Froude numbers is less than one (subcritical), it is very small inside the channel and, thus, the effect of free surface can be neglected. Despite this small number, the similitude of Froude was used, similar to Lyn and Rodi [1990]. In the absence of sediment transport or density differences, the Froude and Hazen numbers have no significant effect. The inlet Re number indicates that the inlet flow is turbulent. Similar to the work of Lyn and Rodi [1990] the traditional hydraulic modelling criteria have been borrowed from pipe studies and Re > 2000 was considered the criterion for turbulent flow. Thus, the values of Re number in the channel indicate that the operating flow remains turbulent, regardless of value of Fr number. A typical time series of the streamwise velocity component, u(t), measured by ADV. Razmi et al (2009) investigated numerically the same open channel sedimentation tank by using K-ε and VOF models to investigate the hydraulic features of the sedimentation tank. In this study, a CFD model on the open channel tank will be generate and compare the results with those results obtained by Razmi et al (2009).

3.2   Model Assumptions:

To limit the computational time requirements, the following assumptions are considered as per Razmi et al (2009):

1) The flow field is the same for all positions (in Y-direction); therefore, a 2D geometry can be used to properly simulate the general features of the hydrodynamic processes in the tank.
2) The water free surface was modelled as a fixed surface; this plane of symmetry was characterized by zero normal gradients for all variables.

3.3   The operating Conditions

The sedimentation tank was investigated for a specific set of conditions used in the Laboratory. The inlet was specified as a plug flow of water at 42 l/min whereas the inlet turbulence intensity was set at 4.5%. The outlet was specified as a constant pressure outlet with a turbulence intensity of 6.0%.

3.4   Sedimentation Tank Geometry

The tank dimensions according to Figure (1.0) will be summarized as follows:

Table (3) The sedimentation tank Dimensions, Razmi et al (2009).

Item	Unit	Value
Tank Length	m	8
Tank Width	m	0.2
Tank Height	m	0.34
Flow Area inside channel	m2	0.068
Flow Rate	l/min	42.00
U_{av} of flow Inside Tank	m/sec	0.010
Hydraulic Diameter	m	0.294

3.5 Mesh Generation

A grid dependency study was performed by Razmi et al (2009) for VOF model to eliminate errors due to the coarseness of the grid and also to determine the best compromise between simulation accuracy, numerical stability, convergence, and computational time. More than 680*53 mesh points were required before the velocity contours changed to independent grid; then, 680*53 grids were chosen by Razmi et al (2009) for the computation.

In addition, a grid dependency study was performed during this study, the mesh density was chosen such that the grid was finest where velocity gradients are expected to be largest. The selected grid was comprised of 46,013 quadrilateral elements. The finest grid with 85,753 elements was also studied but no difference in the results for both grids. Therefore, the first grid (46,013 elements) was considered as a grid independent. Figure 2.0 presents the selected mesh generation of the sedimentation tank geometry for the present study.
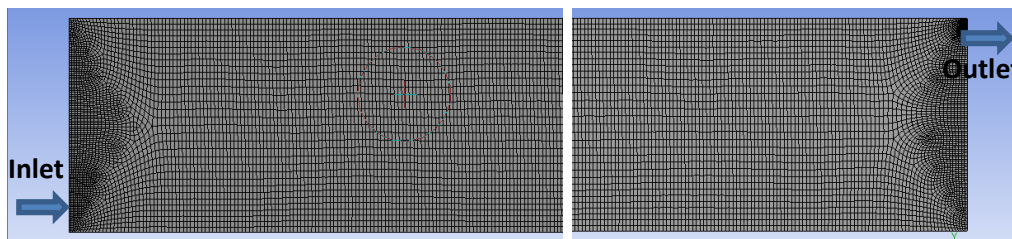

Figure 2.0 Mesh Generation of the Sedimentation Tank Geometry for the Present Study.

3.6 Boundary Conditions

The above equations form a set of partial differential equations. In order to obtain a unique solution, this set needs to be linked to a set of boundary conditions. The boundary conditions include:

1. The inlet was specified as a uniform velocity, k and ω values (i.e velocity Inlet). Whereas the inlet turbulence intensity was set at 4.5%.
2. The overflow outlets were specified in the top row of cells on either side of the overflow weirs. The outlet is specified as a constant pressure outlet with a turbulence intensity of 6.0%.
3. The vertical and horizontal walls were specified as near wall boundaries.
4. The water level is a static free surface was specified as a rigid lid symmetry axis. The variables that were not specified according to a rigid lid symmetry axis were the concentration (for which a zero flux boundary was applied) and the kinetic energy dissipation.

3.7 Simulation Setup
The setup of the CFD modelling by using Fluent 14.5 will be as follows:
a) Solver

Type	: Pressure-Based
Velocity Formation	: Absolute
Time	: Steady
2D Space	: Planer
Gravity acceleration	: -9.81 m/sec in z-direction.

b) Model

Viscous Model	: k-ω(2eqn), SST

c) Boundary Conditions

Boundary	Item	Values
Inlet	Type Velocity Magnitude (m/sec) Turbulent Intensity (%) Hydraulic Diameter (m)	Velocity Inlet 0.035 4.5 0.159577
Outlet	Type Gauge Pressure (pascal) Turbulent Intensity (%) Hydraulic Diameter (m)	Pressure Outlet 0 6 0.071365
Side Walls	Type Wall Motion Shear Condition	Wall Stationary Wall No Slip
Tank Top	Type	Symmetry
Tank Bottom	Type Wall Motion Shear Condition	Wall Stationary Wall No Slip

e) Solution

Solution Method

Pressure-Velocity Coupling	:	PISO
Skewness Correction	:	1
Neighbour Correction	:	1

Spatial - Discretization

Pressure	:	Standard
Momentum	:	Second Order Upwind
Turbulent Kinetic Energy	:	Second Order Upwind
Specific Dissipation Rate	:	Second Order Upwind

Solution Controls
Under-Relaxation Factor

Pressure	:	0.6
Density	:	1
Body Force	:	1
Momentum	:	0.8
Turbulent Kinetic Energy	:	0.9
Specific Dissipation Rate	:	0.8

Turbulent Viscosity	:	1
Monitors		
Residual	:	10^{-6}

4. RESULTS AND DISCUSSION

The main objective of the comparison between the results of the basic study (by Razmi et al (2009) and the present study is to investigate the agreement degree between both results in order to validate and calibrate the model used and can apply on similar phenomena and conditions.

4.1 Flow Pattern

Computed stream lines for the basis results (by Razmi et al (2009) and the present study are shown in Figure 3.0. a large circulation zone present in the surface of the settling tank which occupies about 14.24 % of the total tank volume in the results of Razmi et al (2009) and about 15% in the present study.
The streamline/flow pattern computed in the present study is in a good agreement with that result of Razmi et al (2009).
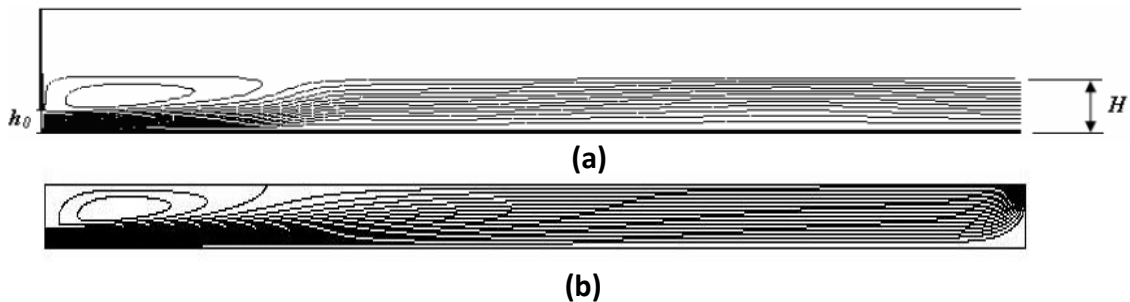

(a)

(b)

Figure 3.0 Predicted streamlines for the rectangular sedimentation tank for; a) K-ϵ & VOF model by Razmi et al 2009 , b) SST K-ω model for the present study

4.2 Turbulent Kinetic Energy (k)

The kinetic energy profile and contours for Razmi et al, 2009 and present validations are shown in Figures 4.0 & 5.0 respectively.
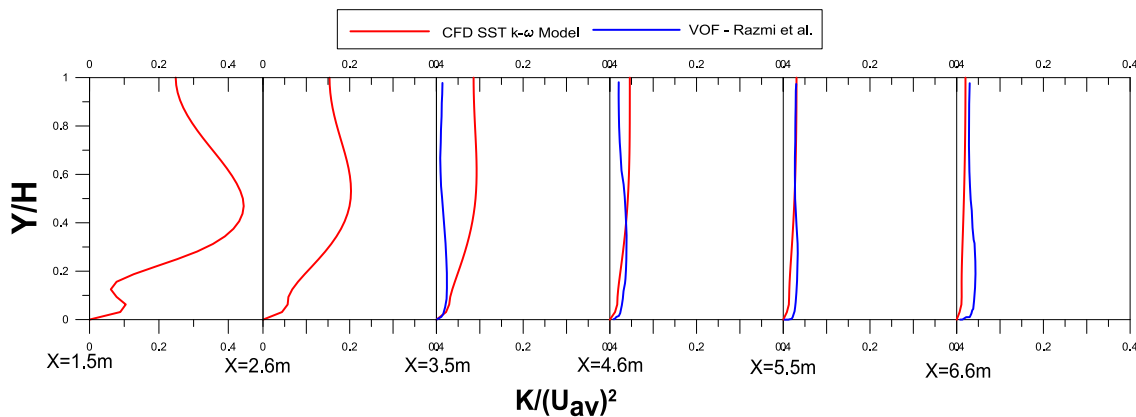

Figure 4.0 The turbulence kinetic energy profile at different position of the rectangular sedimentation tank for; a) K-ϵ & VOF model by Razmi et al, 2009; b) the present study
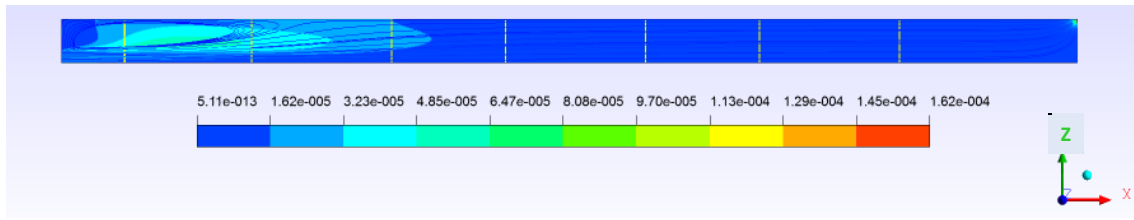
Figure 5.0 Contours of turbulent kinetic energy (m^2 s^{-2}) for the sedimentation tank.

The results indicate that, the turbulence kinetic energy reduced/dissipated at the stations that far away from the tank inlet.

4.3   Velocity Profiles and Contours

The velocity contours for Present Validations are shown in Figure 6.0.

Inlet Zone   Outlet Zone
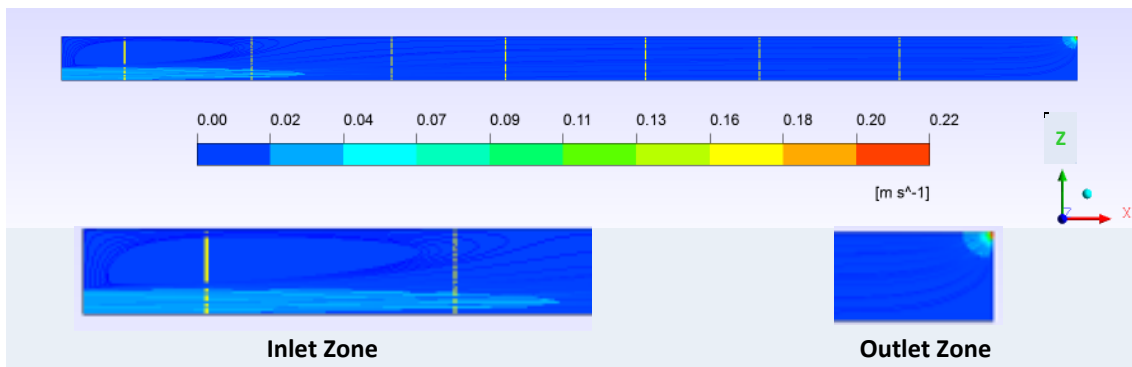
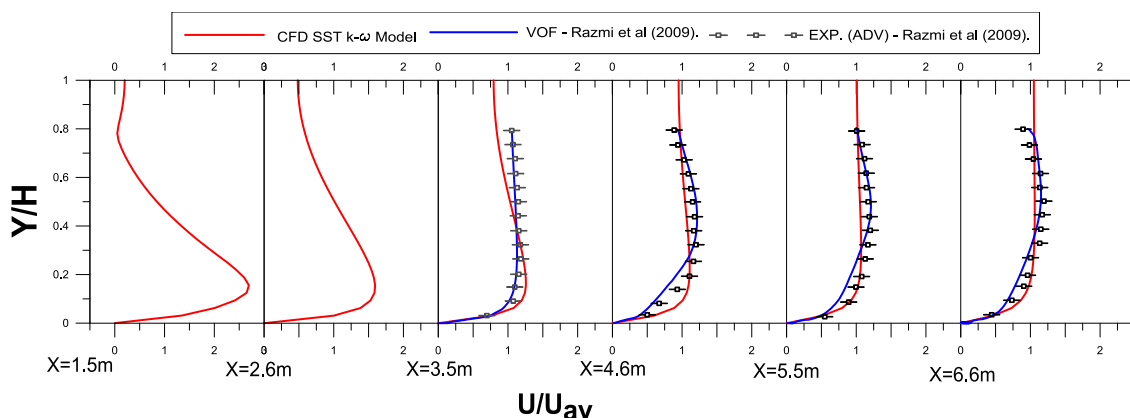**Figure 6.0 Contours of velocity contours inside the sedimentation tank**

Figure 7.0 Velocity profile (U/Uav) at different location inside the sedimentation tank a) K-ϵ & VOF Model by Razmi et al, 2009; b) the present study

The velocity profile at different location inside the sedimentation tank for Razmi et al, 2009 and present study are shown in Figure 7.0.

XVI
World Water Congress
International Water Resources Association (IWRA)
Cancun, Quintana Roo. Mexico. 29 May- 3 June, 2017

In velocity distribution graphs in Figure 7.0, the horizontal axis represents the ratio of time-averaged streamwise velocity (U) to average bulk velocity (U_{Ave}) in the channel, and the vertical axis is the ratio of height at each measurement point (h) to the total flow depth (H = 34 cm).

It is noticeable that measurements near the free surface was not conducted by Razmi et al, because if the downward-looking probe of ADV were out of water, the change of sonic speed in the air and water would lead to poor quality data. Thus, the measurements performed only up to h=H = 0.8.

At a distance from the inlet, flow tends to be more developed. The first reason (which can be seen in Figure 7.0 is that although part of the velocity profiles, at x = 1.5 m have negative values, the other velocity profiles in the five other measuring sections have positive values over the entire flow depth and, therefore, flow travels toward the outlet. Secondly, at a distance from the inlet, velocity profiles tend to become closer to their expected developed shape (u/Uav=1); this means that they tend to fit themselves to the boundary conditions of the channel, such as the effect of the bed's wall and free surface.

It is observed that the numerical data (Razmi et al 2009 and the present study) match the laboratory results very well obtained by Razmi et al 2009.

5. CONCLUSION AND RECCOMENDATION

It is observed that the numerical data (Razmi et al 2009 and the present study) match with the laboratory results very well. But near the surface and close to the bed, some errors were observed by Razmi et al 2009.

Therefore, the investigation of the hydrodynamic behaviour of the sedimentation tanks by using SST k-ω turbulence Model will provide accurate results which were in good agreement with the experimental results.

In addition to the above, the improvement /modifications to improve the performance/ efficiency of the sedimentation tanks can be predicated by CFD model with SST k-ω turbulence model.

REFERENCES

A. Razmi, B. Firoozabadi, and G. Ahmadi, 2009, "Experimental and Numerical Approach to Enlargement of Performance of Primary Settling Tanks", Journal of Applied Fluid Mechanics, Vol. 2, No. 1, pp. 1-12.

A. Tamayol, B. Firoozabadi and G. Ahmadi, 2008, "Effects of Inlet Position and Baffle Configuration on Hydraulic Performance of Primary Settling Tanks", Journal Of Hydraulic Engineering © ASCE, JULY 2008, pp. 1004- 1009,

A. Tamayol, B. Firoozabadi and G. Ahmadi, 2008,"Determination of Settling Tanks Performance Using an Eulerian – Lagrangian Method", Journal of Applied Fluid Mechanics, Vol. 1, No. 1, pp. 43-54.

A. Tamayol, B. Firoozabadi and M. A. Ashjari, 2010, "Hydrodynamics of Secondary Settling Tanks and Increasing Their Performance Using Baffles", Journal Of Environmental Engineering © ASCE , pp. 32-39.

American Water Work Association (AWWA), 1990, "Water Treatment Plant Design", McGraw-Hill, New York.

Anderson N.E., 1945, "Design of settling tanks for activated sludge", Sewage Works J., 17(1), pp. 50-63.

Ansys Fluent, 2012, "Users Guide", release 14.5.

B. Firoozabadi and M.A. Ashjari, 2010, "Prediction of Hydraulic Efficiency of Primary Rectangular Settling Tanks Using the Non-linear k-ε Turbulence Model", Transaction B: Mechanical Engineering © Sharif University of Technology, Vol. 17, No. 3, pp. 167-178.

B. Firoozabadi, H. Asgharzadeh and H. Afshin, 2011, "Experimental Investigation of Effects of Baffle Configurations on the Performance of a Secondary Sedimentation Tank", Scientia Iranica, Transactions B: Mechanical Engineering, Vol. 18, pp. 938–949.

Camp T.R, 1946, "Sedimentation and the design of settling tanks", Trans. ASCE, 111(6), pp. 895-952.

Campbell B.K. and Empie H.J., 2006, "Improving fluid flow in clarifiers using a highly porous media", J. Env.Engng., 132(10), pp. 1249-1254.

Cripps S.J. and Bergheim, 2000, "Solids management and removal for intensive land-based aquaculture production systems", J. Aqua-Cultural Engng., pp. 33-56.

De Clercq, Kinnear and Vanrolleghem, 2003, "On-line dynamic fluid velocity profiling in secondary clari ers", Biomath, Ghent University, Coupure Links 653, B-9000 Belgium.

Gh. Naser, B. W. Karney, and A. A. Salehi, 2005, "Two-Dimensional Simulation Model of Sediment Removal and Flow in Rectangular Sedimentation Basin", Journal of Environmental Engineering © ASCE, pp. 1740-1749.

H. Jamshidnia and B. Firoozabadi, 2010, "Experimental Investigation of Baffle Effect on the Flow in a Rectangular Primary Sedimentation Tank", "Transaction B: Mechanical Engineering, Vol. 17, No. 4, pp. 241-252."

Krebs P., Vischer D. and Gujer W., 1995, "Inlet-structure design for Final

clarifiers", J. Env. Engng., 121(8), pp. 558-564.

Krish RAMALINGAM, John FILLOS, Savvas XANTHOS, Minwei GONG, Allen DEUR and Keith BECKMANN, 2007, "Predicting SS Removal Performance in Rectangular Settling Tanks (FSTs) Using 3-Dimensional CFD Model", Proceedings of the 10th International Conference on Environmental Science and Technology Kos island, Greece, 5–7 September 2007, PP. B693-B701.

M. Latsa, D. Assimacopoulos, A. Stamou and N. Markatos, 1999, "Two-Phase Modeling of Batch Sedimentation", Applied Mathematical Modelling, Vol. 23, PP. 881-897.

McCorquodale J. Alex, Griborio Alonso, and Georgiou Ioannis, 2006, "Application of A CFD Model To Improve The Performance of Rectangular Clarifiers", Proceedings of the Water Environment Federation, WEFTEC 2006: Session 1 through Session 10, pp. 310-320(11).

O. Flamant, A. Cockx, V. Guimet and Z. Doquang, 2004, "Experimental Analysis and Simulation of Settling Process", Process Safety and Environmental Protection-IChemE, Vol. 82(B4), pp. 312–318.

Overcamp T.J., 2006, "Type II settling data analysis", J. Env. Engng., 132(1), pp. 137-139.

Patricia Rodrıguez Lopez, Antonio Guti´errez Lav´ın, Manuel M. Mahamud Lopez and Julio L. Bueno de las Heras, 2008, "Flow Models For Rectangular Sedimentation Tanks", Chemical Engineering and Processing, Vol. 47, pp. 1705–1716.

R. Kahane, T. Nguyen, M.P. Schwarz, 2002**,** CFD modelling of thickeners at Worsley Alumina Pty Ltd, Appl. Math. Model. 26 (2002) 281–296.

Scott Arthur Lowe, 1990,"Numerical and Experimental Modelling of Sedimentation Tanks", Doctor of Philosophy thesis, Department of Civil and Mining Engineering, University of Wollongong, http://ro.uow.edu.au/theses/1253.

Swamee, P.K. and Tyagi, 1996, "Design of class-I sedimentation tanks", J. Env. Engng., 122(1), pp. 71-73.

Van Der Walt, 2002, "The Modelling of Water Treatment Process Tanks", Thesis submitted for the degree of Doctor of Philosophy of the Rand Afrikaans University, South Africa.

XVI
World Water Congress
International Water Resources Association (IWRA)
Cancun, Quintana Roo. Mexico. 29 May- 3 June, 2017