Computational Analysis of Sedimentation Process in the Water Treatment Plant

Tulus1, Suriati2, M Situmorang1 and D M Zain1

1Department of Mathematics, University of Sumatera Utara, Jl. dr. Mansur Kampus USU Medan 20155, Indonesia
2Department of Informatics, Sekolah Tinggi Teknik Harapan Medan, Jl. HM Joni No.70 Medan 20218, Indonesia

Email: tulus@usu.ac.id

Abstract. This study aims to determine how the distribution of sludge concentration and velocity of water flow in the water treatment plant in equilibrium state. The problems are solved by implementing the finite element method to a momentum transport equation which is a basic differential equation that is used for liquid-solid mixtures with high solid concentrations. In the finite element method, the flow field is broken down into a set of smaller fluid elements. The domain is considered as a container in the space of three-dimensional (3D). The sludge concentration distribution as well as the water flow velocity distribution in the inlet, central and outlet are different. The results of numerical computation are similar compared to the measurement results.

1. Introduction
The Regional Water Company is a regional-owned company engaged in the processing and clean water industry for the civil society. There are several types of water treatment processes: filtering and precipitation, coagulation, flocculation, sedimentation, filtration, and disinfection. In the untreated water, there is a process of sedimentation in the water, so it must be processed firstly through the process of clean water to be feasible in consumption for the civil society.

Sedimentation as a process of sediment or sediment formation caused by the precipitation of the forming material or its origin in a place called the settling environment of a river, estuary, lake, delta, estuary, shallow sea to deep sea [1,2]. This sedimentation does not occur directly, before it reaches the bottom and becomes sediment, the substance hovers in a liquid. Sediments are fractions, minerals, or organic materials that are transformed from various sources and deposited by air, wind, ice, or water media, and include deposited materials from materials floating in water or in the form of chemical solutions [3,4]. Some studies on the sedimentation process have been performed by the author for the case of irrigation channels in the agriculture sectors [5,6]. The derivation of the phenomenological theory for settling of sedimentation with compression was conducted by Concha and Bustos [7]. This study aims to find out how the distribution of sludge concentration and flow velocity of water in water container vessel premises using the implementation of finite element method at the momentum transport equation.
2. Numerical Methods using Finite Elements
The finite element method is a numerical method used to solve engineering problems and mathematical problems of a physical phenomenon [8]. In the finite element method, the flow field is broken down into a set of small fluid elements. The analysis of the distribution of sludge concentrations and the velocity of the water in this water reservoir is complex and requires considerable calculations. Therefore, to finish this final project software used COMSOL Multiphysics software. The momentum transport equation is a basic differential equation used for a solid-liquid mixture with a high solid concentration. The momentum transport equation can be expressed as follows.

\[
\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p - \nabla \cdot \left( \rho c_s \left( 1 - c_i \right) \mathbf{u}_\text{slip} \right) + \nabla \cdot \left[ \eta \left( \nabla \mathbf{u} + \nabla \mathbf{u}^T \right) \right] + \rho g \tag{1}
\]

where \( \mathbf{u} \) is the mass averaged mixture velocity (m/s), \( p \) denotes the pressure (Pa), \( g \) refers to the acceleration of gravity (m/s\(^2\)), \( c_s \) is the dimensionless particle mass fraction, and \( \mathbf{u}_\text{slip} \) gives the relative velocity between the solid and the liquid phases (m/s), and \( \eta \) is the mixed viscosity (Ns/m\(^2\)).

3. Finite element analysis for Sedimentation
For the issue of sedimentation this study assumes the following:

i. Compressible fluid,
ii. Laminar flow,
iii. Water flows and does not rotate in water reservoirs.

The momentum transport equations as the governing equations in this study are as follows.

\[
\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot \left[ \rho \left( \mathbf{u} \cdot \nabla \mathbf{u} + \nabla \mathbf{u}^T \right) \right] - \nabla \cdot \left( \rho \mu_c \left( 1 - c_i \right) \mathbf{u}_\text{slip} \right) + \rho g + \mathbf{F} \tag{2}
\]
\[
\rho (\nabla \cdot \mathbf{u}) = 0 \tag{3}
\]

3.1. Initial and Boundary Conditions
The boundary conditions are considered with respect to the real condition of the model, that are wall, slip wall, inlet, middle and outlet, as shown in the equations (4) - (7).

Wall: \( \mathbf{u} \cdot \mathbf{n} = 0 \) \( \tag{4} \)

Inlet: \( \mathbf{u} = \mathbf{u}_0, \left[ \left( \mu + \mu_f \right) \nabla \mathbf{u} + \nabla \mathbf{u}^T \right] - \nabla \cdot \left( \rho \mu_c \left( 1 - c_i \right) \mathbf{u}_\text{slip} \right) + \rho g \right] \mathbf{n} = 0 \) \( \tag{5} \)

Middle: \( \mathbf{u} = \mathbf{u}_0, \left[ \left( \mu + \mu_f \right) \nabla \mathbf{u} + \nabla \mathbf{u}^T \right] - \nabla \cdot \left( \rho \mu_c \left( 1 - c_i \right) \mathbf{u}_\text{slip} \right) + \rho g \right] \mathbf{n} = 0 \) \( \tag{6} \)

Outlet: \( \mathbf{u} = \mathbf{u}_0, \left[ \left( \mu + \mu_f \right) \nabla \mathbf{u} + \nabla \mathbf{u}^T \right] - \nabla \cdot \left( \rho \mu_c \left( 1 - c_i \right) \mathbf{u}_\text{slip} \right) + \rho g \right] \mathbf{n} = 0 \) \( \tag{7} \)

3.2. Simulation using COMSOL Multiphysic
The container is modeled in three-dimensional (3D). The domain of the problem is modeled as depicted in the Figure 1(a), and the mesh of the domain is depicted in Figure 1(b). The materials and variables used are listed in Tables 1 and 2, respectively, to obtain the appropriate model.
Figure 1. Model: (a) Geometry and (b) Tetrahedron mesh of the geometry.

Table 1. Specification of the material properties.

| No | Parameter                        | Symbol | Value | Unit   |
|----|----------------------------------|--------|-------|--------|
| 1  | Continuous phase density         | $\rho_c$ | 1000  | kg/m³  |
| 2  | Continuous phase viscosity       | $\mu$  | 0.001 | Pa·s    |
| 3  | Dispersed phase density          | $\rho_d$ | 1500  | kg/m³  |
| 4  | Dispersed phase particle diameter| $D_d$  | 2E-4  | m      |

The parameters used in the computation are considered as a description for the material properties of water and the averaged diameter of particle dispersed in water as described in Table 1. On the other hand, some values are given in initial values of the computation with deal to the condition in the channel as listed in the Table 2.

Table 2. Some variables that used in the computation.

| No | Variables                           | Symbol | Value                     | Unit     |
|----|-------------------------------------|--------|---------------------------|----------|
| 1  | Inlet velocity                      | $v_{in}$ | 1.25*step function        | m/s      |
| 2  | Outlet velocity                     | $v_{out}$ | 1.25*step function        | m/s      |
| 3  | Inlet dispersed phase volume fraction| $\phi_{d,in}$ | 0.003 | -                |
| 4  | Dispersed phase mass-out flux       | $q_{d,out}$ | $2\pi\phi_{d,in}$          | kg/(m·s) |

4. Results and Discussions

This section will show the distribution of velocity and volume fraction of the dispersed phase in the vessel in some units of time. Figure 2 shows the distribution of velocity in the vessel for some slices of the geometry on the stable condition. The computational observation is performed to the distribution of velocity. There are some ununiformly velocity that can cause a turbulence in the vessel, especially in the region near inlet. Figure 2 shows also that the maximum velocity occurs near of inlet of the vessel. The velocity become slower in the position near outlet.
Figure 2. Water velocity distribution.

Figure 3(a) shows the graph of concentration on the surface of the vessel. On the other hand, Figure 3(b) shows the distribution of the sludge concentration at each slice in the vessel. Comparing the Figure 3(b) with the Figure 2, it can be seen the relationship between the mixture velocity and sedimentation rate. The smaller the speed of water, the faster the rate of sedimentation. The maximum concentration occurs in the middle of the vessel.

Figure 3. Mixture distribution: (a) on the surfaces and (b) on the slice of the geometry.

The results of the computational simulation are compared with the measurement of the sediment in the water. Figure 4 (a) and (b) show the graphics of the concentration from the computational results and measurement, respectively. From the Figure 4(a) and (b), it can be seen that the physical phenomenon of the concentration of sludges in the water is similar with respect to the results from the calculation and from the measurements. The concentrations in the region near outlet are higher than the concentrations in the middle and near inlet.
5. Conclusions
A 3D numerical computation based on momentum transport equation for the mixture is presented to search the relationship between the sediment processes by using computational method and measurements. From a computational point of view, results have shown the concentration of sediments in the vessel have a similar form from the both of method, computations and measurements. From a physical point of view, results highlighted the concentrations of sediment in the vessel.

Acknowledgments
The authors gratefully acknowledge that the present research is supported by Ministry of Research and Technology and Higher Education Republic of Indonesia. The support is under the research grant TALENTA USU of Year 2017.

References
[1] Ancey C, Davison A C, Böhm T, Jodeau M and Frey P 2008 J. Fluid Mech. 595 83–114
[2] Ballio F and Radice A 2015 Int’l J. Sediment Res. 30 321–327
[3] Mao Z, Zhao S and Luo S 2005 Study on the separation zone in open channel confluence Adv. in Water Sci. 6(1) 7–12
[4] Liu T, Chen L and Fan B 2012 Int’l J. Sediment Res. 27 178-187
[5] Tulus and Situmorang M 2017 Modeling of Sedimentation Process in the Irrigation Channel IOP Conf. Series: Mat. Sci. Eng. 180 012021
[6] Tulus, Suriati and Situmorang M 2017 Computational Analysis of Suspended Particles in the Irrigation Channel IOP Conf. Series: J. Phys. 801 012094
[7] Concha F and Bustos M C 1985 Theory of sedimentation of flocculated fine particles Proc. EF Conf. Flocculation, Sedimentation and Consolidation 275-284
[8] Cao Z, Pender G, Wallis S and Carling P 2004 J. Hydraulic Eng. 130(7) 689–703