Effects of Turbulence Models on Micro-bubble Distribution in Dissolved Air Flotation Process for Water Treatment

Min A Park, Sejong University, Republic of Korea
Kyun Ho Lee, Sejong University, Republic of Korea
Jae Dong Chung, Sejong University, Republic of Korea

The Asian Conference on Sustainability, Energy, and the Environment 2015
Official Conference Proceedings

Abstract
The dissolved air flotation (DAF) system is one of the water treatment processes that clarifies wastewaters by floating internal contaminants to the water surface by attaching micro-bubbles to them. Since the first use of DAF for drinking water treatment in Scandinavia during the 1920s, DAF has been very widely used in treating industrial wastewater effluents such as water treatment.
In the present study, the two-phase flow of micro air bubbles and water mixture is simulated by computational fluid dynamics to investigate changes of internal flow behaviors in a DAF system depending on the turbulence models. For a given geometry of a DAF system and boundary conditions, micro-bubble distribution is analyzed with several turbulence models, which are standard k-ε, realizable k-ε, RNG k-ε, standard k-ω, and SST k-ω, respectively. Through analysis, it is observed that the standard k-ε model, which has been frequently used in the previous studies, predicts somewhat different behaviors from other turbulence models. Also, the RNG k-ε and standard k-ω model yield relatively excessive rotational flow inside water. When computation times are compared, the all models are faster to complete computations except for the k-ε model.
From the present results, it is revealed that the selection of a turbulence model should be considered more carefully when an internal flow analysis is conducted on the DAF process.

Keywords: Dissolved Air Flotation, Computational Fluid Dynamics, 2-Phase Flow, Turbulence Model

Introduction

The dissolved air flotation (DAF) system is a water treatment system that clarifies floating contaminants by attaching micro-bubbles on the free surface, as shown in Fig. 1. The micro-bubbles are injected into an internal flow by decreasing pressure rapidly and are attached to solid contaminants like algae and green tide. After that, the suspended solids float to surface, and then it is scraped by the skimmer. The internal flow of a DAF system is separated into two sections, which are the contact zone and separation zone. In the contact zone, contained contaminants in wastewater and scattering micro-bubbles from a nozzle are mixed together. In the separation zone, if a large number of micro-bubbles is combined with contaminants, they will float and be removed easily. Clarified water circulates inside of the tank and sinks down to near the perforated plate, which causes bubbles to rise rapidly to distribute uniformly. This also decreases the possibility that micro-bubbles are going to outlet directly.

The maintenance cost of DAF is much cheaper than that of other water treatment systems using sedimentation method because DAF does not need any additional flocculants and facilities are easy to create. For that reasons, many studies and measurements have been conducted to increase clarifying water efficiency. Experimental studies have been conducted on the internal fluid flow in order to improve the treatment efficiency of the DAF system. Lundh and Amato measured the internal bubbles' speed by using a laser doppler velocimeter (LDV) with respect to a small-scale pilot plant, which was used as a standard for determining the efficiency of contaminant removal according to the inflow conditions of the wastewater. However, a method using a experimental pilot plant in order to observe the internal flow field of the DAF tank reveals the disadvantages of excessive cost and time consumption. To compensate for the experimental study's disadvantages, computational techniques have made to it possible to calculate the liquid- and vapor-mixed complex multiphase flows numerically in recent years. So, analyses that simulate various flow phenomena occurring in the DAF system are being actively investigated using CFD (Computational Fluid Analysis) program.

DAF system analysis using CFD in this manner can predict a wide range of fluid flow at a low cost in a relatively short time. However, in order to derive physically appropriate results, it is important to apply a proper analysis model for a mixed flow field inside the tank. When we reviewed the studies, most studies used the standard k-ε model to analyze the DAF system. Therefore, we examined the effect of various turbulence models in that field by using the CFD method.

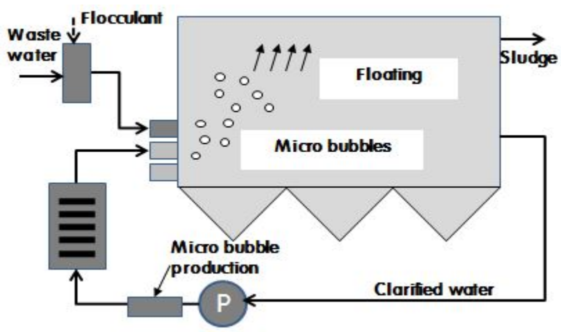


Figure 1: Schematic of a DAF process

Modeling Method

To compare the changes in the flow depending on turbulence model, analysis of the DAF system was performed by adopting the shape and the boundary condition, which were considered in Ryu's study, as shown in Fig. 2. The DAF system size is a length of 9m, and a height of 4m, and the width of the contact zone is modeled as a two-dimensional shape of 0.7 m. We then generated 20,071 structured grids because when we simulate the multiphase flow it changes the distribution of the flow field in accordance with the grid shape, unlike the single-phase flow, which does not depend on grid shape. Also, the waste water from the inlet at the top-right side of the DAF system was introduced. The nozzles were installed on the baffle to inject the micro-bubbles and mix it with the waste water to form a contact zone, and we installed the perforated plate at the bottom.

The boundary conditions applied to the DAF system analysis in this study are summarized in Table 1. First, a velocity inlet condition was used as waste water with micro-bubbles entering from the inlet, and we applied pressure outlet condition to the DAF outlet which set the same pressure as the atmospheric pressure. Then, the degassing condition provided by FLUENT v. 14.5 was used at the water surface to simulate removing the micro-bubbles. In order to better simulate the rotation and laminar flow, the calculation method in the multiphase flow was set to unsteady condition. The diameter of the micro-bubble flowing inside of the DAF system was applied as 120μm, as in Ryu's study and was mixed in the distribution of 0.73% in the circulating water. The percentage of purified water recycling to be used through the inlet was assumed as 10%. Calculation time was set until 2000 sec in time steps of 0.02 sec to observe the distribution of the internal flow field and micro-bubbles after flow fully developed.

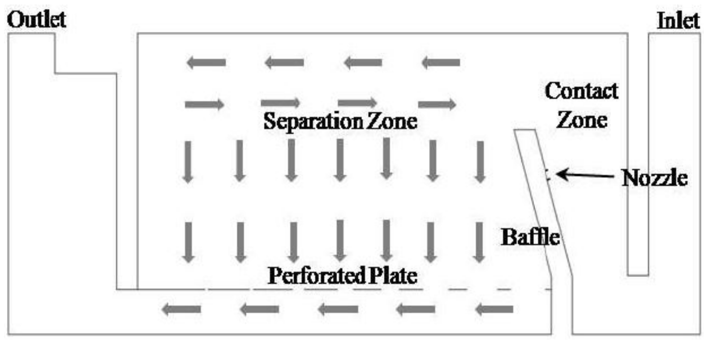


Figure 2: Flow zones of a DAF process

Table 1: Analysis conditions and boundary conditions

Analysis condition		Boundary condition	
Diameter of micro-bubble	120 μm	Waste water inlet	Velocity inlet
Calculation time	2000 sec	Micro-bubble inlet	Velocity inlet
Volume fraction	0.73%	Outlet	Pressure outlet
Flow condition	Unsteady	Water surface	Degassing

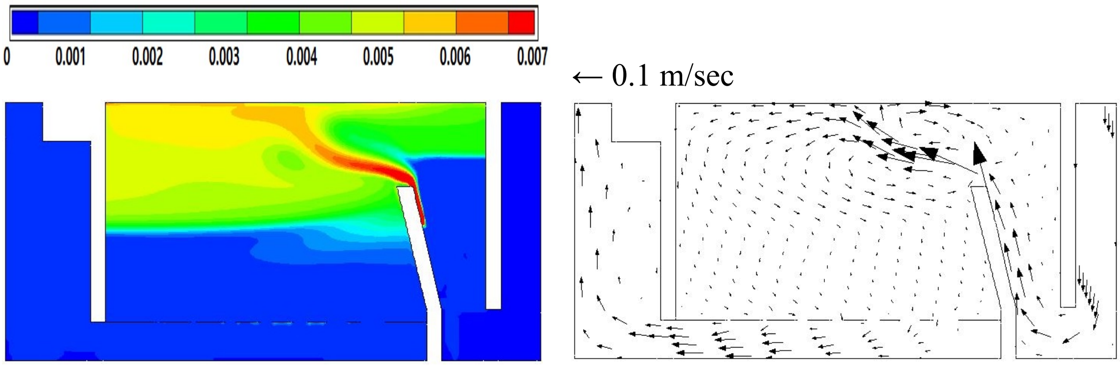
Simulation Results

First, we calculated the change of micro-bubble volume fraction distribution in a two-phase turbulent flow analysis when the five turbulence models provided by the FLUENT were applied to a DAF system. The turbulence models are standard k-ε, realizable k-ε, RNG k-ε, standard k-ω, and SST k-ω, and the micro-bubble volume fraction distributions for each result are presented in Figs. 3(a) - (e).

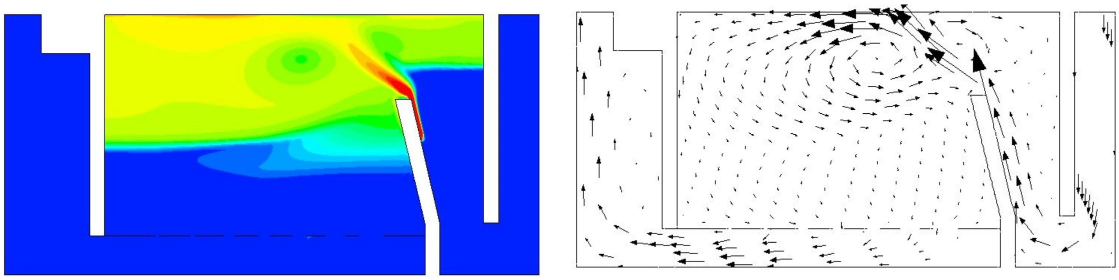
Fig. 3(a) shows the result of standard k-ε, injected micro-bubbles from a nozzle located in the baffle, going over the contact zone and moving across more than half of the separation zone, and then it can be seen that gradually rising to the water surface. On the other hand, when we applied other turbulence models to the DAF system simulation, there were somewhat different tendencies compared with the standard k-ε model. Other models showed very similar results to one another in that micro-bubbles volume fraction are more spaciously distributed in the longitudinal direction and rise to the surface directly, as shown in Figs. 3(e) - (b). The micro-bubbles' average volume fraction was calculated to be about 0.6% while the average of the other results was 0.5% at the mid-point of the surface. The micro-bubbles rise to surface relatively quickly near the baffle because the lateral velocity magnitude underneath the surface is not stronger than the standard k-ε model. It makes micro-bubbles go straight to the opposite side wall making rotating flow, and then go down to the perforated plate. Therefore, other models' micro-bubble volume fraction result at the mid-point of the surface is estimated to be about 0.1% lower than that of the standard k-ε

Meanwhile, the flow simulation results of two turbulence models, which are the RNG k-ε model and realizable k-ε model, were expected to be very similar because they consist of a similar dissipation and turbulent kinetic energy equation. However, the RNG k-ε model predicts the most active circulation flow in the upper part of the separation zone and has a uniform volume fraction distribution over a wide area, as shown in Figs. 3(b) and (c). This was determined due to the RNG k-ε model being theoretically suitable for flow variation having circulation flow with a large deformation, unlike other models. And as shown in Figs. 3(b) and (e), the result of realizable k-ε and SST k-ω models indicate the most similar volume fraction distribution of micro-bubbles.

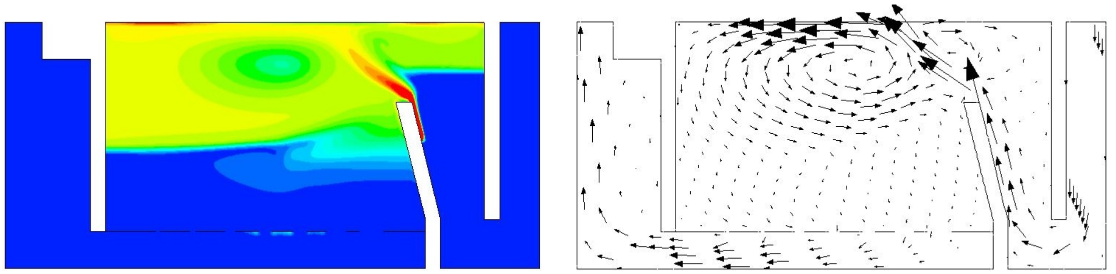
Thus, the difference between the other turbulence models and the standard k-ε model was estimated because the standard k-ε model is suitable for the analysis of a fully developed turbulent flow at a high Reynolds number, while the flow has somewhat low accuracy when flow changes exist like in the case of a circulation flow at a low Reynolds number. To account for this result, the velocity distribution of the internal turbulent flow of each model are compared, as shown in Fig. 4. The maximum speed of the water and micro-bubbles in the separation zone was about 0.3m/sec, and the Reynolds number with respect to micro-bubble diameter was a maximum of 30, which is found to be available for the general water treatment system and represent a low Reynolds number flow. Furthermore, the momentum that rises to surface applied other turbulence model as shown in Figs. 4(b) - 4(e), is smaller than in the results of the velocity vector of Fig. 4(a) applied to the standard k-ε model. As a result, since the flow velocity at the surface is reduced, it can eventually affect rotational flow generated in the separation zone depending on size and intensity compared with other turbulence model results.



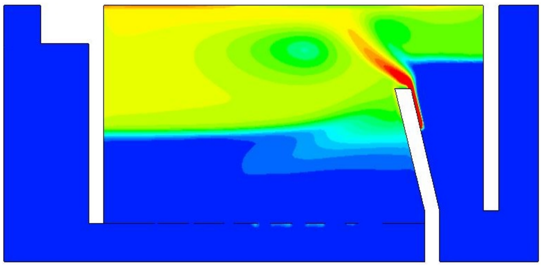
(a) Standard k-ε (a) Standard k-ε



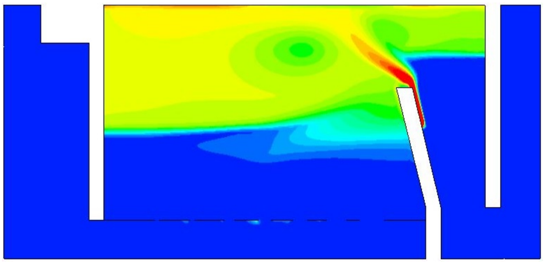
(b) Realizable k-ε (b) Realizable k-ε



(c) RNG k-ε (c) RNG k-ε

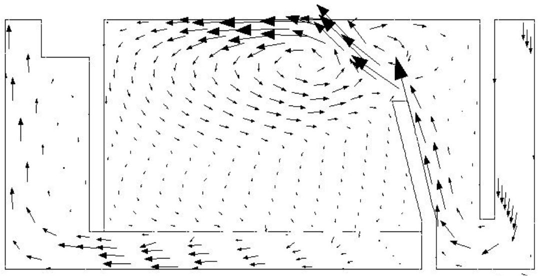


(d) Standard k-ω

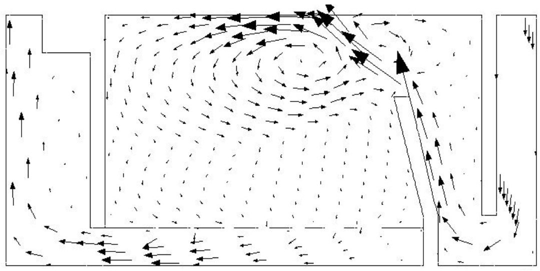


(e) SST k-ω

Figure 3: Volume fraction of various turbulence model



(d) Standard k-ω



(e) SST k-ω

Figure 4: Velocity vectors of various turbulence model

Other turbulence models have improved accuracy at a low Reynolds number flow to complement the standard k-ε model, especially if you are changing the flow, such as rotational flow, the turbulent intensity is predicted to relatively low because the turbulent kinetic energy, dissipation rate, and the viscosity are reduced. So, the simulation of the turbulence flow, including the rotational flow of a low Reynolds area compared to the standard k-ε model, was found to be suitable.
When we simulated the low Reynolds number flow including rotational flow, other turbulence models than a standard k-ε model predicted a mutually similar result of micro-bubble velocity vector distribution and simulated fully developed rotational flow. Thus, simply applying the standard k-ε model for the analysis of DAF system internal flow, as in previous studies, is considered unsuitable.

Conclusion

In this study, we investigated the effect of the turbulence model in the internal flow field of the DAF system by using the CFD method. After applying the five kinds of turbulence models, namely the standard k-ε, realizable k-ε, RNG k-ε, standard k-ω, and SST k-ω, for this study, we implemented comparative analysis of the micro-bubble volume fraction and velocity distribution about each model.
Previous studies typically used the standard k-ε model. In this model analysis result, the micro-bubbles that are injected from the nozzle move through more than half of the separation zone and rise to water surface, showing horizontal behavior. On the other hand, other turbulence models predicted that the volume fraction of micro-bubbles are distributed more widely in the separation zone and immediately rise at the same time due to high longitudinal velocity. When the flow changes, such as in the case of a the low Reynolds number zone, or the circulation flow is estimated somewhat less accuracy because the standard k-ε model is suitable for fully developed turbulent flow analysis with a high Reynolds number flow, if the interpretation of a

low Reynolds number flow to rotational flow is generated as shown in this study, the standard k-ε model cannot be calculated as a relatively fully developed rotational flow. So simply applying the standard k-ε model is not suitable for internal flow analysis of DAF systems, as was done in previous studies.

Acknowledgement

This research was supported by the Small and Medium Business Administration (SMBA) in 2013 (Grants No. S2083708).

References

Kim, Y. M. (2001). Analysis of dissolved air process using computational fluid dynamics. Daejeon: KAIST.

Ryu, G. N., Park, S. M., Lee, H. I. and Chung, M. K. (2010). Numerical study of effect of DAF-Tank shape on flow pattern in separation, *Trans. Korean Soc. Mech. Eng. B,* 8, pp.855-860.

Lundh, M., Jönsson, L. and Dahlquist, J. (2000). Experimental studies of the fluid dynamics in the separation zone in dissolved air flotation, *Water Res.,* 34, pp.21-30.

Amato, T. and Wicks, J. (2009). The practical application of computational fluid dynamics to dissolved air flotation, Water treatment plant operation, design and development*, Journal of Water Supply: Res. Tech. AQUA,* 58, pp.65-73.

Lahghomi, B., Lawryshyn, Y. and Hofmann, R. (2012). Importance of flow stratification and bubble aggregation in the separation zone of a dissolved air flotation tank, *Water Res.,* 46, pp.4468-4476.

ANSYS inc. (2013). *ANSYS fluent theory guide release 15*. Pennsylvania: ANSYS Inc.

Myong, H. G. (2012). *A guide to CFD*. Seoul: Mumundang.

Babaahmadi, A. (2010). Numerical investigation of the contact zone on geometry, multiphase flow and needle valves. Göteborg: Chalmers university of tech.

Seul, K. W., Yoon, D. H. and Ki, N. S. (2013). Thermal-hydraulic detailed analysis inside pipe and tube by using CFD techniques, *Korean Institute of Nuclear Safety.*

Gregory, R. and Edzwald, J. K. (2011). *Water quality and treatment,* Newyork: McGraw-Hill.

Contact email: khlee0406@sejong.ac.kr