Computational Research Progress in Applied Science & Engineering

ISSN 2423-4591

CRPASE Vol. 05(04), 122-126, December 2019

Numerical Simulation of Sediment Transport In Tanks with the Influence of Cavity

A. Terfous*, Y. Liu, A. Ghenaim

INSA Strasbourg, ICUBE - Fluid mechanics

Keywords

Numerical simulation, Sediment tank, Sediment transport, Cavity, Roughness.

Abstract

This paper focused on numerical methods to investigate the flow conditions in sediment tanks with the cavity. The model used for simulation is based on experimental devices, for identifying the effect of cavity on sediment efficiency, the model with no cavity and models with different height of cavity will be investigated. With the effect of cavity, the flow condition in sediment tank would be different, which is the dominate force of the performance of sediment tank, which can also result in the alteration of sediment efficiency. With the change of roughness of boundary, alteration will also be made in sediment transport. With the use of user-defined function in fluent codes, the investigation of different boundary conditions was allowed. The research in this paper mainly aims at promoting the performance of sediment tanks and investigating the factor that will affect sediment transport in tanks.

1. Introduction

For removing particles in sanitary system the sediment tank is used before the effluent is discharged to the external. However the design of the sediment tank could not be obtained before it is constructed, which means the cost spent on constructing a tank will be wasted if the tank could not perform as it was supposed to be. With the development of computer science, it becomes possible to simulate flow in the sediment tank with Computational Fluid Dynamics (CFD) codes, which is also called numerical simulation [1- 3].

Nowadays, more and more researchers choose to investigate the academic research problem by using numerical simulation. Dufresne et al. applied 3D computation fluid dynamic methods to investigate flow, sedimentation and solids separation in a tank equipped with one inlet pipe and two outlets and conducted 23 experiments, it's concluded that the computational fluid dynamics appears as a good way to model sediment transport in a pilot tank and using model in a full scale combined sewer detention tank is to be continued [1]. Lipeme Kouyi et al. carried out the experiment measurements and validation of 3D numerical modeling of the free surface of a storm overflow of a sanitation system, which turned out that both the lag and the velocity profile deformation could predict the deposition pattern in the side-cavity satisfactorily [2]. Kimura and Hosoda clarified that the flows around the side-cavity are characterized by steady recirculations inside the cavity and unsteady vortex shedding due to the shear instability along the interface between the side-cavity and the main channel [3]. Dufresne et al. classified the flow patterns in rectangular shallow reservoirs, in which the flow is the primary control on sedimentation and the flow patterns can be very useful to predict preferential regions of deposition [4]. Dufresne et al. showed that particle tracking is relevant for predicting solid separation in combined sewer overflow chamber and suggested that a great attention must be given to the bed boundary condition [5]. Adamsson et al. turned out that for high flow rates both CFD simulations and measurements are applicable to investigate the hydraulics in rectangular tank, but both methods are not suitable for low flow rates [6]. Adamsson et al. applied a boundary condition based on bed shear stress for storage tank sedimentation, which turned out that user-defined subroutine is necessary for satisfactory result [7]. Jansons et al. investigated the hydrodynamic characteristics of nine stormwater pond configurations, indicating that correct positioning of design elements can significantly enhance the hydraulic characteristics of a pond and the greatest length to width ratio result in the highest hydraulic efficiency [8]. Kantoush et al. presented a series of numerical simulation and laboratory experiments to investigate flow in a rectangular shallow basin, which show the initial flow pattern determine suspended sediment transport and deposition and the using of baffle can change the flow condition [9].

As it can be seen, numerical methods provide us a very efficient way to investigate the phenomenon in hydraulic. In recent years, intelligent methods have been widely applied in

* Corresponding Author:
E-mail address: abdelali.terfous@insa-strasbourg.fr

Received: 20 November 2019; Accepted: 28 December 2019

many industrial fileds and excellent results have been obtained [10- 11].

2. Numerical Simulation Methods

2.1. Geometry Model

The model used in this paper is derived from the experiment devices. With the use of baffle the height of the cavity can be adjusted to 0 mm, 25 mm and 50 mm. The length of the storm tank is 4240 mm, the width of the storm tank is 760 mm and the height of the storm tank is 405 mm. The storm tank contains one inlet, two outlets and one free-surface. The upper outlet is only available when the mass flow is high enough to make the height of the water surface reach the position of the upper outlet.

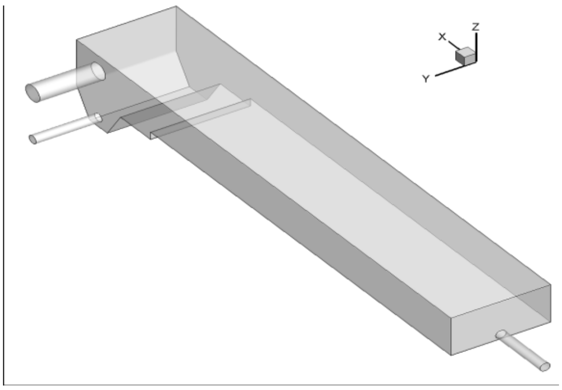

Figure 1. Geometry of tank without cavity

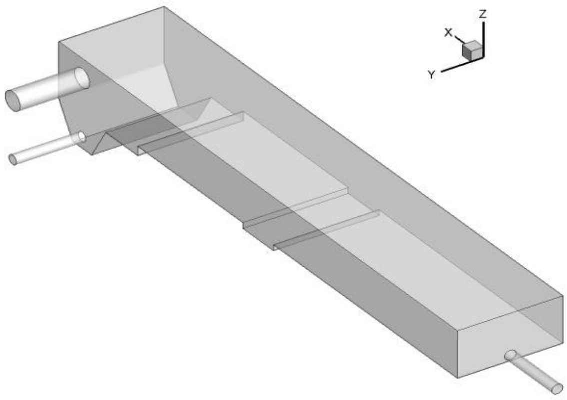

Figure 2. Geometry of tank with cavity

2.2. Mesh Information

For discretization of the computational domain, the commercial software ANSYS ICEM was adopted to create the mesh. According to the problem created by the sharp area between inlet and wall, two solutions were come up. The first is to change the geometry, which has to increase the height of the inlet pipe to diminish the sharp area. And the second is to use mixture mesh, which means in the minimum area containing the sharp area tetrahedral mesh will be adopted and in the rest part of the geometry hexahedral mesh will be adopted. Both ways can provide mesh with higher quality and less number which means the cost of computational resources could be saved and the convergence of the calculation could be assured. The quality of mesh used in the simulation is up to 0.578, the angle of mesh is at the range of 35.37 to 90 degree, the aspect ratio of mesh is at the range of 1.01 to 5.0 and the number of mesh is about 1 million cells.

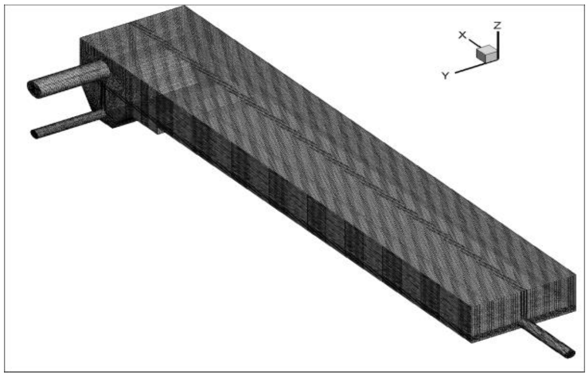

Figure 3. Mesh of tank without cavity

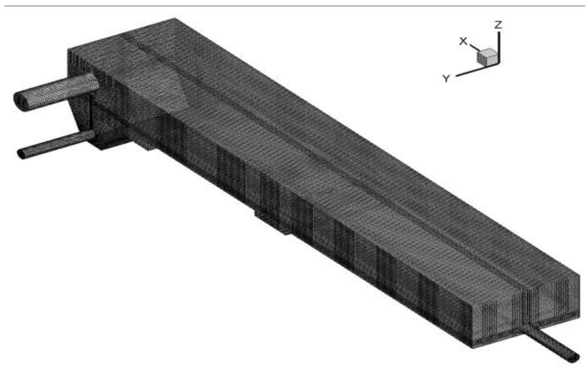

Figure 4. Mesh of tank with cavity

2.3. Boundary Conditions

As a very important part of numerical simulation, the boundary conditions must be taken into consideration. In order to simulate a hydraulic problem, there are four principle boundary conditions to be chosen: inlet, outlet, wall and symmetry.

For the inlet part, the mass flow was already known, the velocity inlet was chosen. The value of the inlet velocity should be provided, generally the uniform velocity was chosen which is not very the same as the real physical condition, but if the length of the inlet part is long enough to make the flow can be fully developed, the uniform velocity condition can be feasible.

For the outlet part, usually there are several options to be selected. However in this paper the objective storm tank contains two different outlet and in FLUENT code different outlet conditions could not be used in one calculation simultaneously, and the main outlet is connected to another tank by the pipe which means the pressure of the main outlet could be adjusted through changing the height of water in the second tank and the pressure at the main outlet is already known. Compared with all the other outlet conditions the pressure-outlet should be the most suitable condition.

For the wall part, the flow at the area near the wall could be very complicated because of viscosity. There are two ways to solve the problem, one is using wall function approach and the other is near-wall model approach. In this paper, the second solution will be adopted. And the most important thing of this solution is to determine the value of the height of the first cell, which is a function of dimensions of the geometry and the mean velocity of the flow in the tank, by the mean time Y+ in the range of 30 to 500 should be taken into consideration. Except for considering the treatment of near-wall mesh, all the walls could be set as standard wall.

For the free surface, because there is no flow traversing through the interface and the variables inside and outside the domains are equal to each other. In Fluent codes, the symmetry boundary is always used to simulate the free surface in reservoir. The principle advantage of this method is that it's not necessary to model two-phase, which means the cost of computational time could be decreased.

Due to the existence of free surface, the interface between water and air should be determined. As multiphase simulation is involved, and there are several multiphase model exited in Fluent codes, including volume of fluid model (VOF), discrete phase model (DPM), mixture model and eulerian multiphase flow model. Because the objective of model multiphase is to track the position of the interface between water and air, and VOF is mainly used for stratified flows which is very suitable for the condition. In order to calculate the deposition of sediment, DPM should be added to the simulation.

3. Numerical Results

All the resolution of the calculation was selected as second order upwind, and in order to obtain the interface of water and air in the tank, modified HRIC model was chosen to calculate volume fraction. As all the calculations were type of pressure based, PISO scheme was applied for accurate result.

3.1. Volume Fraction

As it can be seen from the contour of water volume fraction, the fluid is water where the value of water volume fraction equals to 1. And the fluid is air where the value of water volume fraction equals to 0. As to the position where the value of water volume fraction is in the range of 0 to 1, it means the fluid is mixture of water and air.

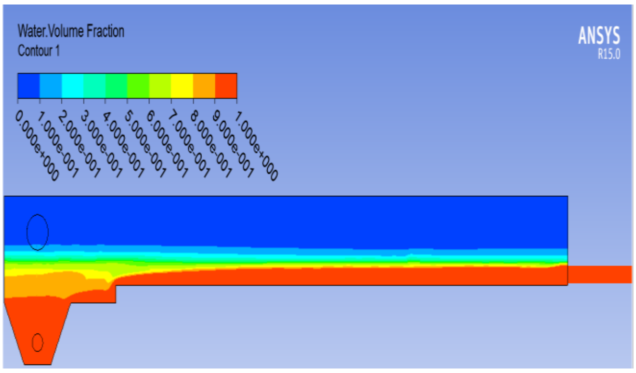

Figure 5. Water volume fraction in the tank without cavity Y=0.05m

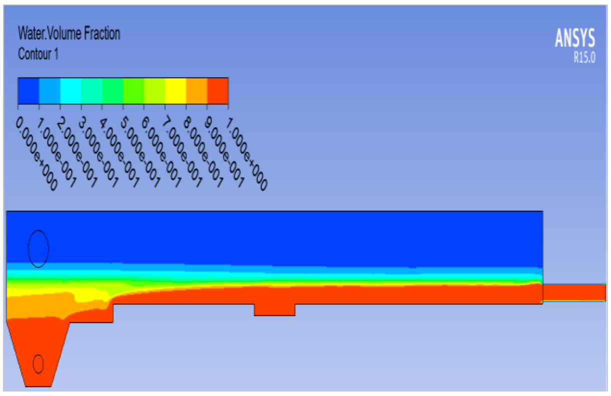

Figure 6. Water volume fraction in the tank with cavity Y=0.05m

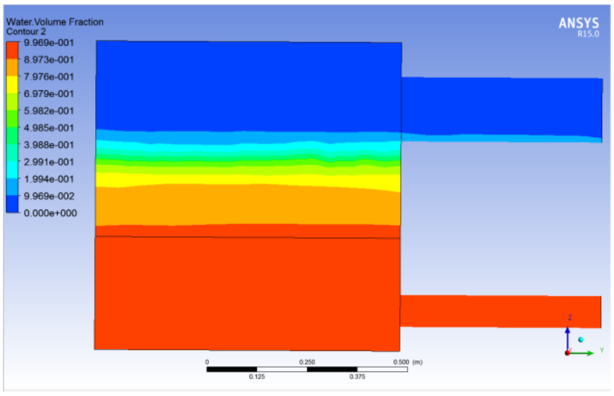

Figure 7. Water volume fraction in the tank without cavity X=3.995m

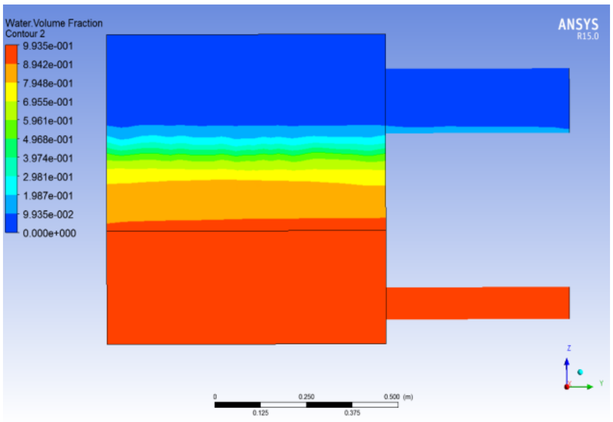

Figure 8. Water volume fraction in the tank with cavity X=3.995m

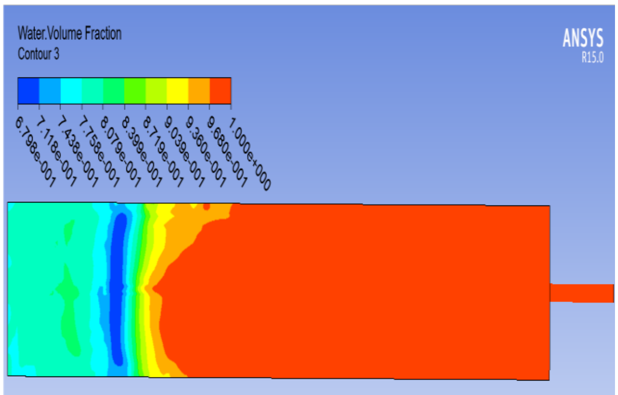

Figure 9. Water volume fraction in the tank without cavity Z=0.05m

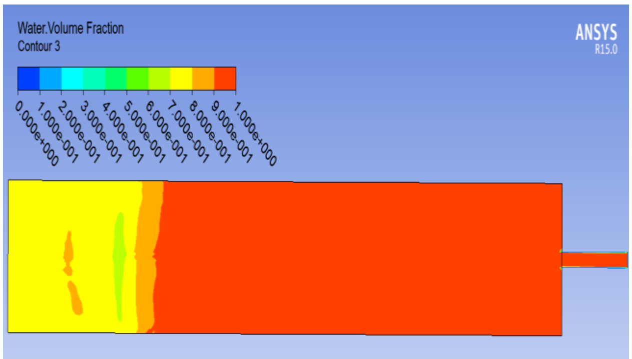

Figure 10. Water volume fraction in the tank with cavity Z=0.05m

Both of the contours of water volume fraction are derived from the calculation in which the inlet volume flow equals to 3 L/s. And from those figures the height of the water in the tank could be obtained which is 8 to 18 cm. In this condition, there is part of the water going out through the second outlet, which is mainly designed for overflow.

From Figure 5 to Figure 8, the water volume fraction is not very different from each other which means the cavity has little effect on the height of water in the tank. From Figure 9 and Figure 10, the distribution of water volume fraction along X in the tank with cavity is more uniform than that of the tank without cavity, which means the cavity may affect the uniformity of the flow in the tank.

3.2 Flow Patterns

Under the circumstance in this paper, the flow is asymmetric, in Figures 13 and 14, it can be seen that two eddy has formed in the flow field, and the size of the two eddy is not equal to each other.

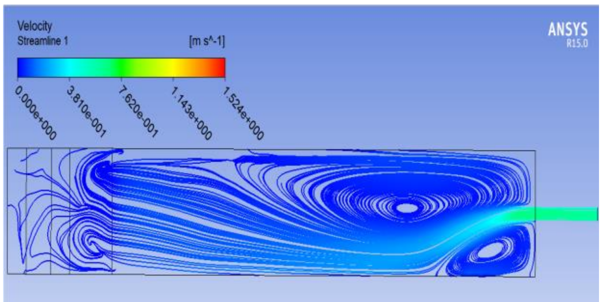

Figure 13. Streamlines in the tank without cavity Z=0.05m

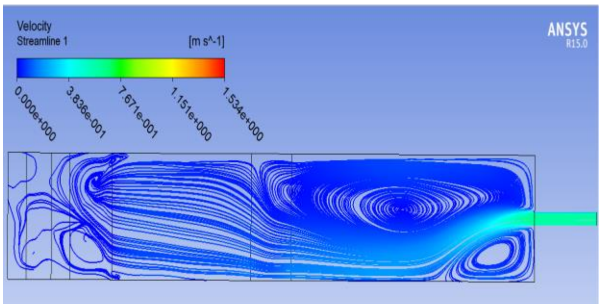

Figure 14. Streamlines in the tank with cavity Z=0.05m

As it can be seen, the flow conditions in two calculations is not very the same to each other, the size of the big eddy is decreased due to the effect of the cavity. However, the location of the cavity along X is too latter to make significant change of the flow conditions. Usually, the location of the cavity should be positioned at the location before the flow condition has fully developed, then the flow conditions might be altered significantly.

3.3 Particle Tracking

Investigation of movement of particle in this paper is the principal objective. In this paper, diameter of the particle added to the flow is at the range of 0.35 to 1.4 mm, and the average diameter is 0.738mm.

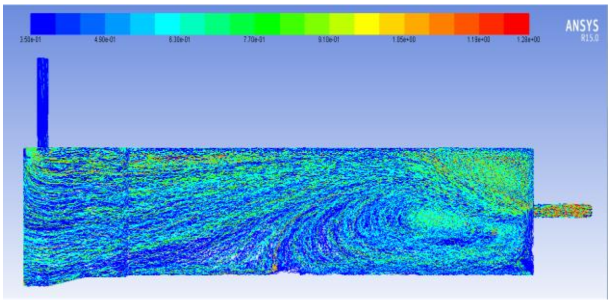

Figure 15. Particle tracking in the tank without cavity

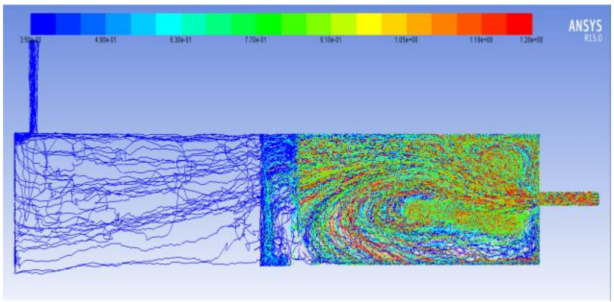

Figure 16. Particle tracking in the tank with cavity

With the existence of the cavity, the portion of sediment move to downstream is reduced significantly, which indicate the cavity do have notable influence to the movement of sediment in the tank.

4. Conclusion

Firstly, the height of the water in the tank is controlled by inlet mass flow and outlet discharge. As the inlet was set as velocity inlet, the mass flow was determined by the value of velocity at the inlet. And outlet were set as pressure outlet, the discharge was determined by the value of pressure at the outlet. In the real physical model, the main outlet was connected to a second tank by the pipe, which meant the pressure at the main outlet was a function of the height difference between the height of the main outlet and the height of water in the second tank.

Secondly, the flow pattern in the tank is usually dominated by length-width ratio, inlet mass flow and outlet discharge. However, under the circumstance that length-width ratio is not changeable, then inlet mass flow and outlet discharge should be the principal cause to the flow patterns. Also the cavity may change the state of eddy which can also affect the flow conditions, though the change is not significant under the circumstance in this paper, it can be taken into consideration to investigate the effect on the flow, for example, the location of the cavity positioned along the length direction of the tank and the height of the cavity.

Thirdly, the mesh used for calculation is very important to the results. The quality of the mesh is critical to the convergence of the calculation. And the number of the mesh is very important to the accuracy of the result. There would be a significant difference on the mass conservation for excessively coarse mesh. In the validation of the mesh, the net of mass flow at inlet and outlet reached to 2.7kg/s for using coarse mesh. Also the refinement of the mesh in the area near the wall is very important to simulate the flow conditions.

Fourthly, the application of the cavity is able to change the flow conditions only if the best location of the cavity could be decided. And also the existence of the cavity do have tremendous effect on the movement of sediment, the objective of using cavity for diminish the discharge of sediment is feasible.

All in all, numerical simulation is a practical method for investigating sediment transport in the tank, and definitely the validation of experiment should be necessary.

References

[1] A. Adamsson, L. Bergdahl, S. Lyngfelt, Measurement and three-dimensional simulation of flow in a rectangular detention tank, Urban Water 2 (2005) 277–287.

[2] A. Adamsson, V. R. Stovin, A. J. Saul, Bed Shear Stress Boundary Condition for Storage Tank Sedimentation, Journal of Environment Engineering 129 (2003) 651–658.

[3] M. Dufresne, B. J. Dewals, S. Erpicum, P. Archambeau, M. Pirotton, Classification of flow patterns in rectangular shallow reservoirs, Journal of Hydraulic Research 48 (2007) 197–204.

[4] M. Dufresne, J. Vazquez, A. Terfous, A. Ghenaim, J. B. Poulet, Experimental investigation and CFD modelling of flow, sedimentation, and solids separation in a combined sewer detention tank, Computers & Fluids 38 (2009) 1042–1049.

[5] M. Dufresne, J. Vazquez, A. Terfous, A. Ghenaim, J. B. Poulet, CFD modeling of solid separation in three combined sewer overflow chambers, Journal of Environment Engineering (2009).

[6] K. Jansons, S. Law, The Hydraulic Efficiency of Simple Stormwater Ponds, Rainwater and Urban Design (2007) 452-459.

[7] S.A. Kantoush, E. Bollaert, A. J. Schleiss, Experimental and numerical modelling of sedimentation in a rectangular shallow basin, International Journal of Sediment Research 23 (2008) 212–232.

[8] G. Lipeme Kouyi, J. Vazquez, J. B. Poulet, 3D free surface measurement and numerical modelling of flows in storm overflows, Flow Measurement and Instrumentation 14 (2003) 79–87.

[9] I. Kimura, S. Onda, T. Hosoda, Y. Shimizu, Computations of suspended sediment transport in a shallow side-cavity using depth-averaged 2D models with effects of secondary currents, Journal of Hydro-environment Research 4 (2010) 153–161.

[10] I. Bargegol, M. Nikookar, R. Vatani Nezafat, E. Jafarpour Lashkami, A.M. Roshandeh, Timing Optimization of Signalized Intersections Using Shockwave Theory by Genetic Algorithm, Computational Research Progress in Applied Science & Engineering 1 (2015) 160–167.

[11] A. Addeh, B. M. Maghsoudi, Control Chart Patterns Detection Using COA Based Trained MLP Neural Network and Shape Features, Computational Research Progress in Applied Science & Engineering 2 (2016) 5–8.