International Journal of Advanced Engineering and Management Research

Vol. 5, No. 02; 2020

ISSN: 2456-3676

CFD MODELLING OF PARTICLE SETTLING IN A SEDIMENTATION TANK

Dr. Marcell Knolmar
Assistant professor, PhD, Department of Sanitary and Environmental Engineering, Budapest University of Technology and Economics, 1111 Budapest, Muegyetem rkp.

ABSTRACT

The examined sedimentation tank is operating at a large industrial plant as the first phase of industrial water treatment. After several years of operation some problems were emerged like not planned sedimentation and capacity limits. CFD computations were executed in order to analyse the hydraulic flow conditions. Particle tracking simulations proofed the anticipated source of problems. Recommendations could be formulated based on the CFD simulations with different loads.

Keywords: industrial water supply; hydrodynamics; simulation; CFD; Fluent

1. Introduction

The industrial water demand is special both in quality and quantity terms. The continuous and reliable operation of the water treatment plant is crucial for the production process and accidently for firefighting. The supplied water should satisfy the quality demands of the industrial production. The sedimentation in water transportation pipes can decrease the capacity of the pipes, which has energy consumption and safety consequences.

The examined sedimentation tank is part of a large industrial plant. The water required for the industrial processes is sourcing from a natural surface water body. The main pollutant of the of the river water is its sediment content. The first treatment step of water treatment is sedimentation in 4 parallel tanks.

The signs of the problems emerged mainly after the sedimentation step, in the transporting pipes. The capacity of the pipes was recently decreasing because the sedimentation on the inner surface of the pipes was reducing their cross sectional area. Capacity problems were appearing even in the sedimentation tanks. In some unwanted spaces of the tanks the sediment was accumulating despite of the regular sediment removal by sludge pumps. From some spaces the removal of the settled and collected material is only possible by first emptying the tanks and then cleaning their bottom manually. For some years the designed scrapers were drawing out continuously the collected materials from the bottom, but recently the scrapers went broken.

The hydraulic conditions after the changed operation have been changed from the originally planned ones. The disadvantageous effects have been anticipated, and ANSYS Fluent Computational Fluid Dynamics [1],[3] simulations were decided to analyse the flow in the tanks in detail.

Vol. 5, No. 02; 2020

ISSN: 2456-3676

2. MATERIALS AND METHODS

For the hydraulic and particle settling analysis we built the detailed hydraulic model of the tanks. The size and shape of the 4 parallel tanks are the same, therefore during the simulation we modelled only one of them. The sludge hoppers at the end of the sedimentation tanks were assumed much smaller than on the original plans, because the hard sedimented parts were impossible to remove by the sludge pumps.

2.1 Previous studies

The performance of sedimentation units could be evaluated by Fluent 3D simulations [4]. The adaptation of measurements in tanks could be improved by fluid flow calculations [5]. Computational fluid dynamics investigations were done for shallow settling tanks to determine the optimal geometry [6]. The settling conditions in sedimentation tanks with different inflow and baffles were successfully modelled by CFD software [7], [8], [9].

2.2 Structural model of the tank

As the first model step we had to build the structural model of the treatment unit. Pumps are forcing the water into an entry chamber through a Ø1m sized pipe (Figure 1). Then 8 buffles are distributing the flow into the main tank.

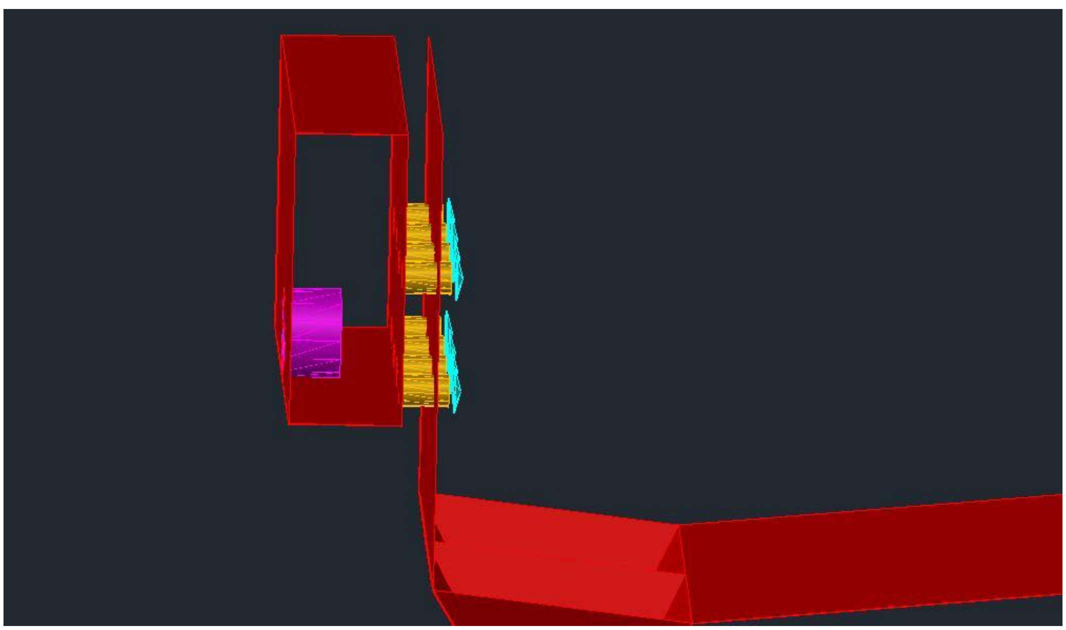


Figure 1: Inflow pipe (purple), entry chamber(red), distributing buffles (yellow and blue) and sediment hoppers (red)

The main part of the treatment unit is a rectangular, longitudinal flow type sedimentation tank. The tank length is 30m and its width is 6m (Figure 2). Over 3 weirs at the lower end of the tank the treated water is flowing out.

ISSN: 2456-3676

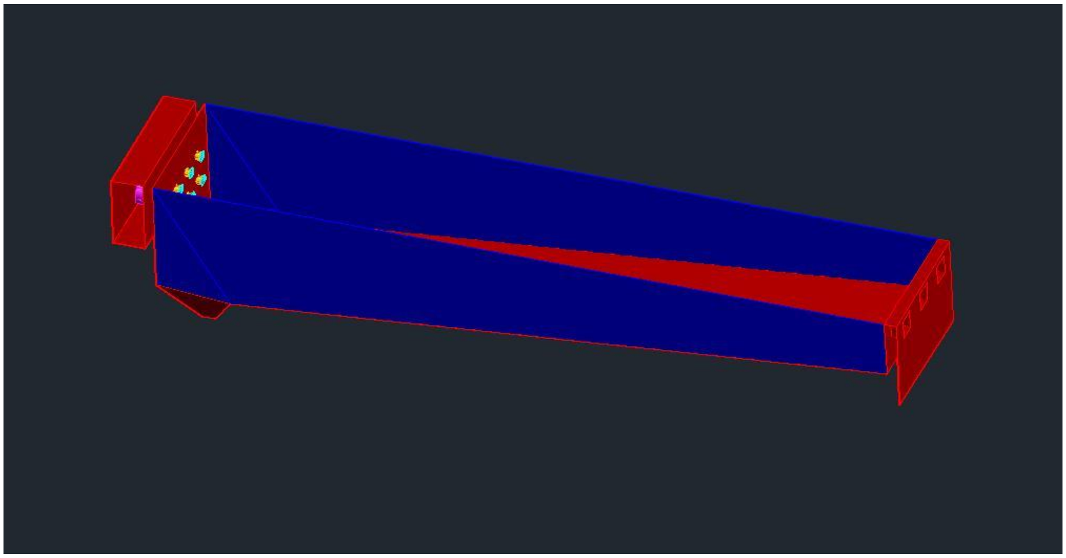


Figure 2: Structure of the sedimentation tank

2.1 Volumetric model of the water body

From the built structure model of the tank, the CFD simulation could have computed the water surface. Because the sedimentation unit is working about permanently, the water surface levels were better to be measured. Instead of the long simulation of filling up the tank and reaching the steady state, we could draw the surface levels from the measured data. The water body first created from AutoCAD 3D faces [2], then we converted to mass element objects. The dissolved solid volume was created by the "combine" operation of AutoCAD from the distinct mass elements (Figure 3).

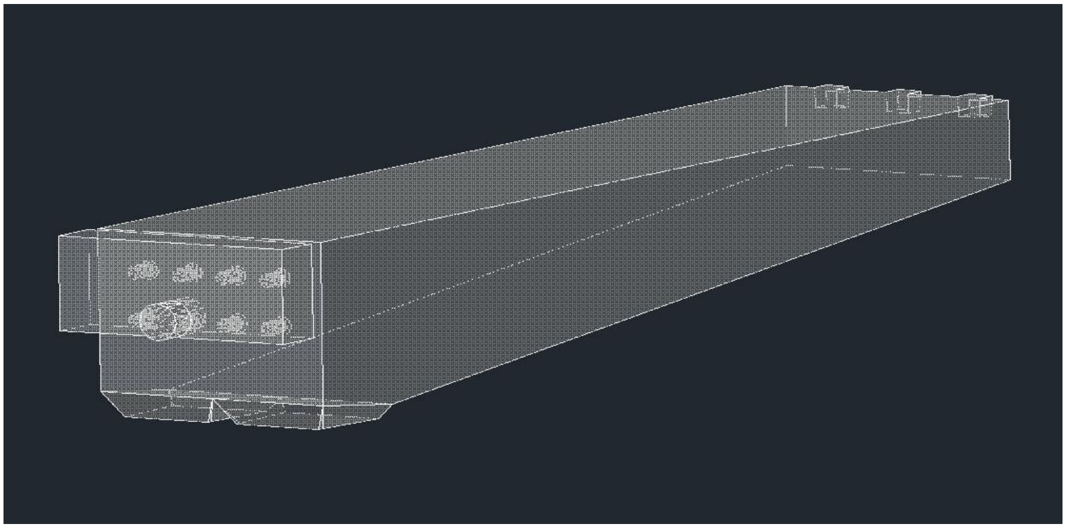


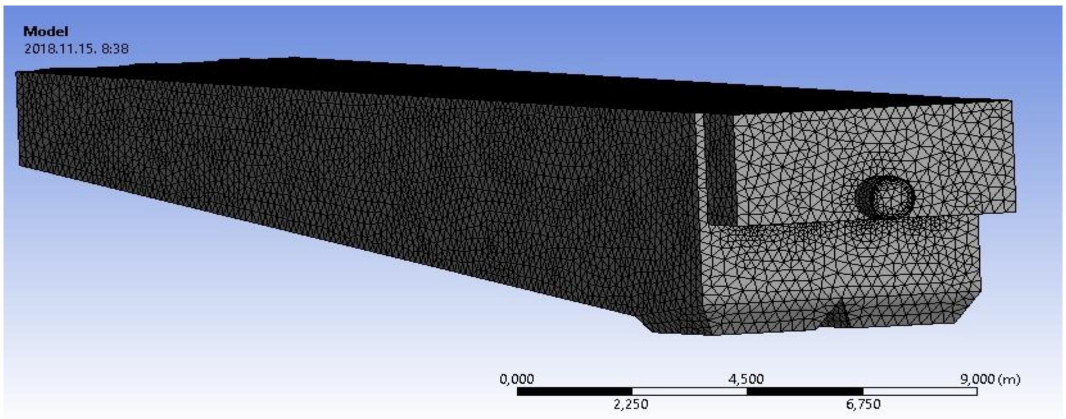
Figure 3: Water body

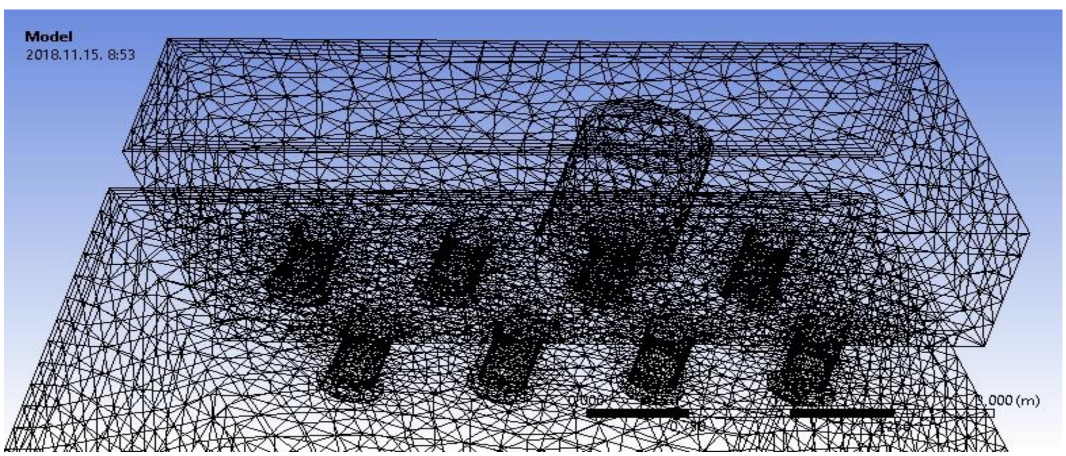
Vol. 5, No. 02; 2020

ISSN: 2456-3676

2.3 Meshed model

The AutoCAD generated solid is convertible to the Fluent CFD using the IGES drawing exchange format. The mesh generation is necessary for finite-volumes computations. The mesh was generated by the Fluent built-in mesh generator (Figure 4). The maximum face size of the tetrahedral cells was set to 0.25m, but at special spaces, i.e. at the inlets and outlets, the meshing was set denser, 0.05m. Automatic inflation was set to smooth the transition edges (Figure 5). In the final meshed model 140 000 tetrahedral cells were generated. The boundaries also had to be defined for the further model steps.





2.4 Fluid flow model

For the hydrodynamic simulations we applied the ANSYS Fluent VOF (Volume of Fluid) model. The boundary conditions were defined at the previously defined boundaries:

- velocity at the inflow pipe
- fix levels at the outflow weirs
- walls at the structure walls
- free water surface at the top of the main tank and the entry chamber

ISSN: 2456-3676

The initial longitudinal velocity in the main tank was set to 0.01m/s. The fluid was assumed incompressible and of constant density independently from the pressure. The applied turbulence model was the hybrid k-ε, which can produce the best reliable results inside the fluids.

The flow computations only slightly changed the surface levels, because the applied measured values as initial values were close to the simulation results.

In order to simulate the amount of the settled sediment, the Discrete Phase Model (DPM) calculation was also set. At the inflow pipe an injection was given. The density of the inflowing particles is 2700 kg/m^3. The inflow rate of injected particles is about 0.001 kg/s, regarding the concentration in the incoming surface water 15 mg/l. 10 diameter fractions were automatically generated by Fluent with diameter size from Ø0.00002 m to Ø0.00005 m. The movement of the particles at the boundaries were set differently: they are reflecting from the sidewalls of the tank, but trapped at the bottom of the tank and the hoppers.

3. RESULTS

Based on the described methodology, we could run the simulations. After about 25000-30000 iteration steps the quasi-steady flow conditions were reached and the calculated hydraulic parameters were accepted. We introduce the detailed results regarding the sedimentation tank.

3.1 Hydraulic behaviour

The Fluent simulations are showing the velocity distribution in each calculation elements. The longitudinal profile in the middle of the tank is showing quite even distribution (Figure 6). The flow computations only slightly changed the initial velocities (0.1m/s) in the majority of the water body. In the inflow pipe the velocity is higher with 1 magnitude.

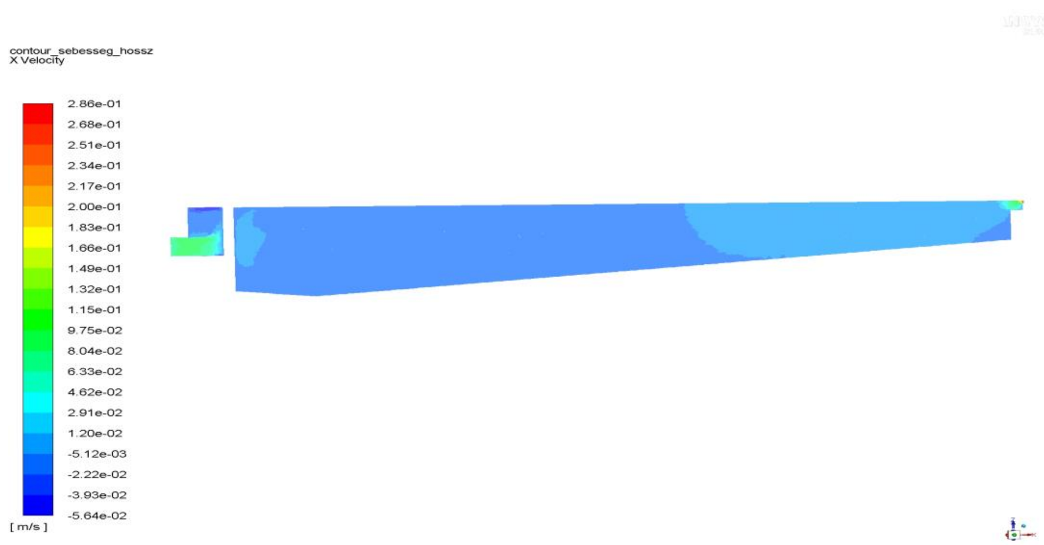


Figure 6: Velocity profile of the sedimentation tank

The velocity profile at the inflow is showing well mitigated and distributed flow in the entry chamber, and even more at the 8 buffles (Figure 7).

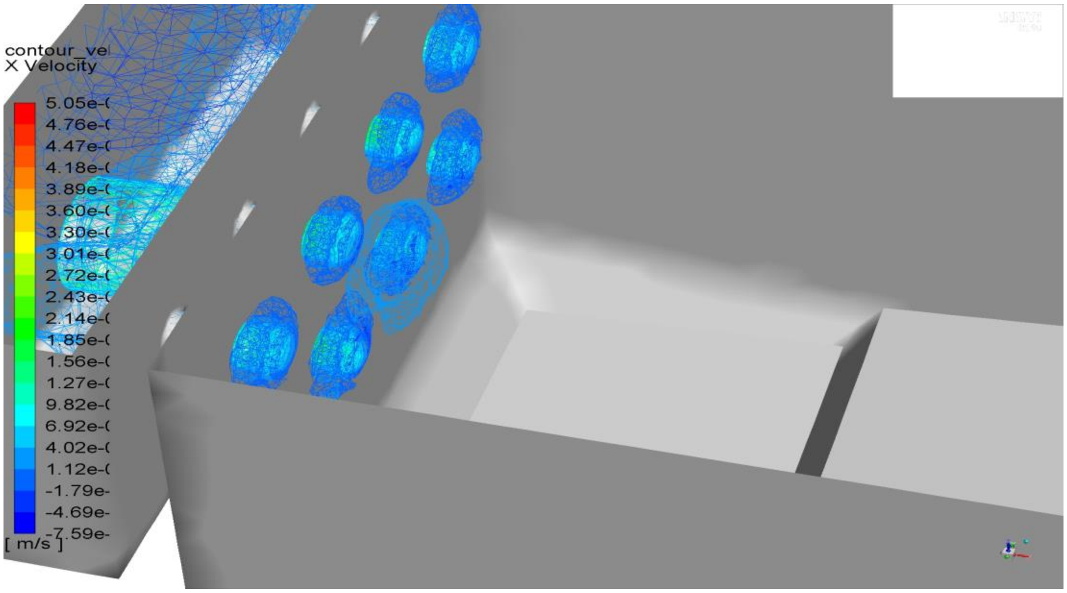


Figure 7: Velocity distribution at the inflow

In the middle of the sedimentation tank the velocity distribution is becoming quite even because of the mitigation and distribution effect of the 8 buffles (Figure 8).

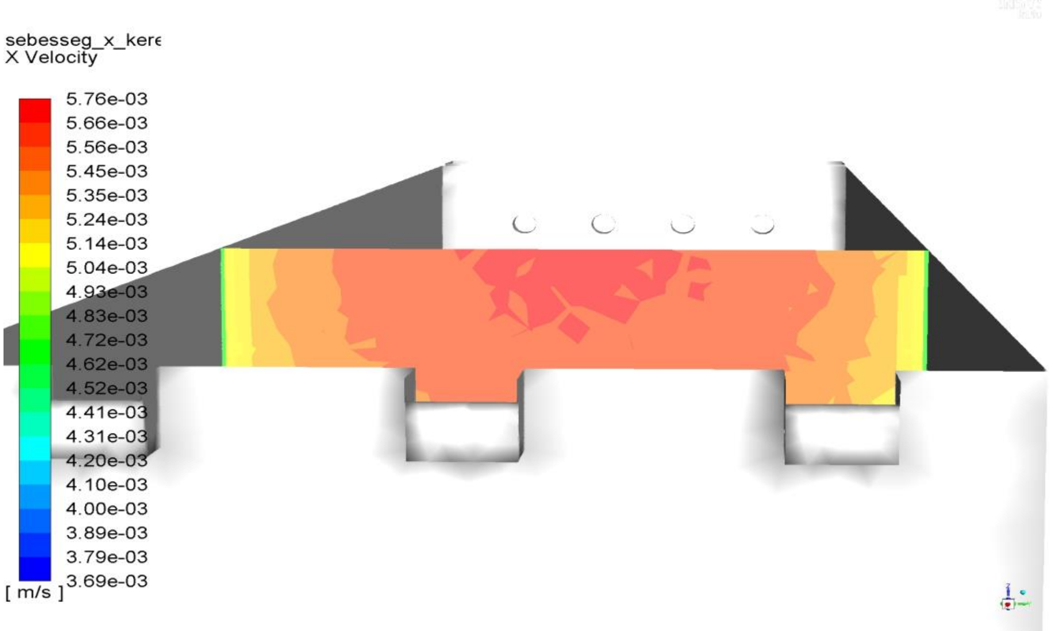


Figure 8: Cross sectional velocity distribution in the middle of the tank

Vol. 5, No. 02; 2020

ISSN: 2456-3676

The water is flowing with higher speed over the outflow weirs at the lower end of the tank (Figure 9.). The velocity has about the same magnitude as at the inflow.

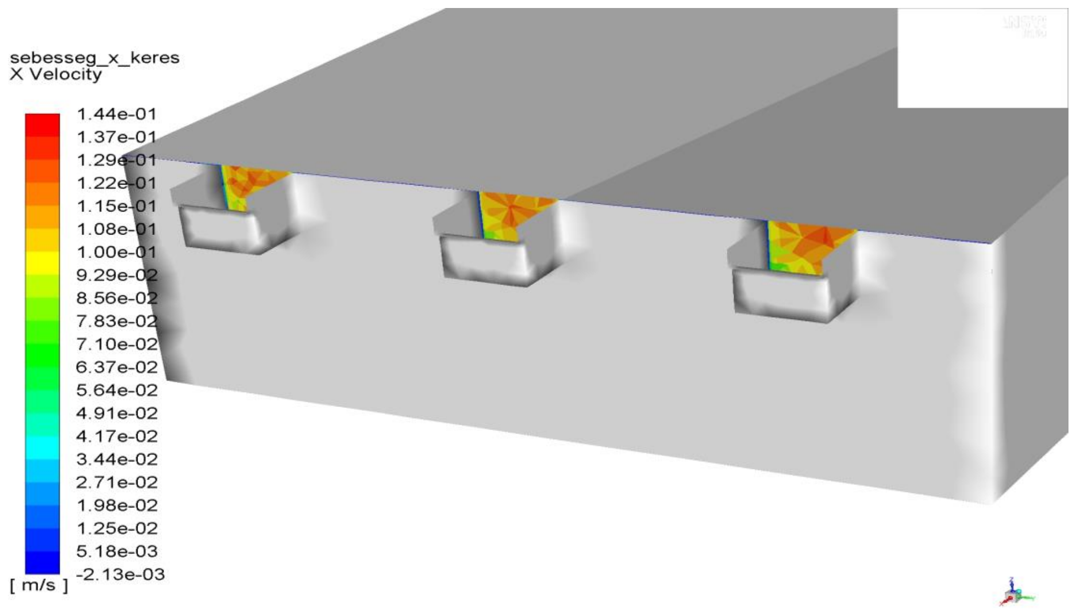


Figure 9: Cross sectional velocity distribution at the outflow of the tank

3.1 Particle tracking

After completing the hydrodynamic simulation, the discrete phase simulation was also possible to execute. The whole route of about 1300 particles generated from the inlet to the outlet (Figure 10). The routes are quite evenly distributed. Continuously moving toward the end of the outlets and also settling to the bottom. Only a few percent are escaping at the outlet weirs.

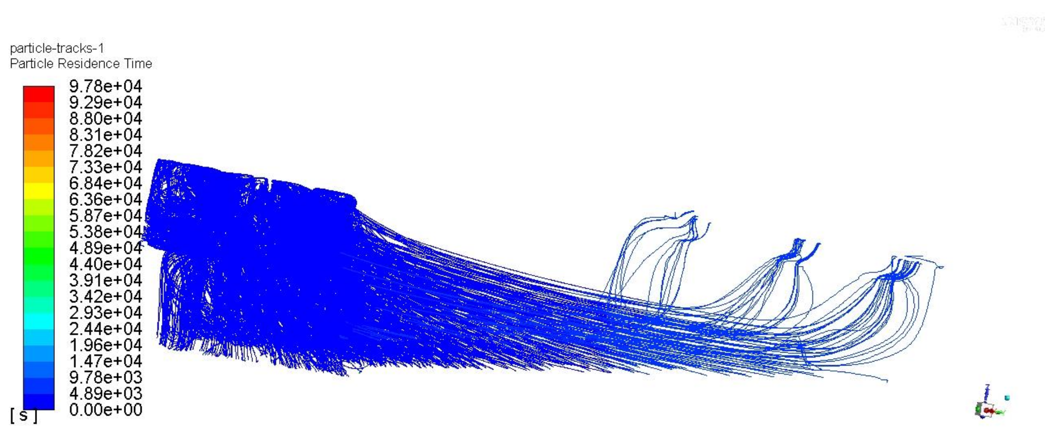


Figure 10: Particle routing from inlet to outlets

The inlet of the tank was designed carefully, the particles are not circulating long at the buffles and going about parallel toward the end of the sedimentation tank (Figure 11).

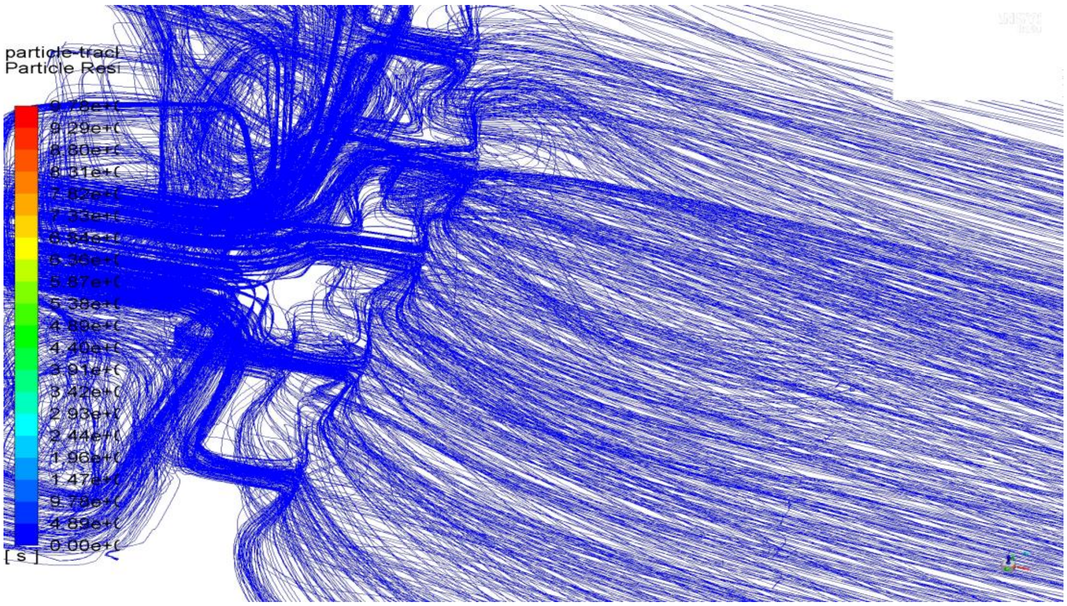


Figure 11: Particle routing at the inflow of the tank

4. Discussion

Based on the CFD simulations the velocity distribution of the flow is favourable for settling of small particles. The settling in the sedimentation tank is satisfactory, corresponding to the planned state.

At the designed hydraulic load, the majority (97-98%) of the sediment is settled on the bottom. In the model we assumed the particles reaching the bottom to be trapped, while under real operation conditions they can enter again into the water body increasing the possibility of their outflow. The continuous operation of scrapers could provide the trapping of the settled particles in the hoppers accordingly to the simulation model.

Series of computations were executed with increased hydraulic load. The load has significant effect, e.g. at 50% increased hydraulic load, the volume of the effluent sediment is increasing by 200%. The efficiency of the settling could be improved by rebuilding the outflow weirs. Increasing the cross sectional area when flowing over the weirs would provide less velocity. Less outflowing sediments and more settled sediments are anticipated on the bottom. The estimation of the settling efficiency could be done by new model computations.

References

ANSYS Inc., Ansys Fluent Manual ver 19. ANSYS Inc., 2018.

Vol. 5, No. 02; 2020

ISSN: 2456-3676

Autodesk. (2018). AutoCAD 2019. User's Guide.

Fluent '19, User's Theory Guide, 2019.

T. Karches, D. Szélpál, Verification of Horizontal Flow Grit Chamber Performance at Various Load Conditions. International Journal of Latest Research in Engineering and Technology. Vol 3. No. 08. pp. 44-47., 2017.

E. Madarász, M. Patziger, Improving and Adaptation of an In Situ Measurement Technic in Aerated Tanks. 10th Eastern European IWA Young Water Professionals Conference, Zagrab, Croatia, International Water Association (IWA), 2018.

M. Patziger, Computational fluid dynamics investigation of shallow circular secondary settling tanks: Inlet geometry and performance indicators, Chemical Engineering Research and Design, 112., 122-131., 2016.

M. Shahrokhi, F. Rostami, M.A. Md Said, S. Syafalni, Numerical Investigation of Baffle Effect on the Flow in a Rectangular Primary Sedimentation Tank, World Academy of Science, Engineering and Technology, International Journal of Environmental and Ecological Engineering, Vol:5, No:10, 2011.

M. Shahrokhi, F. Rostami, M. A. Md Said, Syafalni, Numerical simulation of influence of inlet configuration on flow pattern in primary rectangular sedimentation tanks, World Applied Sciences Journal 15 (7), 1024-1031, 2011.

A. Zhang, CFD Modeling and Optimization of Primary sedimentation tank, Degree Project, Environmental Engineering and Sustainable Infrastructure, Hydraulic Engineering, Master of Science, Stockholm, 2017.

Author Profile

Dr. Marcell Knolmar was graduated at the Budapest University of Technology as civil engineer and has been working for the same university at the Department of Sanitary and Environmental Engineering as researcher and teacher. He has significant results on the fields of computer aided sewer design, geographical information system of sewer networks and designing of rain monitor device. He certificated his PhD degree in the field of the computer aided sewer design. His current research scope is the hydraulic modelling of sewer networks, specially the sediment transport of sewer networks.