International Journal of Latest Research in Engineering and Technology (IJLRET)
ISSN: 2454-5031
www.ijlret.com || Volume 06 - Issue 09 || September 2020 || PP. 49-54

CFD Modeling of a Sedimentation Tank

Marcell Knolmar[1]
[1](*Department of Sanitary and Environmental Engineering,*
Budapest University of Technology and Economics, Hungary)

Abstract: An industrial sedimentation tank was selected for the examination. The tank is suffering from floating problems. We applied computational fluid dynamics (CFD) simulations to explore the weak points. The 3D simulations were assuring the anticipated strengths and weaknesses. The shared experiences are including practical and theoretical knowledges about the model building and flow simulation tasks. Evaluation and suggestions are also included.

Keywords: 3D simulation, CFD, Fluent, sedimentation tank

I. INTRODUCTION

The efficient operation of a sedimentation tank is determined by its flow conditions. The flow conditions of each sedimentation tank are defined by the specific geometrical shape of the tank, inlets, outlets, inside walls, baffles and diffusers. Calculation of flow conditions especially the velocity distribution and sedimentation is the key task in the designing or redesigning phase. Computational fluid dynamics (CFD) is offering comprehensive and efficient tools for the simulation of fluid flows.

Sedimentation tanks are applied both in water and wastewater treatment processes. In the paper we introduce the results regarding a sedimentation tank operated in industrial water treatment process. The tank has been operating for several years and has some floating problems. In the past years the operation company executed several practical trials aiming to improve the efficiency.

The purpose of the CFD simulations described in the paper was to highlight the weak points of the sedimentation process in the tank. Some of the poor flow conditions were already observed or at least suspected. Nevertheless comprehensive 3D CFD simulations were executed to prove the anticipations and to designate all the critical points of the tank. Additional purpose of the paper is to disseminate the solutions we found during the whole modeling process.

In the paper we describe the model building and flow simulation steps. We also evaluate the results and suggest further improvements of the treatment process and the floating problems.

II. PROBLEM RAISE

The tank focused in the paper is treating water for an industrial plant. The main purpose of the sedimentation tank is to provide proper flow conditions for the settling of different solid particles. The longitudinal tanks are providing more or less uniform laminar flow conditions. The small solid particles having higher density than water, are sinking to the bottom. The examined tank is longitudinal and the scraper is missing. The lighter materials floated to the surface can be skimmed periodically, in our cases manually.

The examined tank is part of the supplementary treatment plant of an industrial plant. The supplementary treatment is supplying the necessary volume of water substituting the evaporated volume during the industrial cooling processes. The cooling water coming from the industrial works is flowing first into the focused sedimentation tank. The volume of incoming solid materials are much more less than usual, because of the closed recirculation of the incoming cooling water. There is neither scraper nor sludge hopper, the sludge is removed from the emptied tank by high pressure water manually. The main problem observed by the operators is the materials accumulating on the surface. This organic floating "islands" are growing and meaning additional load to the incoming materials (Fig 1.). The skimming of the floating parts is consuming additional, periodic task of the operators. The target of the CFD simulations was to investigate for the hydraulically critical sites of the tank. The hydrodynamic model can also be used for further improvements of the removal process.

Fig 1. Organic floating "islands" on the surface at the corners

III. GOVERNING EQUATIONS

The Navier-Stokes equations are describing the velocity distribution in case of laminar flow. The conservation of momentum, based on the Newton's second law is:

$$\rho \frac{\partial \vec{v}}{\partial t} + \rho(\vec{v} \cdot \nabla)\vec{v} = -\nabla \mathrm{p} + \nabla \overline{\overline{\mathrm{T}}} + \rho \vec{g} + \vec{\mathrm{F}} \qquad (1)$$

where:

- ρ: density of the fluid
- $\vec{v}$: velocity of the fluid
- p: pressure in the fluid
- $\overline{\overline{\mathrm{T}}}$: stress tensor
- $\rho \vec{g}$: gravitational body force
- $\vec{\mathrm{F}}$: external body force (e.g. from dispersed phase)

Close to the inflow and outflow openings and at the walls, the flow is becoming turbulent, the velocity is fluctuating in a small scale. CFD simulations can apply different turbulent flow models [1] for different purposes at a reasonable computational speed. The k-ω turbulence method is giving accurate results close to the wall, while the k-ε model is better inside the fluid. The shear-stress transport (SST) k – ω model is applying the mixture of the k-ω and the k-ε turbulence models by a weighting function depending on the distance from the wall. The combined model also includes some refinements of the original k – ω model improving the accuracy and reliability. In the SST k – ω turbulence model the turbulent kinetic energy:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + \tilde{G}_k - Y_k + S_k \qquad (2)$$

And the turbulent dissipation:

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \varpi u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\varpi \frac{\partial \omega}{\partial x_j}\right) + \tilde{G}_\omega - Y_\omega + D_\varpi + S_\varpi \qquad (3)$$

where:

- $\tilde{G}_k$: turbulence kinetic energy due to mean velocity gradient
- $\tilde{G}_\omega$: generation of ω
- Γ_k: effective diffusivity of k
- Γ_ϖ: effective diffusivity of ω
- Y_k: dissipation of k due to turbulence
- Yω: dissipation of ω due to turbulence
- D_ω: cross-diffusion term
- S_k: user defined terms
- S_ω: user defined terms

IV. MATERIALS AND METHODS

There were born several papers about CFD simulations in the field of water treatment. CFD is applicable for the efficiency analysis e.g. of a grit chamber [2]. The further improvements can be supported by previous 3D simulations of a tank [3]. Similar CFD investigations regarding the effect of geometry were executed for sedimentation tanks [4].

The ANSYS Fluent CFD simulation software [1] is capable of 3D flow simulations. Water can be modelled as viscous fluid, thus the Navier-Stokes equations can be applied. The SST k – ω turbulence model embedded into Fluent can be used for simulate the critical, turbulent conditions of the tanks. The discrete phase model of Fluent can simulate the solid particle transport. We selected ANSYS 2020 R2 software package for the CFD simulations of the sedimentation tank.

Fluent is capable of modelling of 2 or 3-phase flow, but in case of the sedimentation tank the water-air surface can be set in the model quite accurate without long initial simulation. We could measure existing surface levels on-site and set them in the model. The water body inside the walls of the tank could be drawn by AutoDesk C3D software [5]. The solid of the water body created in C3D was imported into ANSYS SpaceClaim through the standard IGES exchange format. The meshing of the model was prepared in ANSYS Meshing software in order to produce finite volumes at sufficient precision.

The simulations were executed by the ANSYS Fluent software. On Windows PC platform the simulations were lasting less than 60 minutes. The results could be view and report inside the ANSYS workbench. The 3D graphical capabilities of ANSYS software offer convenient and professional presentation of solutions.

V. RESULTS AND DISCUSSION

As the first step the solid of the water body inside the walls was built for the sedimentation tank (Fig 2.). The size of the water body is 46.5m x 17.2m x 3.35m, the shape is cuboid. The two inner walls are subtracted. The wall at the inflow is divided into five sections, the second and fourth sections are the inflow openings, the first, third and fifth sections are walls. A valve on the outflow pipe is closed at about 70 percent, controlling the water level in the tank. The outflow is modelled as 70 percent closed (wall) and 30 percent open circular segments.

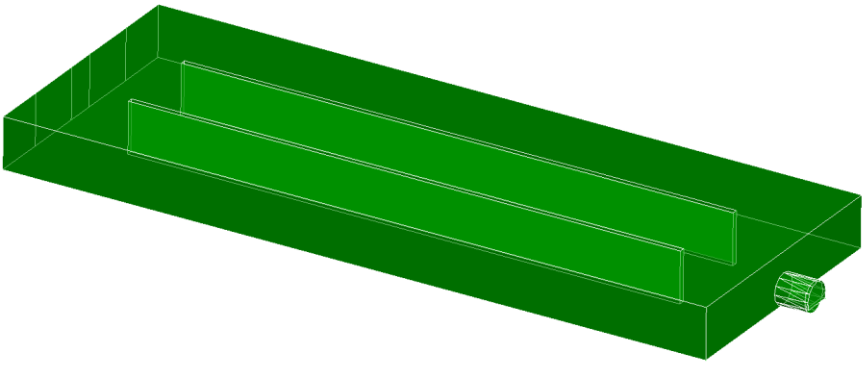

Fig 2. The solid of the water body

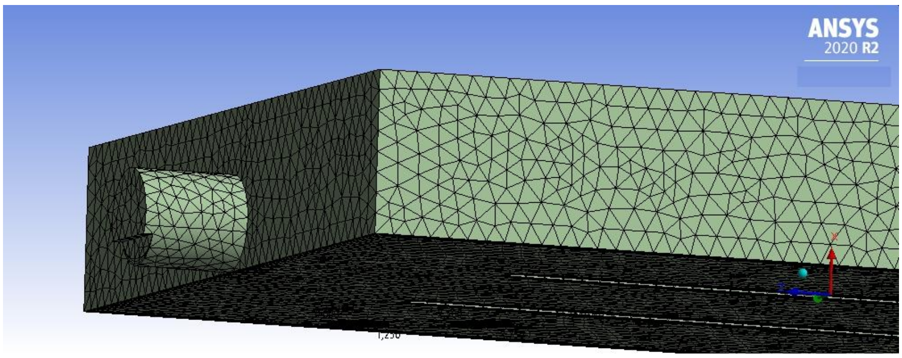

Fig 3. The meshing around the outflow

The meshing network needed for the calculation of the finite-volumes method was prepared by the ANSYS Meshing software (Fig 3.). The total count of the tetrahedral shape elements was 360950. The edge length was in the range 0.18-0.4m, which was sufficiently small for stable CFD calculations even at the inflow, outflow openings and close to the walls.

The simulation of the 3D flow was performed by the Fluent software. The boundary condition at the two rectangular inlets was set as a fixed velocity (0.207 m/s), calculated from the average hourly inflow and the area of the inlet openings. The boundary condition at the circular segment outflow was set as free outflow. The SST $k - \omega$ model was set as turbulence model. The temperature of the fluid was set to 30°C as initial operating condition.

The necessary time-step can be calculated based on the Courant-Fiedrich-Levy (CFL) number, which should be less than 1.0 in order to reach stability:

$$CFL=\frac{U*\Delta t}{\Delta x} \quad (4)$$

where:

- U: flow velocity
- Δt: time-step
- Δx: the minimum edge length of the mesh

The time-step calculated from the condition (4) was 0.87 s. The time step method was set as automatic, i.e. continuously changing. After 1989 time-steps the solution is converged, the quasi steady state flow condition was stabilized, further simulation steps were already not changing significantly the calculated velocitydistribution.

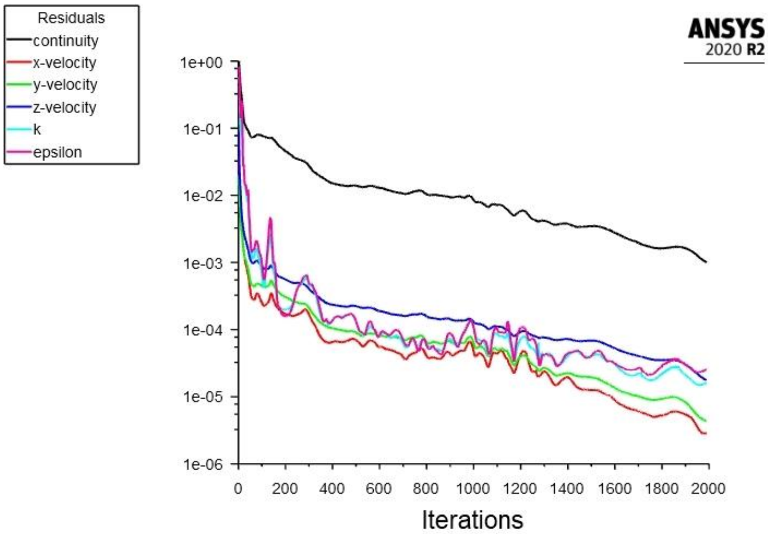

Fig 4. Convergence of the simulation

The flow simulations were targeting primary the velocity distribution. The velocity distribution on the surface (Fig 5.) and on the bottom (Fig 6.) is becoming continuously more uniform from inflow to outflow. The colour range is different in the two sections in order to show the velocity differences inside each sections. On the Fig 7. cross-sectional velocities are displayed in every ten meters and at the bottom. The uniformity change along the longitudinal sedimentation tank can be seen.

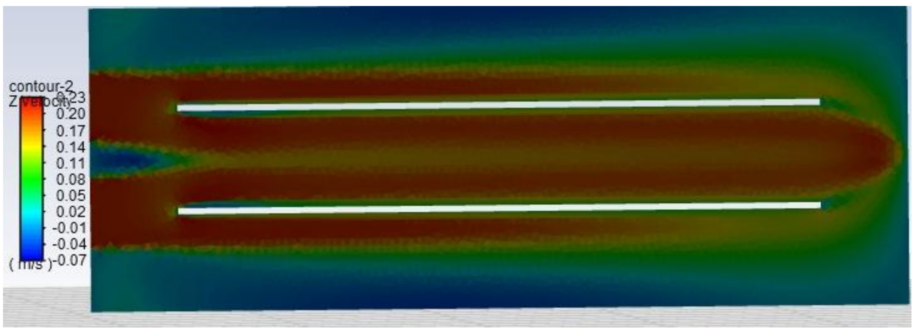

Fig 5. Velocity distribution of the water body on the surface

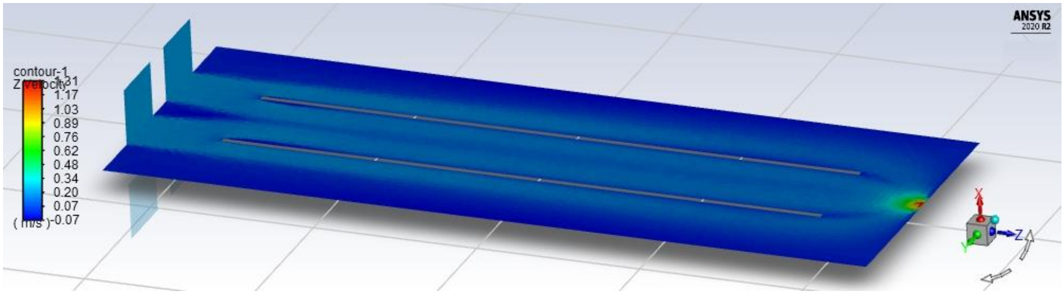

Fig 6. Velocity distribution on the bottom

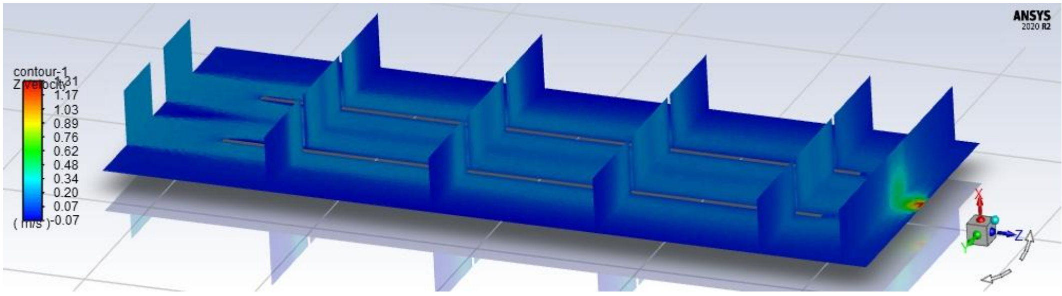

Fig 7. Velocity distribution on the bottom and in cross sections

The simulations calculated circular flows close to the corners, not only on the surface (Fig 8), but also in the whole depth (Fig 9 and Fig 10. The longitudinal sections were prepared at 0.5m distance from the side walls. The vector arrays can demonstrate both of the directions and the magnitude of the velocity.

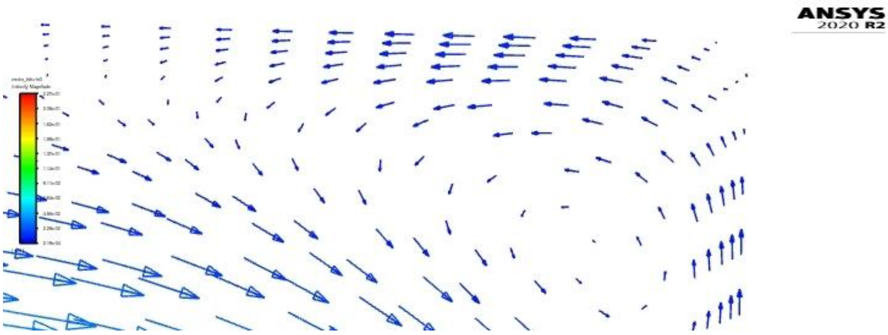

Fig 8. Circular flows near the corner on the surface

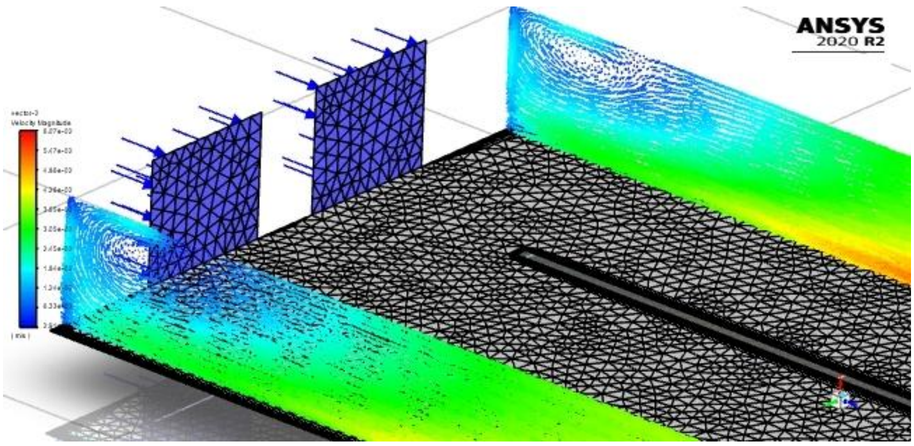

Fig 9. Circular flow close to the inlet

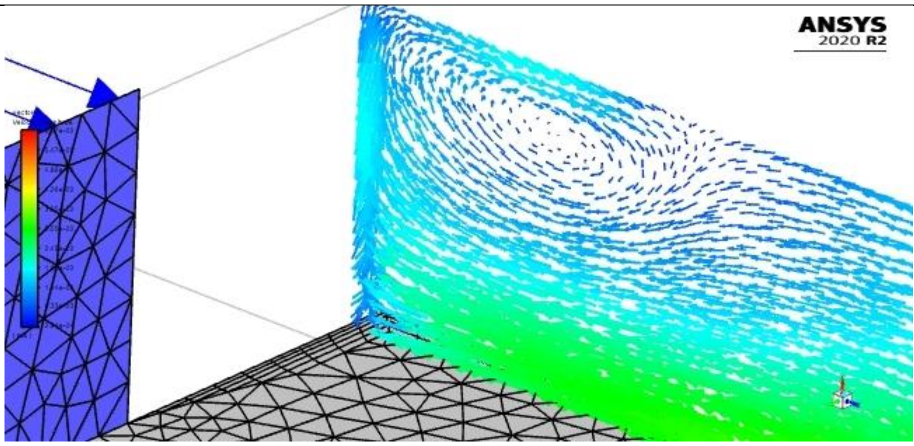

Fig 10. Circular flow close to the inlet, zoomed

VI. CONCLUSION

The model is simulating the present state realistic. The CFD simulations proved, that the sedimentation tank can force the inflow velocities relatively uniformly, therefore the primary purpose of the sedimentation tank i.e. the settling of the fine solids can be performed satisfactory.

However there are shortcomings of the flow distribution in tank. As from observations on the surface (Fig 1.) anticipated, close to the corners the flow is becoming circular. The circular flow is not endangering the main function of the sedimentation tank, but requires continuous operation measures, skimming procedures.

The improvement of different arrangement of the inner walls or installation of a scraper can be investigated by CFD simulations. The presented model can be developed for evaluation of the possible structural improvements of the sedimentation tank. Discrete phase model of Fluent can also be applied to verify the settling of solid particles.

VII. ACKNOWLEDGEMENTS

We thank our colleagues Miklós Patziger, Emese Madarász who provided insight and expertise of fluid flow modelling that greatly assisted the analysis.

REFERENCES

[1] ANSYS Inc, *Ansys Fluent Theory Guide ver 20.2*, 2019.

[2] T Karches, D Szélpál, Verification of Horizontal Flow Grit Chamber Performance at Various Load Conditions, *International Journal of Latest Research in Engineering and Technology*, 3(8), 2017, 44-47.

[3] 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, 2018, 335-341

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

[5] Autodesk, AutoCAD C3D 2019 Help, *http://help.autodesk.com/view/CIV3D/2019/ENU*, 2019.