CFD ANALYSIS OF WASTEWATER SEDIMENTATION TANK

Ing. Michal Holubec, PhD., Ing. Jaroslav Hrudka, PhD., Ing. Réka Csicsaiová, doc. Ing. Štefan Stanko, PhD.

ABSTRACT

Primary sedimentation tanks are structures of primary waste water treatment. Rectangular tanks are quite common in Slovakia although they are known to have some hydraulical problems resulting from their geometry. Furthermore, the inflow channel is almost exclusively oriented perpendicularly to the direction of the flow in these tanks. In this paper, we will show some of the problems caused by these factors, with the use of CFD simulations.

Key words: *CFD, sedimentation tank, wastewater*

1 INTRODUCTION

Present Slovakia waste water treatment plants (WWTP) treat the waste waters from about 60% of inhabitants, which are connected to the public sewer system. Many of the WWTP's use more than 40 years old equipment and need reconstruction. The hydraulic design of these reconstructed structures is mostly based on empirical or semi-empirical methods. Using CFD methods, we were able to analyse one such reconstructed structure. Very often we can find sub-optimal design, mostly on rectangular tanks, which means less effective operation of the treatment process and higher operational expenses. The analysis consisted of simulation using ANSYS CFD software supported by in situ velocity measurements.

Based on our research and experiences from WWTP operators in Slovakia we can declare that there are a lot of WWTP's which have hydraulic problems in their technological processes, even after reconstruction. This paper focuses on CFD analysis of a wastewater settling tank located at a WWTP in the municipality of Humenné, Eastern Slovakia.

The wastewater treatment plant is designed to process wastewater from 96700 equivalent residents and works in two stages – mechanical treatment and biological treatment. The wastewater is transported via a combined sewer system, with maximum inflow of 1050 l.s-1 (1922 l.s-1 during storm events). Storm water tanks and the storm water treatment process line are in place to deal with storm water inflow. This analysis focuses on the primary settling tank at Humenné WWTP. It is a rectangular tank with horizontal flow. The wastewater is transported from the grit chambers through a concrete conduit, perpendicular to the flow direction in the tank. The water flows from the inflow chamber through ten T-shaped steel pipes DN300, which are located on the front wall of the tank, 1622 mm above the tank bed. There is also a small, 600 mm wide floodgate in the same wall. There are 4 holes for collecting the sludge at the bottom of the tank in the inlet zone. The tank is 36 meters long and 12 meters wide, the maximal height of water surface is 3,72 meters, according to the project documentation. The volume of the tank is divided by a 3,55 meters tall and 300 mm thick concrete wall which is situated in the middle of the tank. The tank is equipped with a moving bridge with sludge scrapers. The sludge is hauled into the holes at the inlet zone and pumped out of the tank with sludge pumps. The treated water flows out of the tank through a weir at the far end. The schematics of the tank are shown in Figure 1.

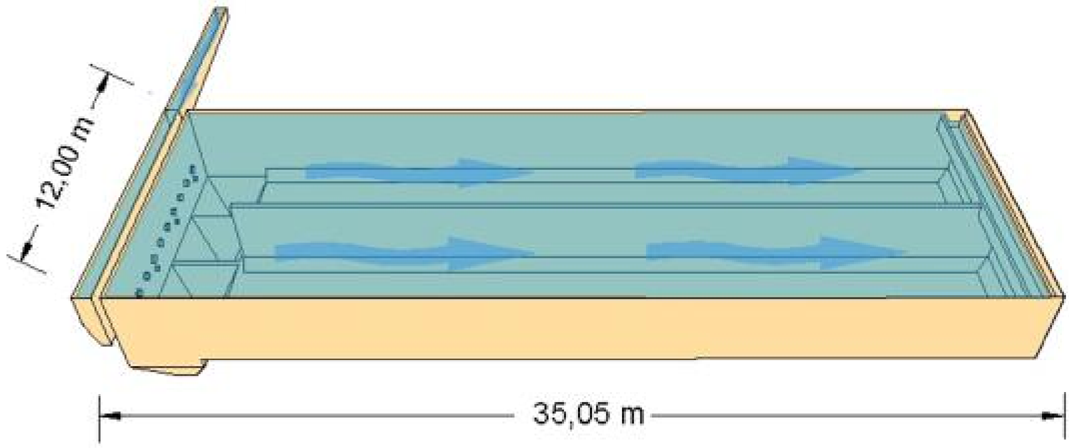

Figure 1 Schematics of the settling tank

The settling tank was simulated in ANSYS Fluent using the default energy equation model and standard K-omega viscous model. The 3D geometry was created in AutoCAD, based on project documentation and measurements and then imported to ICEM CFD, in which the computational mesh was created. Several simplifications were employed for defining the settling tank to conserve time and computational capacity. The surface of the tank was defined as a zero shear stress wall boundary condition as opposed to free surface simulation, which simplifies the surface of the tank to a simple plane with no friction. This means that the surface remains at constant height during the simulations, similar to normal operating conditions in the tank. The height of the surface plane was determined based on field measurements at the site, and so was the flow rate/velocity at the inlet boundary. The weir was simulated as an outflow boundary condition (there is no backflow calculated on this boundary). There were two types of walls with different roughness coefficients to simulate differences between the concrete tank and the steel pipes. The settling tank was simulated with „pure water", meaning without sediment transportation. K-omega turbulence model was used for its precision with low Reynolds number turbulent flows. Four alternative configurations of the settling tank were simulated, to help analyze the flow patterns in the tank. These alternatives are described in Table 1. CFD POST application was used for processing of the results and creating the output images.

No.	Description
1.1	Simulation of real settling tank
1.2	Simulation of the tank with closed floodgate
1.3	Simulation of the tank with closed floodgate and without the separation wall
1.4	Simulation of the tank with closed floodgate with the separation wall elevated above the water surface

Table 1 Description of simulation alternatives

2 METHODS AND RESULTS

The first step in the process of CFD simulation is to create the mesh which will be used for the computations. The mesh for this problem was created in ICEM CFD, based on geometry imported from AutoCAD (**Figure 3**). All the meshes for the various alternatives are composed exclusively of triangles and tetrahedrons, details can be seen in **Table** .

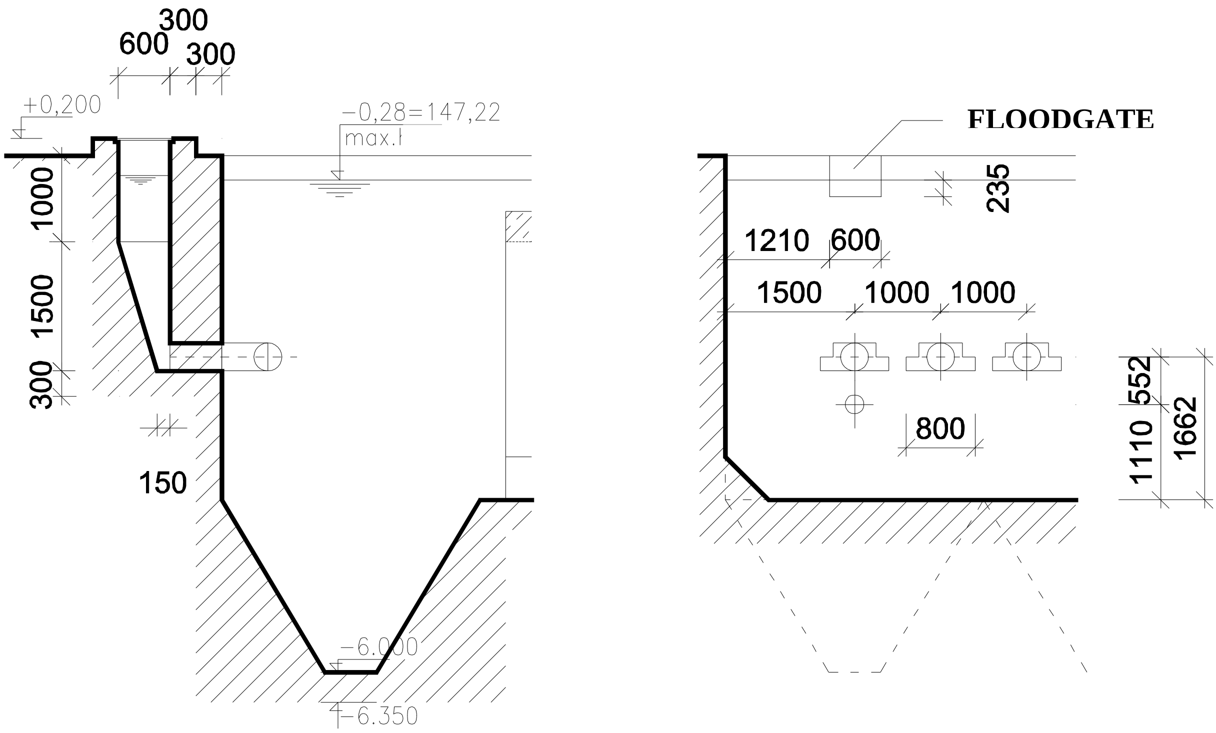

Figure 2 Inlet chamber and inlet pipes

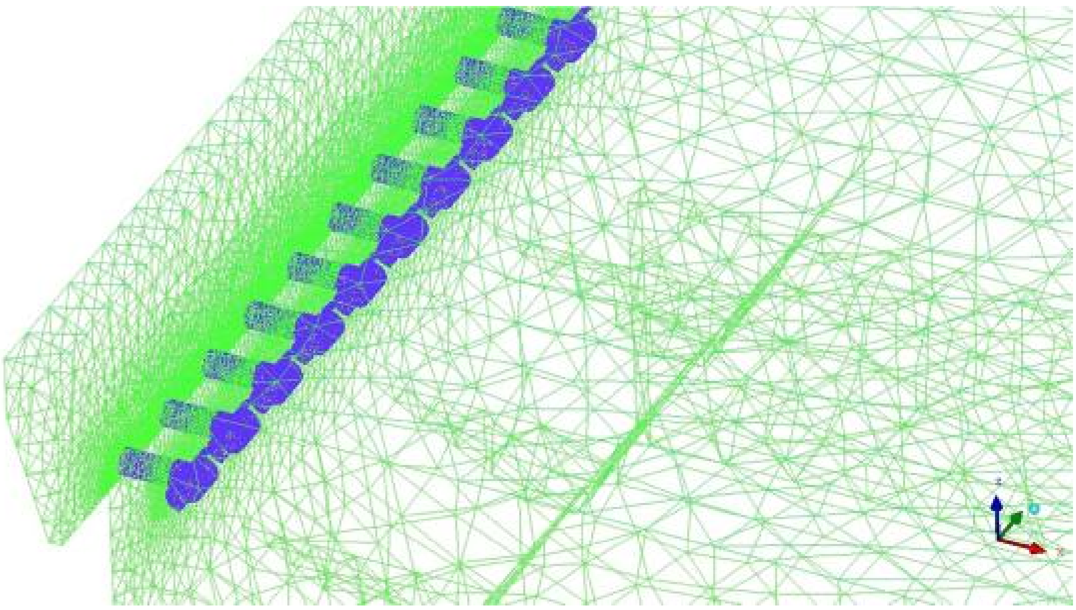

Figure 3 Example mesh, pictured is the inlet zone for alternative 1.1

	No. Of elements	No. Of nodes	Triangle	Tetrahedron
1.1	1243764	214403	73524	1163370
1.2	1212514	208965	70880	1134929
1.3	1210318	208615	70800	1132816
1.4	1213098	209019	69762	1136935

Table 2 Compositions of meshes for the alternatives

After generation, the mesh is loaded into the FLUENT software. The boundary conditions were defined according to Table 1.

Boundary condition	Specification
velocity inlet	v_{st}=0,276 m.s-1 for the floodgate v_r=0,05 m.s-1 for inlet pipes
outflow	-
wall (surface)	Zero shear stress
wall (concrete)	Standard wall function
wall (pipes)	Standard wall function
fluid	Defined as h2o<l>, T=298°K

Table 3 Boundary conditions

The result of the simulation is a lot of information and simulated data, such as velocity, turbulent kinetic energy, pressure, turbulent viscosity, etc.

The results of simulations were evaluated based on three main types of graphical output:

- Velocity streamlines which help us identify some of the hydraulic phenomena in the tank. Allow a better understanding of the flow patterns in the tank.
- Speed contours, which show the velocity field in the tank at user-defined locations.
- Isosurface - connecting points with the same value in the volume of the tank.

The parameter of limit velocity (v_H) was derived for the purpose of evaluation of the different variations of the tank. This velocity is the horizontal component of the velocity vector in the longitudinal direction (in relation to the reservoir). Is derived from the calculation of residence time t [h] proposed by the Slovak standard STN 75 6401. According to the formula:

$$v_H = \frac{L_1}{t} \tag{2.1}$$

$$z_L = \frac{34{,}9}{3} = 11{,}63\, m.\hbar^{-1} = 0{,}0032\, m.s^{-1}$$

Where L_1 is the effective length of the tank and t=3 hours is the residence time for minimal inflow, proposed by the standards. Isosurfaces of this parameter were used with the graphical analysis of the tank.

2.1 Simulation 1.1

This is a simulation of the real settling tank, according to project documentation and on-site measurements. The shape of the inlet pipes coupled with the perpendicular direction of the inflow channel directs the flow in the inlet zone to the right (Figure 4, Figure 5). As a result, we can see higher velocities in the sludge holes on the right-hand side, which could potentially lead to re-suspension of the settled sediments (Figure 6). In general, the velocities in the right-hand side of the tank are higher, which can be seen in Figure 7Figure 5. There is a visible surface current, running from the floodgate directly to the outlet weir which could have a serious negative impact on function of the tank.

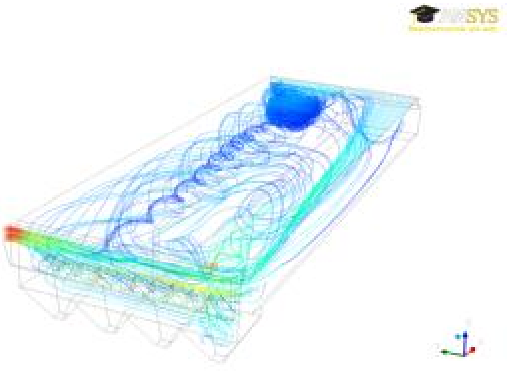

Figure 2 Velocity streamlines (simulation 1.1)

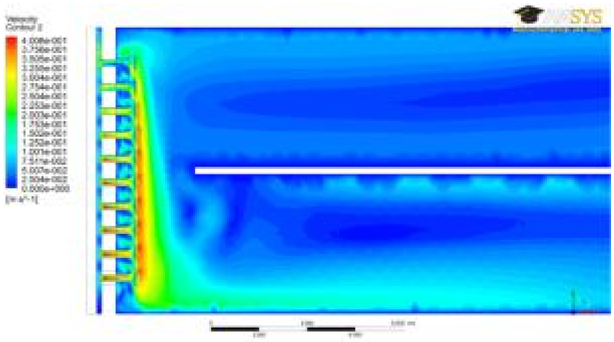

Figure 3 Velocity contours, top view (simulation 1.1)

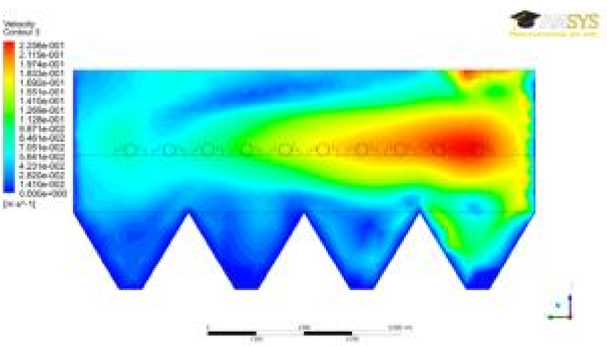

Figure 4 Velocity contours, view of the sludge hole (simulation 1.1)

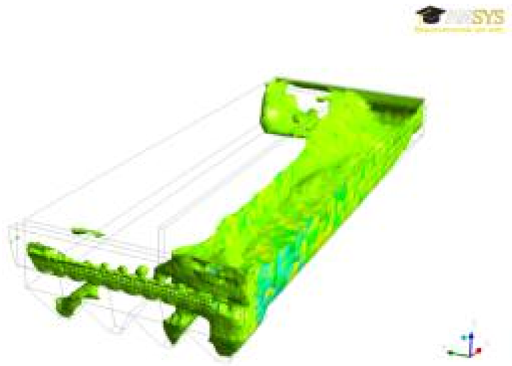

Figure 5 Isosurface for the v_H parameter (simulation 1.1)

2.2 Simulation 1.2

This simulation is almost identical to 1.1, the only difference is the closed floodgate. The flow patterns are similar to 1.1 (Figure 8, Figure 9) including the velocities in the sludge holes. The flow is slower at the surface thanks to the closed floodgate.

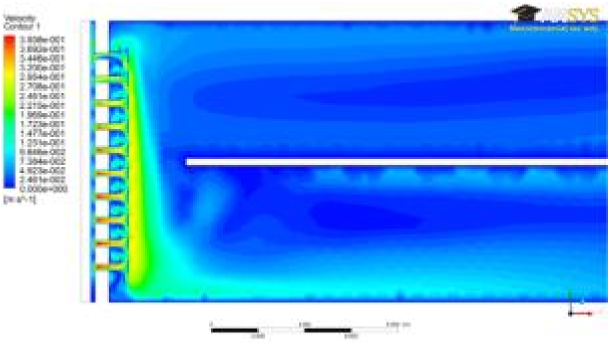

Figure 6 Velocity contours, top view (simulation 1.2)

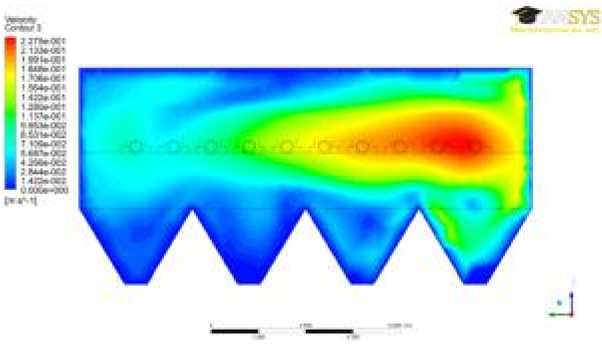

Figure 7 Velocity contours, view of the sludge holes (simulation 1.2)

2.3 Simulation 1.3

This variant simulates extending the separation wall above the surface. As a result, the flow pattern has slightly changed and overall, velocities in the tank grew (Figure 14) The problem in the sludge

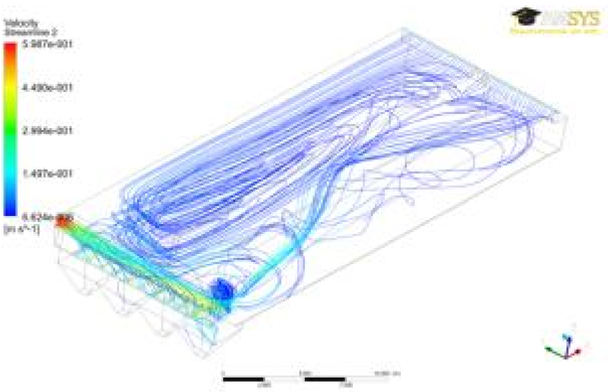

Figure 8 Velocity streamlines (simulation 1.3)

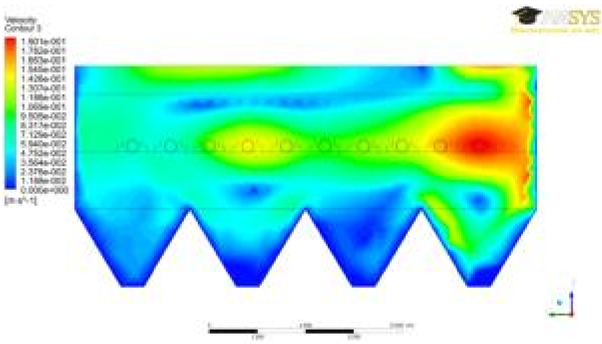

Figure 9 Velocity contours, view of the sludge holes (simulation 1.3)

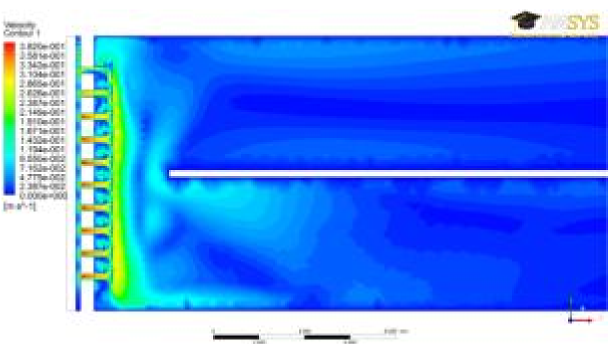

Figure 10 Velocity contours, top view (simulation 1.3)

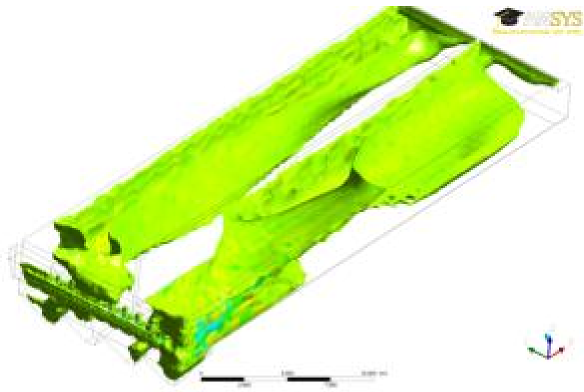

Figure 11 Isosurface for the v_H parameter (simulation 1.3)

hole still persists (Figure 13)

2.4 Simulation 1.4

Variant 1.4 simulates the tank without the separation wall in the middle. Removing the wall creates a right-hand rotating flow system (Figure 16) due to the shape of the inlet pipes and the perpendicular inflow channel. The limit velocity (v_H) has been exceeded in a large part of the tank as seen in Figure 15.

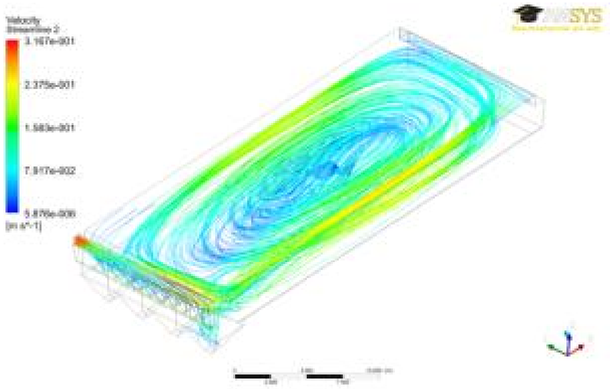

Figure 12 Velocity streamlines (simulation 1.4)

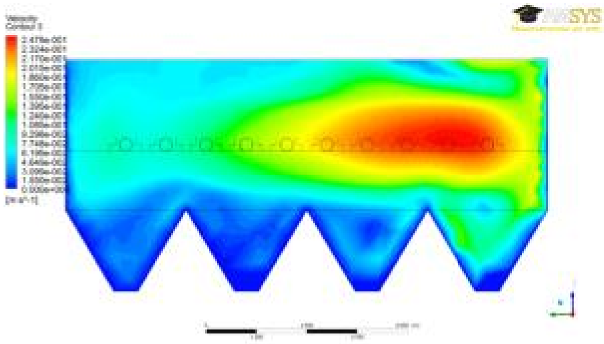

Figure 13 Velocity contours, view of the sludge holes (simulation 1.4)

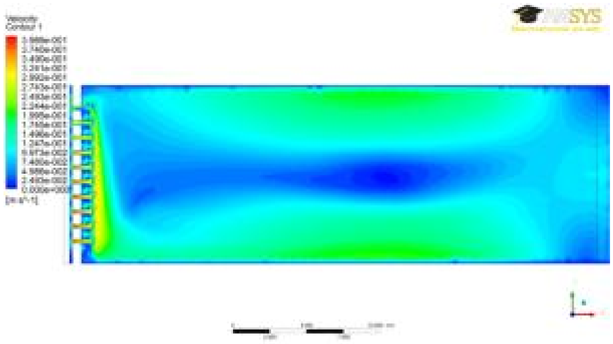

Figure 14 Velocity contours, top view (simulation 1.4)

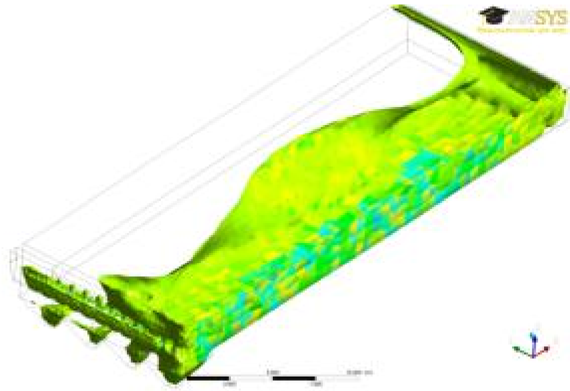

Figure 15 Isosurface for the v_H parameter (simulation 1.4)

3 CONCLUSIONS

All four simulations point to the direction of inflow as the strongest negative influence on the flow in the tank. The biggest problem – increased velocities in the sludge holes on the right side of the tank – is caused by the perpendicular inflow of wastewater into the tank from the left side. The velocities in the sludge holes were a problem in all of the four scenarios, regardless of the changes. These results indicate possible hydraulic problems, which could have a negative influence on overall performance of the tank.

However, another simulation series with particle transport and sedimentation is needed to get conclusive results. As a side note, based on the outcomes, we definitely advise the operator to operate the tank with closed floodgate, as this may lead to short-circuit currents and low settling efficiency.

This paper focuses on the simulation of hydraulics of wastewater treatment settling tank. The settling tank in WWTP Humenné was selected based on the survey of operators of wastewater treatment plants. The WWTP was recently reconstructed and therefor new documentation was available. The work also touches on the subject field measurements of discharge in settling tanks. The work focuses on modeling of various alternatives of the tanks geometry, in order to find the most effective configuration for its operation. ANSYS FLUENT CFD model was utilized for all simulations.

4 ACKNOWLEDGMENTS

This work was supported by the Scientific grant agency of MŠ SR and SAV (VEGA) under the contract VEGA 1/0631/15.

References

[1] 1. **Molnár, Vojtech.** *Počítačová dynamika tekutín: interdisciplinárny prístup s aplikáciami CFD.* Bratislava : Slovenská technická univerzita v Bratislave, 2011. ISBN 9788081060489.

[2] 2. **Ghawi, Ali Hadi.** *A numerical model of flow and settling in sedimentation tanks in potable water treatment plants.* Bratislava : STU Bratislava, Stavebná fakulta, 2008. 978-80-227-2964-2.

[3] 3. **Janssen, Robert H.** Analysis and design of sediment basins. *Eighth National Conference on Hydraulics in Water Engineering.* Gold Coast, AU : The Institution of Engineers, Australia, 2004.

[4] 4. **Rodi, Wolfgang.** *Turbulence models and their application in hydraulics, a state-of-the-art review.* Third edition. Rotterdam : A. A. Balkema, 1993. ISBN 90 5410 150 4.

[5] 5. **Athanasia M. Goula, Margaritis Kostoglou, Thodoris D. Karapantsios, Anastasios I. Zouboulis.** *A CFD methodology for the design of sedimentation tanks in potable water treatment: Case study: The influence of a feed flow control baffle*s. l. : Elsevier B.V., 2008, Zv. 140, s. 110-121.