Computer modeling of flow on the object of the sewer network

M Šutúš¹, J Hrudka¹, G Rózsa¹ and I Škultétyová¹
¹ Slovak University of Technology in Bratislava, Faculty of Civil Engineering, Department of Sanitary and Environmental Engineering, Slovakia

Abstract. This work deals with the creation of a hydrodynamic model of the combined sewerage overflow chamber in Banská Bystrica. As an object on the sewer network, we chose a combined sewerage overflow chamber, because its correct functionality directly affects the quality of the environment in the recipient. Simulations were calculated in the program Ansys Workbench 19.2 – Fluid Flow (Fluent). The basis of the work was the creation of a 3D graphic model. The 3D model serves as a base for flow modeling. The core of the work are simulations in the combined sewerage overflow chamber that should correspond to real flow. The goal of our work is to compare the influence of inflow velocity and flow rakes on hydrodynamic flow.

1. Introduction
In the majority of urban areas in Slovakia, wastewater is drained through a single sewer system. At present, CSOs are used to regulate the rain flow and are therefore a typical object for these systems. Their purpose is to reduce the amount of rainwater supplied to the treatment plant and thus reduce the uneven load of the treatment plant by rainwater.

The efficiency of the CSOs depends on appropriate design and regular maintenance. Since not all of the CSOs are properly hydraulically designed, the recipient often becomes polluted and thus the quality of surface and groundwater is reduced.

The assessment of the functioning of the individual CSOs presupposes the use of a relatively economically demanding monitoring system. The use of such a monitoring system is too costly for long-term observation.

Mathematical-computer modeling can be an effective means of simulating possible phenomena and processes on the sewer network. Mathematical simulations will be created using ANSYS Fluent software.

2. Ansys Fluent – CFD
CFD (Computational Fluid Dynamics) stands for the modern calculation method. This method offers us using various software modeling and calculation methods, which the user can then choose according to the problem he solves in the project. In Ansys Fluent, we can model both 2D and 3D fluid flow; liquids and gases. This program has a huge application in terms of hydraulic, water and research [1,2,3,4].

The ANSYS CFD includes the Fluent solver, which provides real-world simulations of both 2D and 3D fluid and gas flow. Materials are available in a comprehensive material library, or you can create your own material and specify their specific properties. The mesh generator offers automatic mesh creation based on the selected simulation with the possibility to set the mesh and refinement parameters in detail. It allows us to solve sophisticated physical phenomena with maximum accuracy. The results show the flow, pressures, and temperatures with the possibility of their animation, maximum and minimum values, specific sectional views, and nozzles [2,4].

ANSYS CFD is integrated into the ANSYS Workbench - a platform designed for efficient and flexible workflows. It has CAD associativity and powerful features for modeling geometry and
networking. The integrated parameter manager makes it easy to perform a parametric analysis. ANSYS Workbench Workflow makes it easy to link analyzes and solve multiple tasks [2,3,4].

2.1. Modeling
CSO was modeled in modeling software SpaceClaim according to the drawings. Dimensions of CSO are (h x w x l) 2660 mm, 5450 mm and 14150 mm. The inflow sewer has a diameter of DN 1600, the relief sewer has a diameter of DN 1600 and the bypass to the WWTP has a diameter of DN 600.

![Figure 1. Model of CSO.](image)

2.2. Boundary conditions
Calculation used the SIMPLE method and the k-epsilon model [4,5,6]. During the simulation, the model state of extreme operation with inflow velocity $v = 2.0 \text{ m.s}^{-1}$ was considered and in normal operation with inflow velocity $v = 0.5 \text{ m.s}^{-1}$.

The boundary conditions are defined as follows in table 1.

| Boundary condition | Boundary condition | Definition            |
|-------------------|--------------------|-----------------------|
| Velocity Inlet    | Inlet              | 0.5 / 2 m.s$^{-1}$    |
| Wall              | Wall               | 0.013 mm              |
| Wall0             | Water level        | 0 mm                  |
| Outflow           | Outlet             | -                     |
| Solid             | Volume             | $\text{H}_2\text{O}$  |

3. Simulations
Four scenarios were varied in which the inflow rate and the height of overflow edge changed:

1. 0.5 m.s$^{-1}$, height of overflow edge (normal) 290 mm
2. 2.0 m.s$^{-1}$, height of overflow edge (normal) 290 mm
3. 0.5 m.s$^{-1}$, height of overflow edge (increased) 350 mm
4. 2.0 m.s$^{-1}$, height of overflow edge (increased) 350 mm
4. Results
In simulations, speed velocity was compared. Velocity was measured at the outlet to the WWTP, overflow edge and outlet to a recipient. In each simulation, speed was measured in points with the same coordinates. The results are summarized in table 2.
Table 2. Velocity comparison.

| Height of overflow edge | 290 mm | 350 mm |
|-------------------------|--------|--------|
| Velocity [m.s\(^{-1}\)] |        |        |
| Time [sek] | WWTP edge | Outlet | WWTP Edge | Outlet | WWTP Edge | Outlet | WWTP Edge | Outlet |
| 5         | 2.15 - -   | -      | - - -     | -      | - - -     | -      | - - -     | -      |
| 15        | 2.08 1.14  | -      | 0.52 -    | -      | 2.09 1.08 | -      | 0.53 -    | -      |
| 35        | 2.15 1.16  | 2.04   | 0.53 0.23 | -      | 2.16 1.07 | 1.98   | 0.54 0.23 | -      |
| 50        | 2.18 1.16  | 2.05   | 0.53 0.25 | -      | 2.18 1.09 | 1.98   | 0.53 0.26 | -      |

5. Conclusion
This work deals with mathematical modeling and 3D modeling of existing CSO. The core of the work are hydrodynamic simulations of flow in Ansys Fluent. Two load cases were selected that correspond to extreme inflow velocity \( v = 2.0 \) m.s\(^{-1}\) and a more common situation with inflow velocity \( v = 0.5 \) m.s\(^{-1}\). The results show that height change of overflow edge slightly reduced the velocity in the extreme state. In the future, authors want to focus on whether the change of height, change of slope or combination of changes can improve flow in CSO.

6. Acknowledgments
This work was supported by the Scientific Grant Agency of the Ministry of Education, Youth and Sports of the Slovak Republic and the Slovak Academy of Sciences within the project VEGA 1/0574/19, co-funded by the European Regional Development Fund and by the Slovak Research and Development Agency under the contract No. APVV-18-0203.

7. Reference
[1] Molnár V 2011 Computational Fluid Mechanics: Interdisciplinary Approach with CFD Applications *Spektrum STU Bratislava* ISBN 9788081060489
[2] ANSYS FLUENT 12.0 Theory Guide, Canonsburg: ANSYS Inc. (2009)
[3] Kozúbková M 2008 Modeling of fluid flow FLUENT CFX Ostrava: VSB – Technical University of Ostrava
[4] Bojko M 2010 3D flow – ANSYS Fluent Ostrava: VSB – Technical University of Ostrava
[5] Hrudka J, Stanko Š and Holubec M 2017 Analysis of flow and sedimentation processes in secondary sedimentation tank *Pollack Periodica* vol. 12 no. 2 pp 79 – 89
[6] Gregušová V, Škultétyová I and Holubec M 2017 Measurement of flow velocity in different secondary settling tanks for analysis of the flow *Pollack Periodica* vol. 12 no. 3 pp 15 - 22