AKADÉMIAI KIADÓ

Hydrodynamic assessment of combined sewerage overflow chamber in Banská Bystrica

Marek Šutúš*, Jaroslav Hrudka, Gergély Rózsa, Ivona Škultétyová and Štefan Stanko

Department of Sanitary and Environmental Engineering, Faculty of Civil Engineering, Slovak University of Technology in Bratislava, Radlinského 11, 810 05 Bratislava, Slovakia

Pollack Periodica • An International Journal for Engineering and Information Sciences

16 (2021) 2, 50–55

DOI: 10.1556/606.2020.00171

Received: December 31, 2019 • Revised manuscript received: July 17, 2020 • Accepted: August 20, 2020
Published online: March 26, 2021

ORIGINAL RESEARCH PAPER

ABSTRACT

This research deals with creating a hydrodynamic combined sewerage overflow chamber model an ANSYS Workbench 19.2, Fluid Flow (Fluent). The 3D graphics model was created in the SpaceClaim modeling software, which serves as the basis for hydrodynamic modeling. The model was created according to a real combined sewerage overflow chamber in Banská Bystrica. The cores of the work are simulations that should correspond to the estimated flow in the combined sewerage overflow chamber. The aim of this paper is to compare the impact of inflow speed and flow rakes.

KEYWORDS

3D model, combined sewerage overflow chamber, hydrodynamic model

1. INTRODUCTION

Computational Fluid Dynamics (CFD) stands for the modern calculation method. This method offers various software modeling and calculation methods, which can be chosen according to solution of the problem in the project. Therefore, this computer technology has found applications not only in the water but also in the engineering, automotive and aerospace industries. In ANSYS Fluent, researcher can model both 2D and 3D fluid flow; liquids and gases. This program has a huge application in terms of hydraulic, water and research [1–6].

CFD modeling has a huge financial and time advantage over physical models. To create a physical model, a team of workers is needed to create a model on a precise scale. It may take several weeks to build physical model. On the other hand, using CFD modeling there can be created a model in terms of complexity within hours to days. At the same time, the CFD model easily can be modified to achieve optimum status. The disadvantage of CFD modeling is the need for detailed knowledge of the problem [1–6].

As an object on the sewer network, the combined sewerage overflow chamber was chosen, because its correct function directly affects the quality of the environment in the recipient. For the basis of the model, a real combined sewerage overflow chamber was chosen, which is located in Banská Bystrica, in the city district Rudlová.

2. COMBINED SEWERAGE OVERFLOW CHAMBER

On the sewer network, there are rains separating objects, which during the rain divert a part of rainwater to the recipient. If this object is located on a uniform sewer system it is called Combined Sewerage Overflow chamber (CSO).

If CSO is used, sections of sewers carrying wastewater further into WasteWater Treatment Plants (WWTPs) can be designed with smaller cross-section, and thus uneven load of

*Corresponding author.
E-mail: marek.sutus@stuba.sk

AKJournals

WWTPs by rainwater can be reduced [7]. The minimum efficiency of mixed wastewater treatment in WWTPs should neither be reduced nor the conditions in the recipient for the conservation of biological life under short and long-term stress be worsened.

Sewer network and recipient are connected by separation objects. These objects must, therefore, be designed with regard to the safe functioning of the sewer network even during the flow of large waters in the recipient due to possible backwater [7].

The operation and operation of the CSOs are influenced by:

- water quality in recipients, especially low watercourses and recipients with standing waters, resp. slowly flowing waters;
- values of pollution indicators at the entrance to the WWTP in connection with increased inflow of rainwater and treatment of mixed wastewater;
- investment costs for the construction of collectors of a single sewer system; and WWTP objects.

2.1. Principle of combined sewerage overflow chamber

The task of CSO is to divide the mixed wastewater flow Q_v into the permissible mixed wastewater flow rate Q_c to the WWTP and to the lightened flow Q_p into the recipient. The most common designs are CSO with front or side edge. If the mixed water flow level in CSO does not exceed the level of the overflow edge, the entire mixed water flow is directed to the WWTP. If the inflow level exceeds the level of the overflow edge, the flow is divided - part flows to the wastewater treatment plant and part flows through the relief discharge to the recipient. The higher the level in the inflow sewer, the greater the outflow of mixed water to the recipient, but also the outflow to the wastewater treatment plant increases as a result of the height of the overflow.

3. ANSYS FLUENT – CFD

Quick and accurate simulation of fluid flow and heat dissipation helps to predict and understand product behavior, optimize design, and validate its behavior before commissioning. Flexible comparison of alternative designs saves the cost of physical testing and speeds up product innovation. It allows for efficient simulation of compressible and incompressible flow, single and multi-phase flow, combustion, particle flow, aircraft icing, turbo-machines, and many other analyzes.

The ANSYS CFD includes the ANSYS Fluent solver, which provides real-world simulations of both 2D and 3D fluid and gas flow. The materials are available in a comprehensive material library or materials with specified properties can be created. 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 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, 5, 6].

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, 5, 6].

3.1. 3D model

ANSYS Workbench includes a software package that includes the SpaceClaim CAD software. In this program, the existing CSO was modeled according to the drawings.

The commands were model CSO (Fig. 1) with dimensions (w, x, h, x, l) 5,450 mm, 2,660 mm and 14,150 mm. The inflow sewer has a diameter of DN 1,600 mm, the relief sewer has a diameter of DN 1,600 mm and the bypass to the WWTP has a diameter of DN 600 mm.

The next step was to create a model of the volume of water in which the fluid flow will be modeled. This volume was created by overlaying the CSO model and the block object (Fig. 2) and their subsequent difference (Fig. 3).

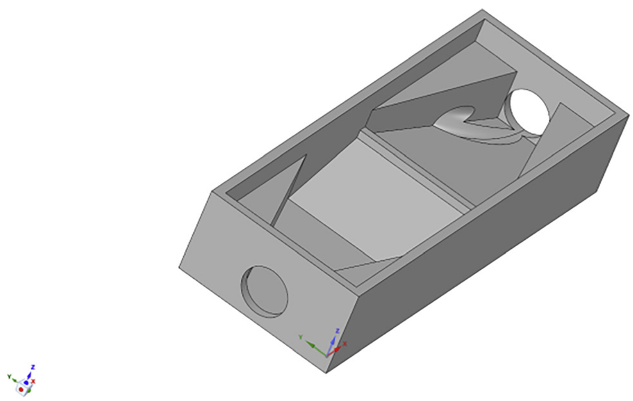

Fig. 1. Model of CSO

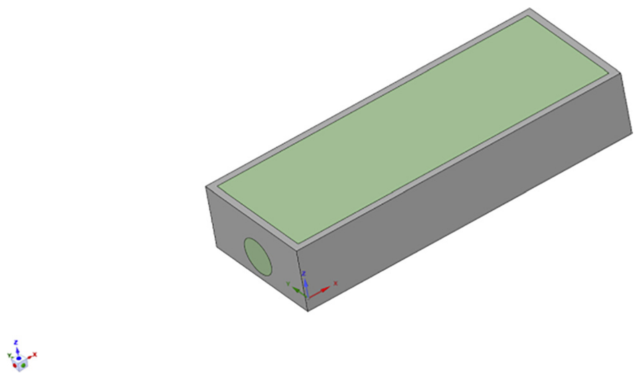

Fig. 2. Overlaying CSO with another object

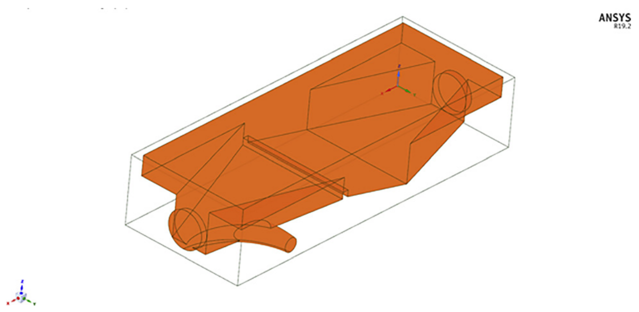

Fig. 3. Volume of simulated medium in CSO

3.2. Mesh

After defining the geometry, the next step is to create a mesh. Mesh is a polygon computing network (Fig. 4) that applies to the model. This mesh is made up of a cluster of vertices, edges, and surfaces and defines the properties of the 3D model. These elements have the shape of pyramids (Fig. 5) [2, 3, 5].

After defining network parameters e.g. minimum and maximum element size, mesh customization or polygyny curvature, the program generates a mesh.

3.3. Boundary conditions

The simulation method used the SIMPLE method and the k – epsilon model [1–4]. The simulation corresponds to the estimated flow in CSO. For detailed measurements, it is necessary to obtain calibration parameters entering the calculations. These parameters could be obtained by field measurement in water relief. During the simulation, the model state of extreme operation with inflow velocity v = 2.0 m.s^{-1} was considered and in normal operation with inflow at v = 0.5 m.s^{-1}.

The boundary conditions are defined as follows in Table 1.

4. SIMULATIONS

Four scenarios were varied in which the inflow rate and the use of rakes were changed:

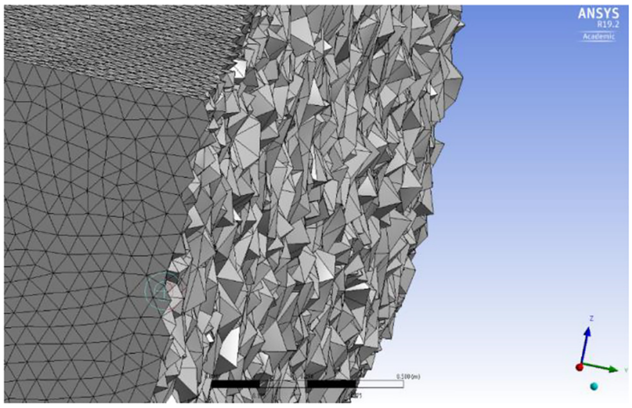

Fig. 5. Pyramid elements

Table 1. Boundary condition

Boundary condition	Boundary condition	Definition
Velocity Inlet	Inlet	0.5/2.0 m s^{-1}
Wall	Wall	0.013 mm
Wall0	Water level	0 mm
Outflow	Outlet	–
Solid	Volume	H_2O

- 1st scenario – 2.0 m.s^{-1}, rake;
- 2nd scenario – 0.5 m.s^{-1}, rake;
- 3rd scenario – 2.0 m.s^{-1}, without rake;
- 4th scenario – 0.5 m.s^{-1}, without rake.

To simplify mesh formation and calculations, square-sectioned rake models were modeled. The rakes have an edge with a length of 14 mm and a height of 900 mm.

4.1. Simulation No. 1

At time t = 50 s the water flows in the ceiling of the CSO (Fig. 6) and then drains. This flow is inadmissible in CSO.

4.2. Simulation No. 2

At time t = 50 s the water continuously fills CSO (Fig. 7) and begins to discharge.

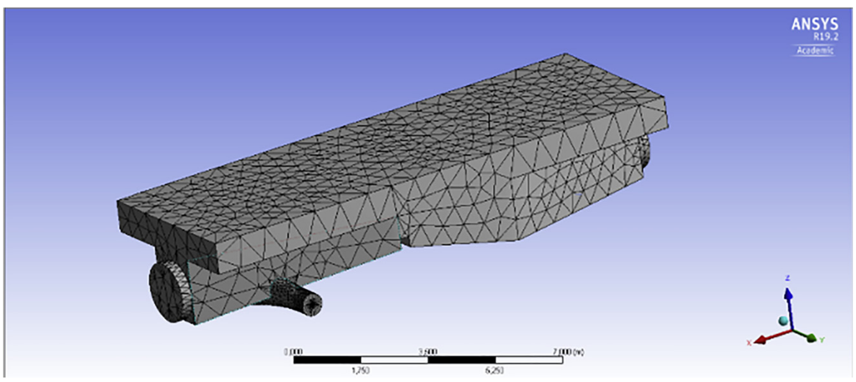

Fig. 4. Generated mesh

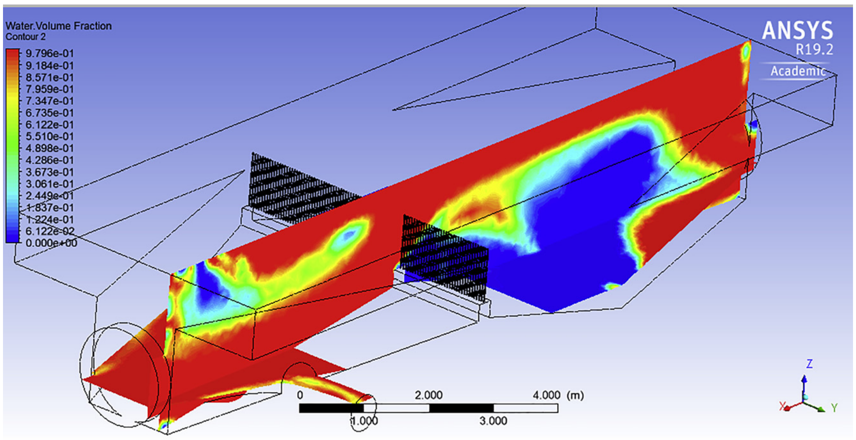

Fig. 6. Simulation No. 1 at time $t = 50$ s

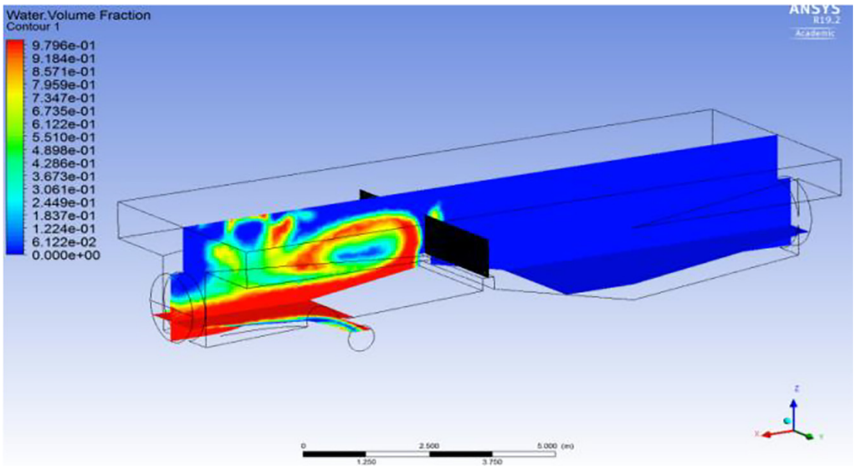

Fig. 7. Simulation No. 2 at time $t = 50$ s

4.3. Simulation No. 3

CSO (Fig. 8) is being filled; a small part of the water was ejected through the edge. At time $t = 50$ s the water begins to discharge.

4.4. Simulation No. 4

The water slowly fills the CSO (Fig. 9) and flows continuously to the WWTP. The water begins to discharge. At time $t = 50$ s the water begins to fall over the edge.

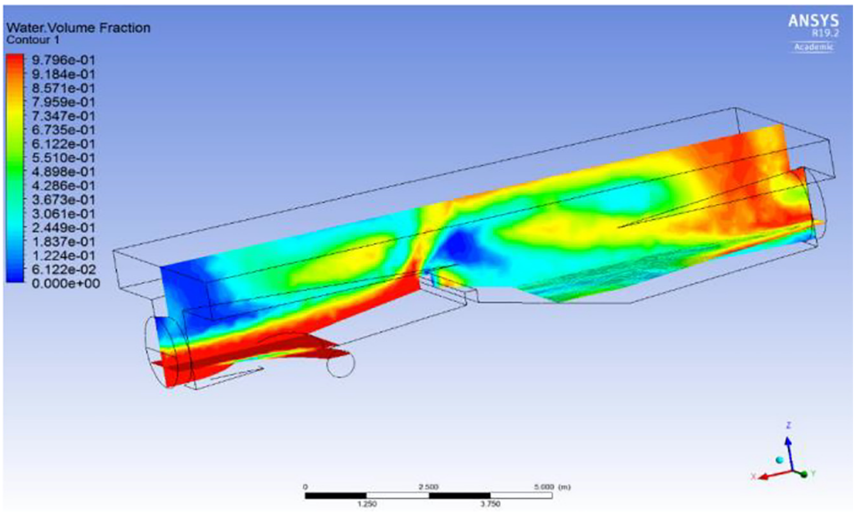

Fig. 8. Simulation No. 3 at time $t = 50$ s

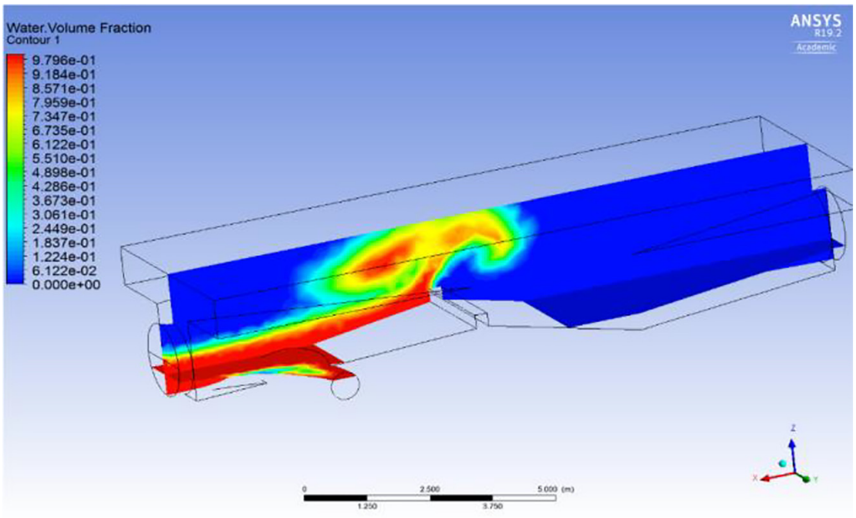

Fig. 9. Simulation No. 4 at time t = 50 s

Table 2. Speed comparison

Sim. n. 1				Sim. n. 2			Sim. n. 3			Sim. n. 4		
Velocity [m.s^{-1}]												
2.0 m.s^{-1}				0.5 m.s^{-1}			2.0 m.s^{-1}			0.5 m.s^{-1}		
t [s]	WWTP	Rakes	Out-let	WWTP	Rakes	Out-let	WWTP	Rakes	Out-let	WWTP	Rakes	Out-let
5	0.95	–	–	0.31	–	–	0.04	–	–	–	–	–
15	1.00	1.56	–	0.04	–	–	0.45	0.86	–	0.05	–	–
35	1.03	1.55	2.33	0.20	0.02	–	0.47	0.87	0.85	0.08	0.04	–
50	1.03	1.55	2.30	0.20	0.08	–	0.47	0.85	0.83	0.09	0.05	–

5. RESULTS

Flow speed was compared (Table 2) in simulations. Measurements were made at the outlet to the WWTP, rakes (overflow edge) and outlet to recipient. In each simulation, speed was measured in points with the same coordinates.

From the results summarized in Table 2 it is clear that the model is not accurate as the values of velocity at the WWTP do not match. These should coincide at the same inflow velocities as they are not affected by rakes and turbulences.

6. CONCLUSION

This work deals with mathematical modeling and 3D modeling of a real combined sewerage overflow chamber in Banská Bystrica. The core of the work is the hydrodynamic simulations of the current in ANSYS Fluent. The research focused on creating a 3D model based on the drawing documentation in the space application, after creating the polygon computational network and entering the boundaries of the mathematical spectrum to overtake the calculation. Two load cases were selected that correspond to extreme inflow load $v = 2.0$ m.s^{-1} and a more common room with inflow floor $v = 0.5$ m.s^{-1}.

This modeling serves as a basis for further work. In the future, the main purpose is to focus on verification, calibration and subsequent validation of the model. For this, it is necessary to obtain input values by field measurement, in which the measuring devices are fitted to CSO.

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.

REFERENCES

[1] P. Urcikán and D. Rusnák, *Sewerage and Wastewater Treatment, Sewerage II* (in Slovak). Bratislava: Spektrum STU, 2008.

[2] V. Molnár, *Computational Fluid Mechanics: Interdisciplinary Approach with CFD Applications* (in Slovak). Bratislava: Spektrum STU, 2011.

[3] *ANSYS FLUENT 12.0, Theory Guide*. Canonsburg: ANSYS Inc, 2009.

[4] M. Kozúbková, *Modeling of Fluid Flow, FLUENT, CFX* (in Czech). Ostrava: Technical University of Ostrava, 2008.

[5] M. Bojko, *3D Flow – ANSYS Fluent* (in Czech). Ostrava: Technical University of Ostrava, 2010.

[6] J. Hrudka, Š. Stanko, and M. Holubec, "Analysis of flow and sedimentation processes in secondary sedimentation tank," *Pollack Period.*, vol. 12, no. 2, pp. 79–89, 2017.

[7] V. Gregušová, I. Škultétyová, and M. Holubec, "Measurement of flow velocity in different secondary settling tanks for analysis of the flow," *Pollack Period.*, vol. 12, no. 3, pp. 15–22, 2017.