Application of mathematical models in design and assessment of sewer network facilities

R Csicsaiová¹, J Hrudka¹, I Marko¹ and Š Stanko¹

¹ Slovak University of Technology in Bratislava, Faculty of Civil Engineering, Department of Sanitary and Environmental Engineering, Slovakia

Abstract. The application of mathematical models has been expanding in the field of sanitary and environmental engineering. Mathematical models are used in the design and assessment of sewer networks and their facilities. Sewer network models make it possible to create a model of hydraulic and Physico-chemical processes in wastewater flowing through the sewage network. The number of extreme rainfall events is increasing due to climate change. It causes a collapse in the infrastructure of urbanized areas. It is possible to investigate the flow of wastewater under extreme rainfall and to propose measures to eliminate adverse events using mathematical models. Nowadays, it is possible to use modern calculation procedures, which are used to dimension and assess existing facilities. This paper aims to focus on the application of numerical models in the design and assessment of combined sewer overflow chambers. The combined sewer overflow serves to carry away a part of the rain flows from the network to the nearest suitable receiving water body. Their main task is to reduce the uneven load of wastewater treatment plants by rainwater. The combined sewer overflow chambers distribute the inflow into the flow going to the wastewater treatment plant and the lightened flow going to the receiving water. The aim of this paper is to summarize the knowledge of CFD modelling and to get acquainted with the basic principles. In brief, the normal flow describes its simulation using two basic models. Finally, it focuses on the recapitulation of foreign studies and their use in the assessment and design of relief chambers and regulatory objects of the single sewer networks.

1. Introduction

Effective and ecological management of rainwater is an integral part of wastewater disposal solution from urbanized areas. It is necessary to design efficient regulatory facilities of a combined sewer network which would regulate the amount of rainwater flowing into the wastewater treatment plant (WWTP) from the urbanized areas [1].

At present, combined sewer overflow (CSO) chambers and rain separators are used to regulate the stormwater inflow to WWTP. Their effectiveness depends on the appropriate design and regular maintenance. The operation of the CSO chambers also has its drawbacks, but the correct configuration of the shape could at least partially eliminate these negative aspects. The most crucial factor in assessing the impact of extreme precipitation on sewage network facilities in the wastewater flow [2].

The application of mathematical models in sanitation engineering is now commonplace. A lot of research deals with flow modeling in sewer networks. Due to the rapid urbanization of the territory and climate change, it is necessary to assess and reconstruct the sewer networks. The development of computer technology creates the assumption that flow modeling will be used not only for the assessment of the functioning of sewer networks and their objects, but also for the design and dimensioning itself [3].

Many studies deal with the effectiveness of regulatory objects using mathematical models. 1D and 3D models used. Older studies have investigated the interaction of several CSO chambers in one sewer
basin. The action of several CSOs in a river basin may create critical conditions in the recipient during tidal rainfall. Alpa and Melching’s studies have focused on creating 1D models and simulating hydrodynamic flow [4]. The most significant changes in the flow occur during heavy rainfall events. During such a collision event, the water flow in facilities is unsteady, with rapid changes in velocities and the possibility of creating vortices [5]. In this case, when examining the wastewater flow in regulatory facilities are used 3D models [3].

The aim of this paper is to summarize the knowledge of CFD modeling and to get acquainted with the basic principles. In brief, the normal flow describes its simulation using two basic models. Finally, it focuses on the recapitulation of foreign studies and their use in the assessment and design of relief chambers and regulatory objects of the single sewer networks.

2. Flow modeling
Computational Fluid Dynamics (CFD) deal with Fluid flow modeling. It combines scientific knowledge from physics, mathematics, fluid mechanics, thermomechanics into simulation - the model of physical phenomena. CFD allows simulating fluid movement using numerical models. It used to solve various engineering tasks in the field of aerodynamics, environmental engineering, material engineering, biological engineering, civil engineering, engineering industry, etc.

In the case of water resourcing engineering, CFDs can used to simulate wastewater flow in wastewater treatment tanks, CSO chambers, and sewerage facilities [6]. It is a useful aid in the design and assessment of CSO chambers. In 3D models, we can simulate the behavior of fluid in the CSO chambers. When designing reconstructions of steel structures, the simulations used to determine the most appropriate additional constructions (floating barrage, screens).

CFD examines fluid flow for changes in their physical properties such as speed, density, temperature, pressure, and viscosity. It is necessary to consider every property as a whole to obtain the most accurate results. The individual mathematical models differ depending on the nature of the problem being solved, i.e., heat transfer, changes in the solid phase, or ongoing chemical reactions. [7].

The flow equations which defining fluid flow modeling are based on three basic flow principles:

- the law of conservation of energy,
- the law of conservation of mass
- the low of conservation of momentum.

Navier - Stokes equations, which form a system of partial differential equations, are also derived from these principles. They are formulated from basic equations of fluid mechanics, namely: continuity equation, motion equations, and energy equation [8].

These equations describe not only fluid flow but also heat transfer between different materials. Mathematical models define the hydrodynamic properties of liquids by four basic equations:

- continuity equation,
- equation of motion,
- energy equation,
- turbulent flow equations.

CFD software solves mathematical models either by finite element method, finite volume method, or finite difference method [9].

These equations present the flow of the laminar liquid. However, this type of flow is rare in nature. The following chapter characterizes the turbulent flow in the CSS facilities and describes the individual solutions of the turbulent flow.

2.1. Turbulent fluid flow
Turbulence as a physical phenomenon affecting the flow pattern occurs in real fluid flow. Therefore, it is necessary to incorporate it into a mathematical model. The flow turbulence determined by the geometry of the object and the buoyancy. Swirls create fluctuations in speed, fluid temperature, and pressure. Turbulent flow generates several vortices with different dimensions. Large eddies gradually interfere to form smaller vortices, which in turn transmit their energy to heat [10].

Simplified solutions have developed to deal with turbulent flow, which can simulate flow relatively accurately. Direct Numerical Simulation (DNS) was created using exact equations.
It is a direct numerical solution of Navier-Stokes equations. There are two types of models using simplified equations. The first type uses a spatial filter to simulate Large Eddy (Large Eddy Simulation - LES). The second type is based on the solution of averaged equations (RANS). Based on the Boussinesq hypothesis, a model with two equations $k - \varepsilon$ or $k - \omega$ and a model with one equation $k$ developed. The Prandtl Baldwin-Lomax model, the Cebeci-Smith model, and the mixing length model also belong to this group of models. The Reynolds stress model, with six additional differential equations, is a separate group [9]. Figure 1 shows the distribution of turbulence models.

![Figure 1. Models of turbulence.](image)

2.1.1. Model Standard $k - \varepsilon$. The $k - \varepsilon$ model is a model with two additional equations that describe turbulence using two transport equations. It is a semi-empirical model based on the Boussinesque hypothesis characterized by two variables:

- Turbulent kinetic energy ($k$),
- The measure of its dispersion ($\varepsilon$).

Unlike previous turbulence models, the $k-\varepsilon$ model focuses on mechanisms that affect turbulent kinetic energy. The underlying assumption is that the turbulent viscosity is isotropic, i.e., the ratio between Reynolds stress and mean strain rate is the same in all directions [11].

2.1.2. Model Standard $k - \omega$. The model $k - \omega$ is solved similarly to $k - \varepsilon$ two differential equations. The equation for the specific energy dissipation rate $\omega$ has several disadvantages compared to the equation $\varepsilon$. This model based on the Wilcox $k - \omega$ model, which includes modifications for low Reynolds numbers, compressibility, and shear flow propagation. The Wilcox model shows its imperfections by sensitivity in solving the values of $k$ and $\omega$ in the free current outside the shear layer. The Standard $k - \omega$ model partially eliminates them.

The Standard $k - \omega$ model is an empirical model based on the solution of transport equations for the kinetic energy of turbulence $k$ and the specific rate of energy dissipation $\omega$. The scattering rate is essentially the ratio $\varepsilon / k$ [12].

3. Use of 3D models in flow modeling in combined sewer overflow chambers
The CSO chambers serve to divert a portion of the rain inflow from the network to the nearest suitable receiving water body. Their main task is to reduce the uneven load of wastewater treatment plants by rainwater by dividing the inflow to the CSO into the flow flowing to the WWTP and the overflow flowing to the receiving water [13].

The basic principle of flow distribution in CSO is the overflow of water over the overflow edge. If the wastewater level reaches the level at the level of the overflow edge, the water will begin to overflow through it and flow to the receiving water [14]. This overflow contains pollutants, including gross suspended solids, suspended solids, and dissolved solids [14], [15]. High demands placed on the hydraulic design of the building. From the hydraulic point of view, the design of the relief chamber is
most similar to the weir. However, their complex design does not allow direct measurement of flow and velocities in individual parts of the facilities. Therefore, 3D CFD models currently used in the design and assessment of these objects [15].

Chen et al. used a 3D model of the control object to optimize the design of the Edmonton City Sewer System. The conclusion of the case study concludes that the use of the 3D model is a useful tool for the assessment and design of a sewer system [16].

Another study applied a 3D model to simulate the spread of E. coli in the receiving water body. The simulation was part of the analysis of the sewer network discharging the sink from the system into the marine environment. The study resulted in the application of the 1D model of a combined sewer system (CSS) in combination with the CFD model of a facility in a case study in Italy [17].

Department of Sanitary and Ecological Engineering CTU in Prague, in cooperation with Hobas, developed a new type of CSO, which they called Tube CSO or CSO-T. It is a simple construction of two interconnected fiberglass pipes of different diameters superimposed. To verify the functioning of the CSO, they constructed both a physical and a mathematical model, where they tested different variants of the slope of the sewer and varying flow rates of wastewater [18].

3D CFD model can also be applied in the case of invert traps. The model simulated the flow of water with a large amount of sediment. The study demonstrated the applicability of the VOF model to the stochastic discrete phase model [19].

A study by J. C. Quijano et al. evaluated the receiving water quality for the Chicago Metropolitan Area using the Environmental Fluid Dynamics Code (EFDC) with the Water Quality Analysis Simulation Program (WASP) to perform three-dimensional simulations of hydrodynamics and water quality. Two precipitation events with different properties were evaluated [5].

4. Conclusion
The development of urbanized areas, population growth, and climate change call for a change in the wastewater management concept. Many large cities have prepared sewer network assessments. These assessments often include evaluations of individual network facilities that have an impact on the status of the receiving water. The ecological state of the receiving water is affected by the surface runoff and the discharge from the CSO chambers.

Individual studies examined the interactions of multiple CSOs during heavy rainfall events. Intense precipitation causes flow changes in the CSS facilities. It is possible to simulate the behavior of wastewater flow in the CSO chamber using 3D models. It is possible to modify the individual parts of the constructions with the possibility of improving their operation and reducing the discharged pollution based on the simulations.

Many studies dealt with the development of new structures, and simulations subsequently verified their design. 3D simulations were also implemented in studies of individual river basins while optimizing CSS.

These models help to meet the objectives of the Water Framework Directive (CEC), Directive 2000/60/EC. This paper summarizes the knowledge about the use of 3D models in sanitary engineering and describes the basic principles of flow modeling.

5. References
[1] Niyogi D, Lei M, Kishtawal C, Schmid P and Shepherd M 2017 Urbanization impacts on the summer heavy rainfall climatology over the eastern United States. Earth Interactions 21 (5), DOI: 10.1175/EI-D-15-0045.1
[2] Gao B, Gao L, Xu D, Zhang M, Qu X and Li Y 2019 A novel method for evaluating the potential release of trace metals associated with rainfall leaching/runoff from urban soils. Sci. Total Environ. 664 37–44
[3] Morales V M, Mier J M and Garcia M H 2017 Innovative modeling framework for combined sewer overflows prediction Urban Water Journal 14 (1) 97-111 DOI: 10.1080/1573062X.2015.1057183
[4] Alp E and Melching C S 2009 Evaluation of the duration of storm effects on in-stream water quality J. Water Resour. Plan. Manag. ASCE 135 (2) 107–116 http://dx.doi.org/10.1061/(ASCE)0733-9496(2009)135:2(107)

[5] Quijano J C, Zhu Z, Morales V, Landry B J and Garcia M H 2017 Three-dimensional model to capture the fate and transport of combined sewer overflow discharges: A case study in the Chicago Area Waterway System Science of the Total Environment 576 362–373 http://dx.doi.org/10.1016/j.scitotenv.2016.08.191

[6] Hrudka J, Stanko Š and Holubeč M 2017 Analysis of flow and sedimentation processes in secondary sedimentation tank Pollack Periodica 12 (2) 79–89

[7] Lipeme kouyi G 2011 The use of CFD modelling to optimise measurement of overflow rates in a downstream-controlled dual-overflow structure Water science and technology [online] 64(2) 521-527 [cit. 2020-02-28] ISSN 0273-1223

[8] Jahoda M Počítačová dynamika tekutín- užitečný nástroj pro inženýry [online] [cit. 2020 – 01 -12] URL: http://uchi.vscht.cz/uploads/pedagogika/bezpecnostni_inzenyrstvi/CFD.shrnuti.pdf

[9] Molnár V 2011 Počítačová dynamika tekutín: interdisciplinárný prístup s aplikáciami CFD (Bratislava: Slovenská technická univerzita v Bratislave) 492 ISBN 978-80-810-6048-9

[10] Wilcox D C 1998 Turbulence modeling for CFD (DCW Industries, Inc. La Canada California) 3D urban-like models Building and Environment 169 https://doi.org/10.1016/j.buildenv.2019.106569

[11] Zhang Y, Yang X, Yang H, Zhang K, Luo Z, Hang J and Zhou S 2020 Numerical investigations of reactive pollutant dispersion and personal exposure in 3D urban-like models E. coli distribution in coastal areas Building and Environment 169 https://doi.org/10.1016/j.buildenv.2019.106569

[12] ANSYS FLUENT 12.0 2009 Theory Guide. Canonsburg: ANSYS Inc. [online] [cit. 2019 – 01 -10] URL: http://www.ansys.com/documentation/fluentscience

[13] Butler D and Davies J W 2004 Urban Drainage. 2nd Edition (London: Spon Press) ISBN 0-203-34190-2

[14] Saul A J 2004 Increasing knowledge of CSO technologies. In: Marsalek J et al. editors. Enhancing urban environment by environmental upgrading and restoration. NATO science series IV: earth and environmental sciences 43 105-110

[15] Insel S, Dufresne M, Fischer M and Vazquez J 2014 Assessment of the overflow discharge in complex CSO chambers with water level measurements – On-site validation of a CFD-based methodology Flow Measurement and Instrumentation 35 39-43

[16] Chen Z, Han S, Zhou F and Wang K 2013 A CFD Modeling Approach for Municipal Sewer System Design Optimization to Minimize Emissions into Receiving Water Body Water Resour Manage 27 2053–2069 https://doi.org/10.1007/s11269-013-0272-9

[17] De Marchis M, Freni G and Napoli E 2013 Modelling of E. coli distribution in coastal areas subjected to combined sewer overflows Water Sci. Technol. 68 (5) 1123–1136 http://dx.doi.org/10.2166/wst.2013.353

[18] Pollert J 2013 Ecological aspect in combined sewer overflows chamber design Water Practice & Technology 8 No 3-4

[19] Mohsin M and Kaushal D R 2017 Three-dimensional computational fluid dynamics (volume of fluid) modelling coupled with a stochastic discrete phase model for the performance analysis of an invert trap experimentally validated using field sewer solids Particuology 33 98-111

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.