Numerical Modelling of Enhancing Suspended Solids Removal in a CSO Facility

Cheng He,* Jiri Marsalek and Quintin Rochfort

National Water Research Institute, Environment Canada, P.O. Box 5050, Burlington, Ontario L7R 4A6

One of the most common methods of combined sewer overflow (CSO) treatment is by conventional settling, the efficiency of which depends on the properties of solid particles and the characteristics of the flow transporting these solids. Since the geometry and hydraulics of CSO facilities are often very complex, traditional design methods based on many simplifying assumptions may not predict well the actual operational performance. Therefore, there is a growing need for new tools assisting engineers in design or operation of CSO storage and treatment facilities. In a study of such a CSO facility, a commercial computational fluid dynamics (CFD) model (FLUENT) was used to investigate flow and sediment behaviour, and to explore the ways of optimizing the overall facility performance by adding flow conditioning baffles to improve particle settling. A two-stage approach was adopted; flow patterns were simulated first by means of a volume of fluid (VOF) model and subsequently formed a basis for simulating particle transport by the discrete phase (DP) model. The simulation results for water surface, flow fields in different structure configurations, and particle capture rates in three parallel CSO storage/treatment tanks are presented for various flow conditions.

Key words: CSO storage and treatment, numerical modelling, CFD, hydrodynamics, sediment removal

Introduction

Combined sewer overflows (CSOs) represent a major cause of deterioration of water quality in receiving waters and impairment of beneficial water uses (Weatherbe and Sherbin 1994). Although during the past 50 years all new sewers in Canada were built as separate sewers, CSOs can be found in older areas serviced by combined sewers and the abatement of CSO pollution remains to be one of the priorities of all large Canadian cities with combined sewers (XCG 2004). The means of CSO abatement are varied but can be classified into four broad categories: (a) control of inflow of stormwater into combined sewers, (b) sewer separation (requiring many other measures as well), (c) CSO storage and/or treatment (serving to increase the system collection efficiency and treat stored flows at central or peripheral wastewater treatment plants), and (d) increased collection efficiency by improved system operation (e.g., real-time control).

In Canadian practice, CSO treatment options have received increased attention of municipal engineers in recent years, particularly retention treatment basins (RTBs) serving to store and treat CSOs by settling (XCG 2004). In RTBs, the efficiency of treatment depends on CSO settleability and the hydraulic conditions in the basin (Li et al. 2003). The hydraulics of RTBs has been traditionally considered in a simplified manner, by addressing the “lumped” properties of the basin, typically described in general terms by the surface loading rate (SLR) defined as the inflow divided by the surface area of the sedimentation basin (Metcalf and Eddy 2003). During the past 10 to 15 years, research has shown that the efficiency of CSO settling can be affected by the settling basin hydraulics, and consequently, research into basin hydraulics was conducted using computational fluid dynamics (CFD) modelling (Saul and Ellis 1992).

Saul and Ellis (1992) reported that the internal geometry of an on-line CSO storage tank influenced the efficiency of settling of a particular wastewater. The length to width ratio, the longitudinal and benching gradients, and the dry weather flow channel were the most significant geometrical properties affecting sedimentation. Other CFD applications to CSO hydraulics problems followed. Svejkovsky and Saul (1993) used the 3D FLUENT model to model the StormKing hydrodynamic separator, Pollert (1999) modelled a CSO overflow structure with free surface flow, and Hrabak et al. (1999) evaluated the general hydraulic performance of a CSO side weir structure. Harwood and Saul (1999) listed the advantages of CFD modelling, which include: (a) no need for laboratory test facilities, (b) the CSO structure geometry in CFD tests can be changed quickly, thus avoiding the time and costs associated with reconstructing a physical model, and (c) flow parameters, such as pressure and velocity, are calculated at all points within the facility, which may not be attainable in physical models.

The numerical simulation of particle transport in CSO facilities is even more useful because of difficulties with modelling particle transport in physical models. Stovin (1996) and others (Stovin and Saul 1998, 2000; Adamsson et al. 2003) have examined the effect of CSO storage chamber configurations on gross solids...
retention efficiency, the prediction accuracy of particle removal rate in a settling chamber under various wall boundary conditions, and simulated flow and sediment deposition patterns. Pollert and Stransky (2003) recently combined hydrologic and hydrodynamic models to evaluate the efficiency of suspended solids separation in CSOs.

The objective of this study was to investigate the applicability of CFD modelling in CSO treatment by examining the feasibility of enhancing particle settling in an existing CSO storage/treatment facility using flow conditioning baffles. For this purpose, a commercially available CFD model, FLUENT, was chosen recognizing that other models capable of simulating flow behaviour and particle transport would also be applicable. To reduce computing time, the flow hydrodynamics and particle transport were simulated separately, because the interaction between particles and the carrier can be neglected for particle concentrations less than 1000 mg/L (Adamsson et al. 2003). Flow conditions were calculated using a two-phase volume of fluid (VOF) model, which includes the influence of air pressure on flow behaviour. The simulated velocity fields were compared with measurements from a physical scale model. In particle transport simulation the continuous particle tracking model was used with a continuous influx of particles into the facility and a constant inflow at all times. The main objective was to compare the particle capture rates under various flow conditions, rather than to obtain the actual particle removal efficiency of the facility. Therefore, the verification of the particle transport model was not required and was considered outside of the scope of this paper.

Methods

This section describes the selection of a CFD model and characteristics of the North Toronto CSO storage/treatment facility, which was investigated in this study.

Numerical Model

A general purpose computational fluid dynamics model capable of solving a variety of complex fluid flow problems was chosen for this study. The mathematical description of the flow consists of the continuity equation and three components of the Reynolds equations. The resulting equations of conservation of mass, momentum, energy and chemical species are solved using a finite difference method employing a control volume. Full details of the CFD software and its use can be found elsewhere (Fluent 2003).

To be able to predict accurately the flow field in an open tank, it is essential to choose a model which can compute the water surface profile and its changes in time. There are several multiphase models which have been developed for this purpose. The VOF model with unstructured mesh was chosen in this study, on the basis of its favourable applicability, computer running time and numerical stability.

The VOF model has been used widely in commercial CFD software for simulating multiphase flows. Its formulation is based on the concept of two or more mutually insoluble fluids (or phases) occupying a computational cell. For each additional phase, a new variable is introduced, describing the volume fraction of the phase in the computational cell. In each control volume, the sum of all volume fractions of all phases equals one. The fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each phase is known at each location. For the VOF model the time step size has to be very small in order to resolve the interface of the two phases with a large density difference. In this study 0.01 seconds was used as the time step in all flow field simulations with a segregated solver (explicit scheme), which would satisfy the criterion of Courant number less than 1.

After obtaining the flow field from the VOF model, the particle transport model was run on the basis of flow information. When considering the influence of turbulence and flow pattern on particle settling, the numerical simulations of suspended particle transport may be classified into two major categories depending on the method used to follow particle movement. The first category treats suspended particles as a group and uses temporal and spatial changes in particle concentrations to describe the particle movement.

The second method, the Lagrangian particle tracking or the Euler–Lagrange approach, tracks individual particle movement by calculating the balance of forces on the particle, which is written in a Lagrangian reference frame. This force balance equates the particle inertia with the forces acting on the particle, and can be written (for the x direction in Cartesian coordinates) as:

$$\frac{d u_p}{dt} = C_D (u - u_p) + g \frac{(\rho_f - \rho_p)}{\rho_p} + F_x$$  \hspace{1cm} (1)

where \(u\) is the fluid phase velocity, \(u_p\) is the particle velocity, \(\rho\) is the fluid density, \(\rho_p\) is the density of the particle, and \(C_D\) is the drag coefficient.

The first term on the right-hand side of equation 1 is the drag force due to the velocity difference between particle and its carrier. The second term represents the force due to gravity. The third term \(F_x\) represents secondary additional forces, such as the Brownian force for submicron particles, the Saffman’s lift force, or lift due to shear, the force required to accelerate the fluid surrounding the particle, and other forces.

When a particle reaches a physical boundary (e.g., a wall or inlet boundary), FLUENT applies a discrete phase boundary condition to determine the fate of the
trajectory at that boundary depending on the particle properties, particle velocity before and after it contacts the solid wall, and its impact angle.

The Lagrangian particle tracking method was used in this study, since it considers more forces in calculation of the particle movement and usually gives a better prediction of particle movement than the first method, but with higher computing times. The Lagrangian particle tracking approach assumes that the suspended particles are spherical and do not interact with each other. Even though the actual combined sewer solids might not be discrete and spherical, it was felt that this approach would still give good insight into the hydrodynamics of solids inside the settling tanks. In coupled simulation, the flow field and particle trajectories are calculated simultaneously. However, this computing-intensive approach is only used to simulate the transport of particles at high concentrations (≥1000 mg/L), which affect the continuous phase flow field. In CSO flows, suspended particle concentrations are typically well below the limit of 1000 mg/L (e.g., an influent concentration of 250 mg/L was used in this study), and consequently, a faster uncoupled approach is preferred. Thus, the earlier calculated flow field and particle physical characteristics are used to determine particle trajectories in an uncoupled simulation.

The dispersion of particles in the fluid phase due to turbulence can be predicted using a stochastic tracking model. The stochastic tracking (random walk) model includes the effect of instantaneous turbulent velocity fluctuations on particle trajectories. At the same time, particles do not directly impact the generation or dissipation of turbulence in the continuous phase.

**CSO Facility Studied**

The North Toronto (NT) CSO storage/treatment facility (S/TF) was built in 1924 and expanded in 1991. The schematic layout of the facility is shown in Fig. 1. In wet weather, CSOs released from an adjacent combined sewer enter the facility inlet channel, and continue through four connecting pipes into a distribution channel, and via inlet weirs into three storage tanks. The inlet weirs are arranged step-wise at three different levels, so that storage tanks are filled sequentially, starting with the most downstream one (Tank 1), and small events are stored in one or two tanks, reducing the need for, and cost of, additional tank cleaning. During the CSO flow through the settling tank, some suspended particles will sink to the tank bottom due to gravity settling. Overflow from the CSO tanks, via the downstream outlet weir, is conveyed by the effluent channel into a stormwater tank where it mixes with discharge from a storm sewer. The individual tanks are about 43 × 7 × 4 m (length × width × depth) with a bottom benching and slope toward the upstream end of the tank.

Since the study addressed flow characteristics and particle transport in the three tanks, numerical modeling focussed only on the orifices of the four feed pipes, the three tanks, and an upstream section of the effluent channel, rather than simulating flows in the whole facility (see Fig. 2). This simplification, compared to modeling the entire facility, greatly reduced the computing times. For clarity, the only grids shown in Fig. 2 are those on the vertical walls. There are about 188,014 nodal points and 168,275 unstructured elements used in the numerical calculation. The unstructured mesh pro-
vides an exact geometrical match for the sloping bottom of the three CSO tanks.

Results

Simulations of the Existing CSO Facility Operation

The simulated and measured flow fields in all three CSO storage/treatment tanks are shown in Fig. 3a and 3b, respectively. The measured velocity field was obtained from a physical scale model (1:11.6) of the facility built in the hydraulics laboratory of the National Water Research Institute (He et al. 2002). Both panels in the figure show similar flow patterns and clearly indicate that flows in the inlet zone of the three tanks are very unevenly distributed. In both outside tanks, the main flow advances along the outside wall, with a strong lateral rotation in the inflow zone, which can be seen more clearly in the flow field with streamlines displayed in Fig. 4. Particularly in Tank 3, the streamlines reflect between two sidewalls with upward and downward motion. In the middle tank, flow enters the tank along both sidewalls. Low flow velocities in some parts of the tank inlet reduce the effective width of the tank inlet, resulting in increased flow velocities in certain regions. The reason for flow paths being partially (hydraulically) blocked at the tank inlet can be attributed to strong turbulence generated in the distribution channel because of fast inflow impacting on the downstream wall of the distribution channel. The turbulence consisted of eddies of varying sizes which blocked the flow entry.

The flow rate distribution among the three tanks was computed and found to be $T_1:T_2:T_3 = 1.08:1.04:1$, which agrees with the measurements in the physical model (He et al. 2002).

A highly turbulent flow in the distribution channel enters the CSO tanks with a non-uniform velocity distribution across the three tank inlet weirs, and at some locations, strong downward and rotational velocities are generated in the tanks. These flow features may produce velocity shear, flow turbulence and strong bottom shear stress in tanks, which impair solids settling. In fact, such flows may even resuspend unconsolidated sediment deposits on the tank bottom. Consequently, numerical simulations were carried out to explore the feasibility of modifying flow patterns in the CSO tanks and thereby increasing the effectiveness of the facility in removing solids by sedimentation.

Simulations of a Modified CSO Facility

One commonly used method for improving flow patterns is the use of flow conditioning baffles. An
improved settling of suspended solids in the individual CSO tanks should be facilitated by a uniform velocity distribution in the inflow zone near the water surface. This flow pattern would produce less velocity shear, flow turbulence and bottom stress, and also, after particles move out of the surface flow layer by gravitational force, most of them should continue to sink without being entrained by ambient flow. Numerical modelling was used to test various baffle designs, which would help achieve this ideal flow pattern. For brevity, only the most promising baffle layout among those tested is discussed in this section.

As illustrated in Fig. 5, flow conditioning baffles were placed at the inlet to the three CSO tanks. The main part of this arrangement was a $2 \times 7$ m (length $\times$ width) horizontal baffle attached to the inflow weir crest and extending into the tank. The purpose of this baffle is to force inflow to enter the tank horizontally along the water surface. On top of this baffle, there are two or three vertical baffles mounted which divide the tank inlet width into several smaller channels and thereby make the flow distribution uniform in the lateral direction. Because the flow patterns in tanks 1 and 3 were similar, the design of baffles in these two tanks is identical. The length of the three longitudinal baffles increases from the inner wall to the outer wall, as shown in Fig. 5. Furthermore, the position of (and hence the flow intercepted by) these baffles can be adjusted in the longitudinal direction to control flow distribution. In tank 2, flow enters along both sidewalls, and consequently the configuration of the baffles is different. Two baffles with a specific angle of attack (see Fig. 5) were used to restrict the inflow near both sidewalls and enhance the inflow through the middle section.

Tentatively, the following baffle designs were proposed:

- Tanks 1 and 3: 3 baffles, 2 m high, and 4.2, 3.2 and 2.6 m long, respectively.
- Tank 2: 2 baffles, 2 m high, and 2.13 m long, rotated by 20 degrees.

Simulated flow patterns in the CSO tanks with the proposed baffles are shown in Fig. 6. By comparing these patterns with those for the case without baffles (Fig. 3), it is apparent that flow conditions were significantly improved. In particular, at all tank inlets, inflow from the distribution channel is more evenly distributed in the lateral direction, the sinking and rotational flows

![Fig. 4. Simulated velocity in the three CSO tanks displayed with streamlines. Flow rate distributions through the vertical cross sections in the middle of each CSO tank were calculated.](image)

![Fig. 5. Proposed arrangement of flow conditioning baffles.](image)

![Fig. 6. Improved flow fields in the three CSO tanks after adding flow conditioning baffles.](image)
were greatly reduced, and more uniform flow velocity fields were achieved in all three tanks.

The simulated turbulent kinetic energy distribution in the three CSO tanks, with and without baffles, is displayed in Fig. 7a and 7b, respectively. As expected, with baffles, turbulent kinetic energy of flow in tank inlet zones was reduced because of improved flow conditions. However, it can be also noted in Fig. 7b that the turbulent kinetic energy in the middle tank, referenced longitudinally, is greatly increased. This requires further investigation. Figure 8 shows a longitudinal cross-section of flow velocity vectors in the middle section of tank 3, and provides an explanation for high turbulent kinetic energy generated in the middle of the tank—the strong sinking current. The full velocity field plot, displayed with streamlines in Fig. 9, indicates that flow enters the CSO tank and continues to move along the water surface until it reaches the middle of the tank, and then starts to sink to the tank bottom along a relatively narrow path.

The reason for the inflow following the water surface for over half a length of the tank needs to be addressed, i.e., whether it is caused by the horizontal baffle projecting into the tank. In order to answer this question, the flow field was recalculated for the same baffle arrangement but with the horizontal baffles removed. Figures 10 and 11 show simulations of the flow turbulent kinetic energy distribution and flow velocity in the three CSO tanks without horizontal baffles. Comparisons of Fig. 10 to Fig. 7b, and Fig. 11 to Fig. 9 show that the results are very similar, which indicates that the horizontal baffles have minimal influence on the flow patterns after the addition of vertical baffles to the inlet weir structure, and the inflow follows the water surface even without the horizontal baffles. An explanation for this phenomenon follows from the fact that the vertical baffles break up the lateral rotational motion at the tank inlet, which prevents generation of strong upward and downward vertical velocities and reduces dissipation of the kinetic energy, as described earlier for the existing design without vertical baffles (see Fig. 3 and 4). After adding vertical baffles, the strong inflow produces a counter-clockwise eddy in the

**Fig. 7.** Comparison of turbulent kinetic energy distributions in the three CSO tanks for two structural configurations: (a) strong turbulent kinetic energy in the tank inlet zones before adding baffles, and (b) reduced turbulent kinetic energy in the inlet zones after adding flow conditioning baffles.

**Fig. 8.** The velocity profile for a cross-section along the middle axis of tank 3 shows strong downward velocities in the middle zone of the tank after adding the flow conditioning baffles.

**Fig. 9.** Streamlines showing the flow movement through the CSO tanks from right to left. Two opposite rotating eddies force the flow to sink to the bottom along the narrow path between them.
upstream half of the tank, with suppressed vertical velocities in the middle of the eddy. However, this strong counter-clockwise eddy will bring down the surface flow, at the eddy downstream end, close to the half-length point of the tank. The bottom flow in the downstream half of the tank then creates a surface clockwise eddy as indicated in both Fig. 9 and Fig. 11. These two eddies rotating in opposite directions force the flow to sink to the tank bottom in a very narrow space between the two eddies, which creates a large velocity gradient and produces relatively strong turbulent energy.

The flow characteristics in the two longitudinal halves of the tank differ greatly, as shown in Fig. 9. In the upstream part, the main flow in the top layer of the vertical eddy should enhance removal of suspended particles, because this flow guides suspended particles to sink to the bottom after they leave the main flow layer. However, the bottom flows in both eddies could scour unconsolidated particle (sludge) deposits on the tank bottom, if they are strong enough. This is of particular concern in the downstream eddy because scoured particles may be brought to the water surface and leave the tank via the outlet weir. The above reasoning explains the simulated spatial distribution pattern of bottom sludge deposits displayed in Fig. 12; there is much more sludge found in the upstream half of the tank than in the downstream half.

Adding flow conditioning baffles has improved the overall flow patterns in the CSO tank, particularly in the upstream half, but further modifications are needed to improve flow patterns in the downstream part of the tank. The efficacy of such modifications should be assessed in terms of improved particle settling. For this purpose, a particle transport model should be applied and model results, with and without baffles, compared.

Sediment transport was simulated by the particle tracking model described earlier. The model was conceptually verified in an earlier study, with a reasonably good agreement between the simulations and the measurements (He 2003). The present study focusses on comparisons of the effects of different flow conditions on particle settling and the simulation accuracy of absolute particle removal rates is of secondary interest.

After the flow field has been simulated, particle tracking simulations can be done in two ways: (a) a discrete release of one group of particles in the beginning of the simulation run and the tracking of their trajectories, and (b) a continuous release of particles at the inlet and tracking their trajectories. The first method has some limitations. Because the number of particles released from the inlet is limited by the number of mesh cells at that point and the particle tracking is a stochastic process, repeated tracking runs are necessary to obtain a sufficiently representative sample of the infinite number of simulated outcomes. Also, a very large number of particle tracking time steps have to be used to ensure that the particle tracking is not prematurely aborted, i.e., before the particle reached the bed or the outlet (assuming no resuspension). The second particle simulation method reflects better the actual operation and assessment of the performance of settling basins (i.e., continuous influx and exit of particles). Furthermore, an earlier
study confirmed the advantages of the continuous release/tracking of particles method (He 2003), which was therefore adopted in this study.

The choice of tracked particles was somewhat arbitrary in this investigation of the tracking methodology and could be further improved in the future to closely reflect the properties of sludge found in the settling tank. Two particle sizes, 5 and 50 µm, with density of 2800 kg/m³ (approximating natural sand), were simulated in the particle transport study. At each time step, a total of 76 particles, distributed evenly over the surface of the four inlet pipes and corresponding to a suspended particle concentration of about 250 mg/L, were continuously released into the tank. In an attempt to balance computing time and particle settling time, 3600 time steps were applied in all particle transport simulations, with the time step of 0.5 s and 30-min duration of the whole simulation period. A reflective boundary condition, with a constant rebounding coefficient of 0.7, was specified for the tank bed and walls and used to follow particle trajectory after the particle reached a solid surface.

Figure 13 shows the simulated percentage of total particles trapped in the settling tank during a 30-min simulation period. A number of observations can be inferred from the data in this figure. Firstly, even after 30 min, equilibrium has not been reached; thus, some incoming particles are still being just dispersed in the tank water, which contributes to high removals in the early parts of the simulation. In future testing, the equilibrium state indicated by a constant rate of particle trapping (i.e., invariable in time) should be achieved by extending the simulation duration. When comparing the corresponding pairs of runs with and without baffles, the baffle presence improves the rate of trapping by about 6% (relative to the initial concentration), for both particle sizes. This improvement may seem small, but recognizing that it is achieved at relatively low costs, it would be worthwhile to adopt the structural modification producing this beneficial effect.

Conclusions

A CFD model has been found effective in assessing flow conditions in an existing CSO storage and treatment facility. A VOF (volume of flow) submodel of the software package was used to simulate flow patterns in the CSO tanks, and the simulated flow field compared well with that observed in a scale model of the same facility. The results show that the flow is very unevenly distributed in the existing tanks, especially in the tank inlet zone, where the main flow advances along the side walls with strong lateral rotation. Such conditions impair solids settling in the tank and should be corrected by flow conditioning baffles. For this reason, a baffle arrangement consisting of a horizontal baffle extending into the tank was attached to the inlet weir crown, and supported three vertical baffles. Computer simulations showed that this new arrangement improved the lateral distribution of flow in the individual tanks, eliminated rotational flow components, and greatly reduced the turbulent kinetic energy in the inlet zone. The effects of this arrangement on solids settling were investigated using a particle transport model based on the Lagrangian particle tracking method. Two particle sizes, 5 and 50 µm, with density of 2800 kg/m³, were used in CFD simulations. Results showed that the modified tank with flow conditioning baffles captured about 6% more of incoming particles than the existing design. The numerical simulation has demonstrated the feasibility of analyzing or optimizing the CSO facility performance by CFD modelling. Compared to field observations in the prototype and recognizing limitations of scale modelling, CFD modelling offers an attractive alternative for optimizing the existing CSO storage/treatment facilities. The methodology proposed in this study can be readily applied to other similar installations.

Acknowledgements

The support received from the Great Lakes 2020 Sustainability Fund, the City of Toronto staff Patrick Chessie and Sandra Ormonde, and the NWRI Research Support Branch staff Bill Warrender, John Cooper and Brian Taylor, is gratefully appreciated.

References


Received: July 27, 2004; accepted: August 30, 2004.