Three-dimensional flow measurements and CFD modelling in a storm-water tank

Mesures et modélisations tridimensionnelles de l'écoulement dans un bassin d'orage

Matthieu Dufresne*, José Vazquez**, Abdelali Terfous*, Abdellah Ghenaim*, Jean-Bernard Poulet*

* Institut National des Sciences Appliquées de Strasbourg, 24 bd de la victoire 67084 Strasbourg cedex France (matthieu.dufresne@insa-strasbourg.fr).

** Ecole Nationale du Génie de l'Eau et de l'Environnement de Strasbourg, 1 quai Koch BP 61039 67070 Strasbourg cedex France.

ABSTRACT

The flow field is the primary control on sedimentation in storm-water tanks. Particle image velocimetry and acoustic Doppler velocimetry are employed to measure the 3D mean velocity and the turbulent kinetic energy in transversal planes of a storm-water tank. The measurements are compared to the simulated results of the computational fluid dynamics (CFD) software Fluent. When no overflow occurs, a large non-symmetrical circulation dominates the flow field. When an overflow occurs, the flow field is divided into a clockwise circulation on the right side of the tank and a counterclockwise on the left side. These results will be used to study, in the near future, the solid transport.

RESUME

La sédimentation en bassin d'orage est principalement contrôlée par l'écoulement. La vélocimétrie par imagerie particulaire et la vélocimétrie acoustique Doppler sont utilisées pour mesurer le champ de vitesse 3D moyen ainsi que l'énergie cinétique turbulente dans des plans transversaux d'un bassin d'orage. Les mesures sont comparées avec les résultats d'une modélisation Fluent. Quand aucun déversement n'a lieu, l'écoulement présente une grande boucle de recirculation asymétrique. Dans le cas contraire, il présente deux boucles de recirculation symétriques. Ces résultats seront utilisés pour étudier le transport solide en bassin d'orage.

KEYS WORDS

Acoustic Doppler velocimetry, Computational fluid dynamics, Particle image velocimetry, Storm-water tank.

INTRODUCTION

Since the European Directive 91/271/CEE (21 May 1991) was implemented in France, the treatment of wet weather wastewater flows has been imperative in order to protect receiving water quality. But during rainfall events, in a combined sewer system, the wastewater treatment plant cannot continuously treat all the flows generated. A storm-water tank may temporarily store the influents and send them towards the treatment plant. Moreover, most of the pollution in wet weather flows is carried by particles that tend to easily settle from the water under quiescent conditions (Ashley et al., 2004). Thus, a storm-water tank can also be used to treat wet weather pollution by settling. The behaviour is as follows. Low liquid discharge (dry weather) that enters into the tank goes directly to the outflow and then to the treatment plant. Overflow can occur when liquid discharge is so high (wet weather) that the tank is full of water. In this case, the liquid discharge is divided into two parts: the first part goes to the outflow; the second part goes to the overflow and is then discharge into the receiving river. But the lack of knowledge about flow and especially pollution in tanks remains a big obstacle for their design.

Measurements are probably the best way to understand the behaviour of flow and pollution but they cannot be made before the tank has been built. Computational fluid dynamics (CFD) has been shown to be effective to describe the mean flow velocity and the sediment transport in small (Stovin and Saul, 1996, 1998) and large (Adamsson et al., 2005) physical models of rectangular tanks equipped with one inlet (pipe) and one outlet (pipe or weir).

The aim of the whole study is to investigate the use of CFD for the design of the storm-water tank about flow and pollution (Terfous et al., 2006). The concentrations are sufficiently low so that the solid transport doesn't influence the flow field. That is why we can study (both experimentally and numerically) firstly the flow field, which is the primary control on sedimentation (Stovin and Saul, 1994), and secondly the solid transport. This article only deals with the flow field.

Particle image velocimetry (PIV) and acoustic Doppler velocimetry (ADV) are employed to measure the mean flow velocity and the turbulent kinetic energy in transversal planes of the storm-water tank. The experimental results are compared with CFD simulations.

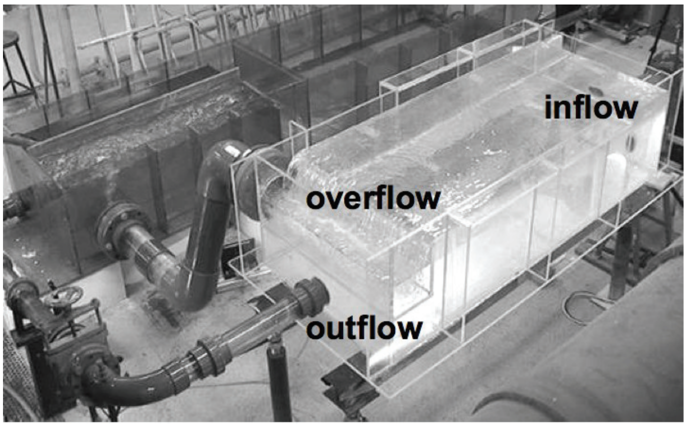

Figure 1. Photograph of the storm-water tank.

1 EXPERIMENTAL DEVICE

1.1 Storm-water tank

The pilot is a rectangular basin of 1.80 m length and 0.76 m width. It is equipped with an inlet pipe (diameter 80 mm), an outlet pipe (80 mm) and an overflow limiting at 0.40 m the water depth (figure 1). A pump outputting between 2 and 6 l/s does the replenishment of the tank. A flowmeter measures the inflow rate. Two ultrasonic sensors measure the water depths of the outflow and the overflow. The flow rates are then calculated by using a weir law.

1.2 Particle image velocimetry

The mean velocity is measured in transversal planes of the tank using 3D particle image velocimetry (PIV), which is a non-intrusive measurement technology (Dantec, 2004). Two cameras record the displacement of very small particles in a planar laser light sheet between two very close moments (2.5 ms). 300 couples of images, corresponding to 150 s, are acquired for each camera. The data are treated by image analysis: cross-correlation in small areas (32*32 pixels, corresponding approximately to 5*5 cm^2) and moving-average validation for each couple of images, time-averaged statistics for each camera and finally 3D vector processing using the calibration previously made with a target (black dots on white background).

1.3 Acoustic Doppler velocimetry

The instantaneous velocities are measured at different points of the tank (approximately 20 per transversal plane) using acoustic Doppler velocimetry (Sontek, 2001). Each acquisition lasts 120 s, which corresponds to 6000 measurements (frequency 50 Hz). The use of phase-space threshold despiking filters the data. The 3D mean flow velocity and the turbulent kinetic energy are then calculated.

2 FLOW SIMULATION

The flow field in the storm-water tank is simulated using the CFD software Fluent (Fluent, 2001). The computational grid is built with Gambit; it is composed of 25 000 cells. The mesh is shown in figure 2.

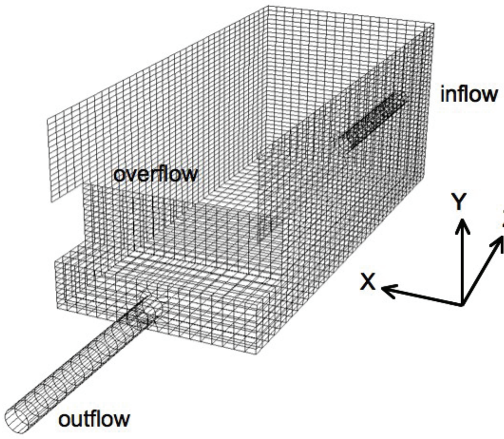

Figure 2. Computational mesh of the storm-water tank for the Fluent simulation.

The three-dimensional Reynolds equations are resolved in the entire domain with the algorithm SIMPLE. The free surface is modelled with the biphasic volume of fluid (VOF) method (Hirt and Nichols, 1981): water and air fractions are computed in each cell of the computational grid using a convection equation; cells where water fraction equals 0.5 define the free surface. The k-ε model describes the turbulence in the tank. The boundary conditions are 'velocity-inlet' for the inlet pipe, 'pressure-outlet' for the outlet (pipe or weir), 'pressure-outlet' for the top of the tank and 'roughness' for the walls and the bed of the basin.

3 RESULTS

3.1 Experiments

Four experiments have been carried out, as shown in table 1. CFD simulations have been carried out for the same conditions.

Experiment	Q_{inflow} (l/s)	$Q_{outflow}$ (l/s)	$Q_{overflow}$ (l/s)
2-0	2	2	0
2-2	2	0	2
6-0	6	6	0
6-6	6	0	6

Table 1. Hydraulic conditions of the experiments.

When no overflow occurs (experiments 2-0 and 6-0), the flow field is dominated by one asymmetrical circulation (figures 3 and 4), as observed by Stovin and Saul (1996) in similar conditions. This circulation can be clockwise or counterclockwise. When an overflow occurs (2-2 and 6-6), the flow field is divided into two symmetrical circulations: clockwise on the right side of the tank and counterclockwise on the left side (figure 5).

3.2 Mean velocity: PIV versus ADV

Figure 3 and 4 show the 3D mean flow velocity respectively measured by PIV and ADV in the transversal plane located 0.456 m away from the inlet for 6 l/s and no overflow (experiment 6-0). The water depth equals 0.17 m. The mean flow velocity cannot be measured by PIV near the free surface because of fluctuations of the water depth, and near the bed because the calibration target we used was not big enough. The ADV sensor measures the velocity at a point located 0.05 m under it and must stay submerged during measurements, so that it cannot measure in the top of the flow.

We see on both figures a counterclockwise circulation with a negative V_Z peak (dark) on the right side (from the inlet to the outlet) and a positive one (light) on the left side (from the outlet to the inlet). Planar vectors (V_X, V_Y) are similar in both cases too. But the velocity amplitude of the right peak equals 0.32 m/s for the PIV and 0.48 m/s for the ADV. Moreover, the right peak is not exactly located at the same position: X=0.08 m for PIV and 0.13 m for ADV. The left peak is located at the same position (X=0.20 m).

The surface area of cross-correlation can explain the difference. The three components (V_X, V_Y, V_Z) are calculated for approximately 5*5 cm^2: Thus, the velocity is space-averaged by the PIV whereas the ADV measures the three components at one point (a small volume of 1 cm^3).

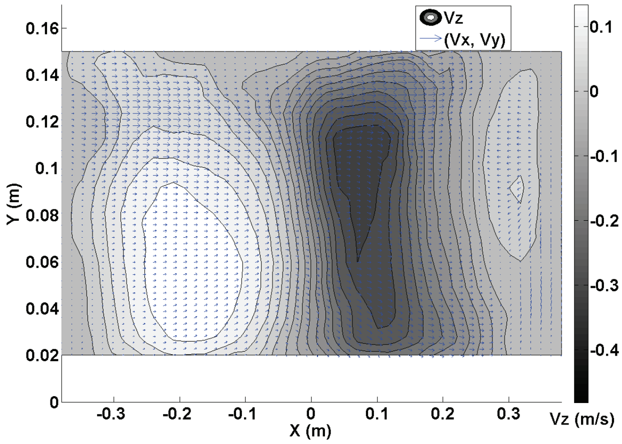

Figure 3. Mean velocity measured by the PIV 0.456 m away from the inlet (exp. 6-0).

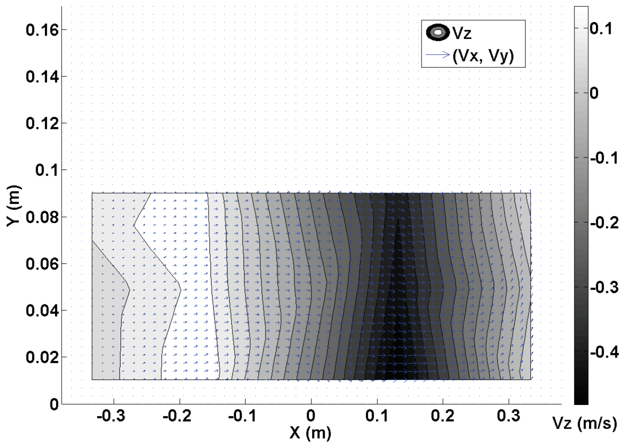

Figure 4. Mean velocity measured by the ADV 0.456 m away from the inlet (exp. 6-0).

3.3 Comparison between experiments and modelling

3.3.1 Mean velocity

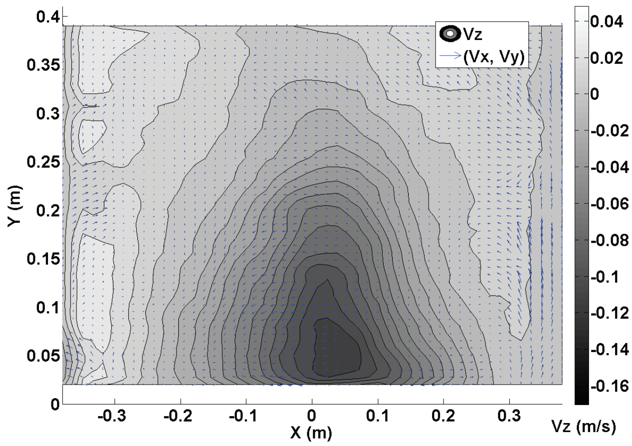

Figure 5. Mean velocity measured by the PIV 0.976 away from the inlet (exp. 2-2).

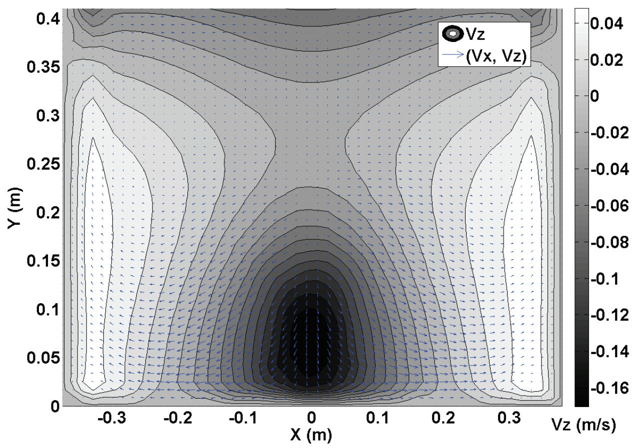

Figure 6. Simulated mean velocity 0.976 away from the inlet (exp. 2-2).

The mean velocity is well simulated for experiments 2-0, 2-2 and 6-6. For the experiments 2-2 and 6-6 (overflow), the velocity field is divided into two symmetrical circulations. For the experiment 2-0 (no overflow), an asymmetrical circulation that can be clockwise or counterclockwise dominates the simulated flow. But for the experiment 6-0 (no overflow), the simulated velocity field is symmetrical (two circulations) whereas the measured one is non-symmetrical. Figures 5 and 6 show respectively the measured and the simulated mean velocity field for the experiment 2-2 in the transversal plane located 0.976 m away from the inlet. The water depth equals 0.41 m. The behaviour is very similar but the values are quite different.

The amplitude of the central negative peak equals 0.11 m/s for the PIV and 0.17 m/s for the simulation. The two positive peaks (left and right side) are non-symmetrical for the measurements whereas they are symmetrical for the simulation; the amplitude of the left peak is 0.05 m/s for the simulation and 0.03 m/s for the measurements.

3.3.2 Turbulent kinetic energy

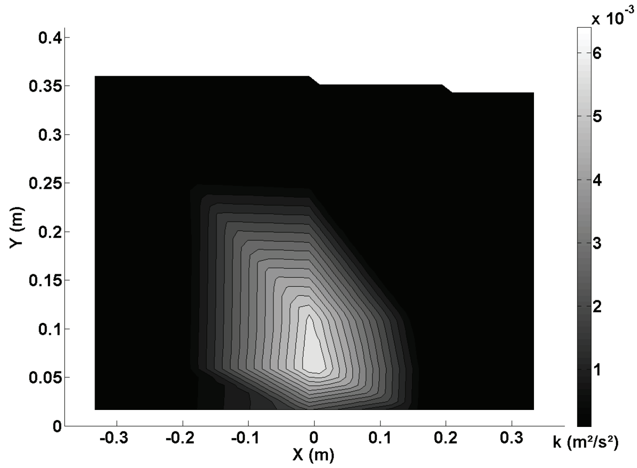

Figure 7. Turbulent kinetic energy measured by the ADV 0.456 m away from the inlet (exp. 2-2).

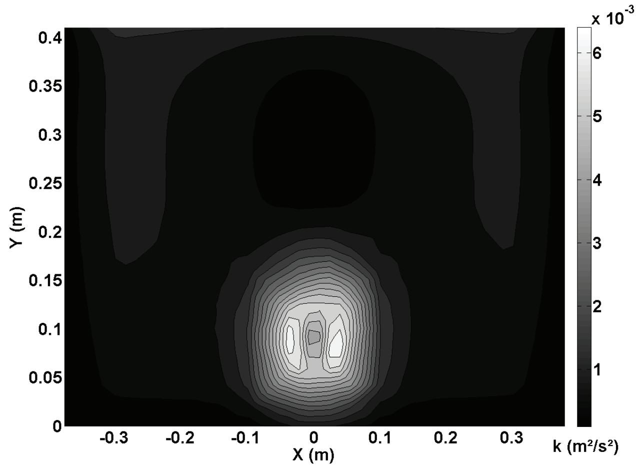

Figure 8. Simulated turbulent kinetic energy 0.456 m away from the inlet (exp. 2-2).

The turbulent kinetic energy k is defined as follows: $k = \frac{1}{2}\left(\overline{V_X'^2} + \overline{V_Y'^2} + \overline{V_Z'^2}\right)$ Where V_i' is the difference between the instantaneous velocity V_i and the mean velocity $\overline{V_i}$:

$$V_i = \overline{V_i} + V_i'$$

The turbulent kinetic energy has been calculated in planes using the ADV measurements and then compared to the simulated values. Figure 7 and 8 show the comparison for a transversal plane located 0.456 m away from the inlet (experiments 2-2). The measured and the simulated values are in good agreement. The central peak equals 0.0060 m^2s^{-2} for the simulation and 0.0056 m^2s^{-2} for the measurements.

4 CONCLUSION

The particle image velocimetry (PIV) and the acoustic Doppler velocimetry (ADV) have been employed to measure the 3D mean velocity field and the turbulent kinetic energy in transversal planes of a storm-water tank. The measurements have been compared to the simulated results of the computational fluid dynamics (CFD) software Fluent.

- PIV is a good technology to measure the three-dimensional mean velocity in a tank but the peaks can be undervalued because of the surface area of cross-correlation analysis.
- When no overflow occurs (which corresponds to low water depth in the tank), the flow field is dominated by one large asymmetrical circulation (clockwise or counterclockwise).
- When an overflow occurs (high water depth), the flow field is divided into two symmetrical circulations: a clockwise one on the right side of the tank and a counterclockwise one on the left side.
- The simulated mean velocity and the simulated turbulent kinetic energy are in good agreement with measurements.

Using these results we intend, in the near future, to study the sediment load phenomena.

BIBLIOGRAPHIE

Adamsson, Å., Bergdahl, L., and Lyngfelt, S. (2005). Measurement and three-dimensional simulation of flow in a rectangular detention tank. *Urban Water Journal*, 2 (4), 277-287.

Ashley, R., M, Bertrand-Krajewski, J.-L. and Hvited-Jacobsen, T. (2004). *Solids in sewers*. IWA Publishing, Scientific and technical report n°14.

Dantec (2004). FLOWMAP 3D-PIV System, Installation & User's guide. Dantec Dynamics.

Fluent (2001). *Fluent 6.0 software User's Guide*. Fluent Inc.

Hirt, C.W. and Nichols, B.D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*, 39(1), 201–225.

Sontek (2001). SonTek ADVField Acoustic Doppler Velocimeter: technical documentation. SonTek/YSI, Inc.

Stovin, V.R. and Saul, A.J. (1996). Efficiency prediction for storage chambers using computational fluid dynamics. *Water Science and Technology*, 33(9), 163-170.

Stovin V.R., and Saul, A.J. (1998). A computational fluid dynamics (CFD) particle tracking approach to efficiency prediction. *Water Science and Technology*, 37(1), 285-293.

Stovin, V.R. and Saul, A.J. (1994). Sedimentation in storage tank structures. Water Science and Technology, 29(1-2), 363-372.

Terfous, A., Vazquez, J., Dufresne, M., Poulet, J.-B., Ghenaim, A., Lipeme-Kouyi, G. and Vasile, C., (2006). Contribution to the 3D modelling of the suspended sediment flow in a storm-water tank by the use of the particle image velocimetry (PIV). International conference Hydroinformatics 2006, Nice, France.