World Applied Sciences Journal 7 (6): 735-743, 2009
ISSN 1818-4952

Numerical and Experimental Study of Velocity Profiles in Sewers

H. Bonakdari

Department of Civil Engineering, University of Razi, Kermanshah, Iran

Abstract: Sound management of sewer networks and minimization of the pollution discharged into receiving waters through combined sewer overflows need in-depth knowledge of the flow rates and pollutant loads conveyed in sewers. A precise knowledge of the pollutant discharge relies on a good assessment of the spatial distribution of the velocities in a cross section, as they are involved both in flow-rate and distribution of concentration. Our research is studying the representativeness of velocity profiles in real sewers by using experimental and numerical tools. This paper presents the experimental site and results before explaining the numerical study that is in progress and comparing the experimental and numerical results.

Key words: Computational Fluid Dynamics · Field measurements · Flow velocity · Sewer

INTRODUCTION

While sewer systems have existed for centuries in many countries, legal requirements are now increasingly stringent as a result of the May 1991 European Community Directive and the January 1992 national water policy law, which stipulates that any town producing a daily pollutant load of more than 900 kg has to be equipped with a wastewater collection network [1]. Moreover the wastewater collection systems have recently been recognised as fully included in the wastewater treatment process. Like any industrial process, wastewater collection networks need measuring means for real-time control of flows, as well as for performance evaluation [2].

Sound management of these networks and minimization of the pollution load discharged into receiving waters through combined sewer overflows necessitate in-depth knowledge of the flow rates and pollutant loads conveyed in sewers. Wohrle and Brombach, [3], have shown that the usual hypothesis about spatial homogeneity is not true. Thus a precise knowledge of the pollutant discharge needs a better assessment of the spatial distribution of the velocities in a cross section, as they are involved both in flow rate and distribution of concentration. Moreover, Ashley *et al.* [4], have pointed out that a large amount of data are needed, covering both dry and wet weather situations.

The availability of high performance digital computers and development of efficient numerical models technique have accelerated the use of Computational Fluid Dynamics (CFD). The control over properties and hydraulic behaviour of fluid flow and relative parameters are advantages offered by CFD which makes it suitable for the simulation of applied problems [5-10].

Since very little data are available for real sewers, the current research project combines experimental investigation inside a real sewer along with numerical modeling. Numerical models offer considerable potential in representing the evolution of flow rates within a sewer network. Thus, a research program is in progress in Razi University, to study how numerical simulations might be used to improve the representativeness of measurements from individual sensors in sewers [11-14]. At the same time mathematical modeling of velocity profiles is developed. This paper presents the experimental site and results before explaining the ongoing numerical and mathematical study of velocity profiles and comparing the experimental with numerical results.

METHODS

Experimental site and set-up: The experimental site is located in an area called "Cordon Bleu", a few kilometers upstream of the treatment plant on the main sewer line of the city of Nantes (in north-western of France). All of the wastewater in the northern portion of the urban district is conveyed through this line, i.e. an effluent of 600,000 population-equivalents. This facility is convenient to the purposes herein for several reasons. A number of

Correspoding Author: H. Bonakdari, Department of Civil Engineering, University of Razi, Kermanshah, Iran
Tel: +98 831 427 4537

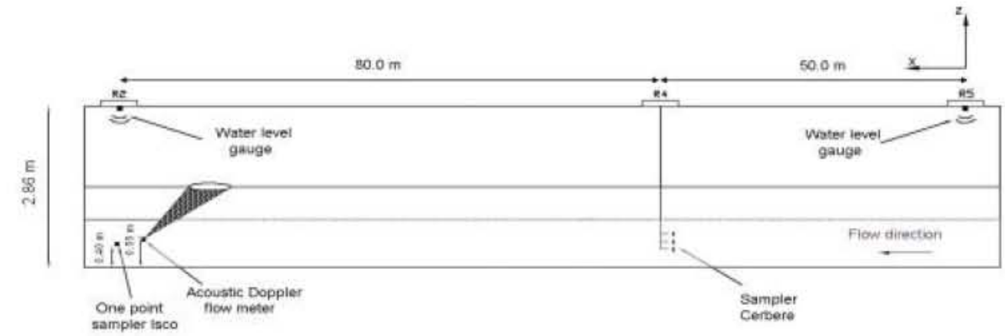


Fig. 1: Instrumentation layout over the experimental area

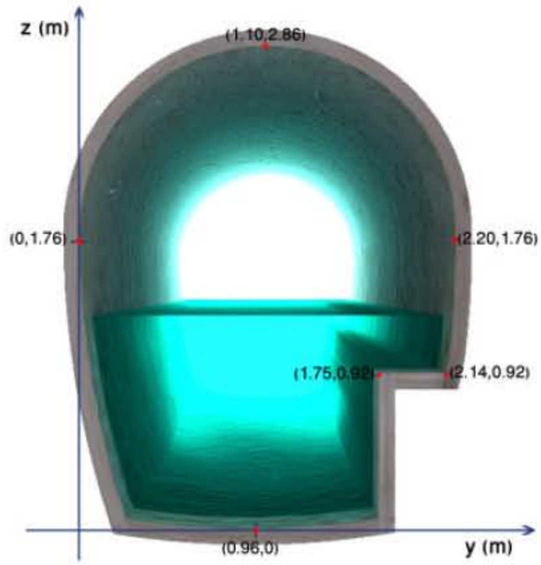


Fig. 2: Cross section of the sewer

Fig. 3: Cerbere the 2D remote-controlled device

sensors have already been implemented (Fig. 1) and the placement provides easy access. In this case the channel is narrow and the flow width (b) to depth (h) ratio is smaller than 5. The cross section corresponds to a narrow and compound section made of an egg shaped channel with a bank. A perspective view of the sewer is shown in Fig. 2 and its geometry and dimensions in the vertical symmetry plane are shown in Fig. 2.

Continuously-monitored Data: One water level gauge is located at the upstream extremity of the site and another is placed 130 m downstream, at the other extremity (Fig. 1). For both devices, the water height is calculated from the time necessary for an ultrasonic signal emitted by the sensor located at the upper sewer invert to reach the free surface and return. Continuous velocity monitoring is carried out using an ultrasonic Doppler flowmeter that indicates the maximal flow velocity (Fig. 1). Data are recorded every 6 minutes.

Velocity Profiles: To investigate the spatial distribution of the velocities in large sewers, Lararrte *et al.*, [15], have developed "Cerbere", a two dimensional remote-controlled device for measuring velocity fields. Cerbere (Figure 3) allows up to three simultaneous measurements in a vertical with three acoustic Doppler velocimeters that are located every 20 cm on a vertical jack. These sensors are remotely-controlled from the ground level. Once velocities have been measured on 3 points at the same time, the carriage is moved 10 cm up to make 3 other measurements and so on. At each point, the water velocity is measured twice with one of the three acoustic sensors. Each measurement represents the mean value of the instantaneous velocity gauged during a 10-second period. This set-up allows to get velocity profiles in a real combined channel either for dry weather or for rainy period as it is possible to scan all the wet area from the bottom to a maximum level of 1.5 m (the horizontal rail is fixed in the 1.5 m of the bottom).

That encompasses roughly 90% of the situations encountered during a given year).

Each result actually represents the mean value of two replicates, with each replicate being the mean value of the instantaneous velocity gauged over a 10-second period. This procedure complies with the NF EN ISO 748 Standard, [1]. Problems result from the drifting material coming into contact with the flow and clogging the sensors. If the difference between two successive measurements taken at the same point exceeds 0.05 m/s, the flowmeter should be removed from the water and cleaned. At the present, 5 to 10 minutes are requested to obtain a vertical velocity profile (depending on water level and glogging, more frequent during high water levels). The local velocities $Vi(t_i)$ are divided by the Manning Strickler velocity at the same time $Vms(t_i)$, in order to make them comparable.

Numerical Study: The numerical computation of fluid flows, generally known as Computational Fluid Dynamics (CFD), which has become a common practice in engineering. CFD models are based on the mass conservation equation and the Navier-Stockes equations for motion. Since most of the practical engineering flows are high Reynolds flows, these equations must be averaged over a small time increment applying Reynolds decomposition, where flow quantities are decomposed in a temporal mean and a fluctuating component. The application of such decomposition results in the Reynolds Averaged Navier-Stokes (RANS) equations, where the turbulence effects appears as a number of terms representing the interaction between the fluctuating velocities and termed Reynolds stresses.

A three-dimensional numerical method is developed to solve flow problems in hydraulic. They have been made possible by advances in computer technology and numerical algorithm development. It solves the three-dimensional turbulent flow equations and utilizes a collocated and cell-centred storage scheme with a finite-volume discretization. The numerical procedure is using CFX software packages for solving three dimensional RANS equations and predicts distributions of velocity and concentration over a cross-section. It involves a finite volume method with tetrahedral meshes. A hybrid scheme is used to discretise the governing equations that are: the continuity equation, the momentum equations and the turbulence model.

The SIMPLEC cell-centered scheme is used to correct the pressure and velocity fields. The boundary conditions are

- at the inlet (upstream) : uniform velocity,
- at the outlet (downstream) : pressure condition,
- on the walls : solid wall condition with no slip and a 2 mm roughness,
- on the free surface: either a roof with free slip condition or a free surface condition based on a volume of fluid method.

The rough wall model is imposed on the solid boundaries and the turbulence kinetic energy is related to the wall shear stress near the wall, as derived from this function. Turbulence dissipation is obtained from the turbulence kinetic energy. The softwares propose various models of turbulence. In order to show their influence, we used the isotropic k-ε and the anisotropic RSM models. The turbulent intensity has been defined a 5% on the inlet.

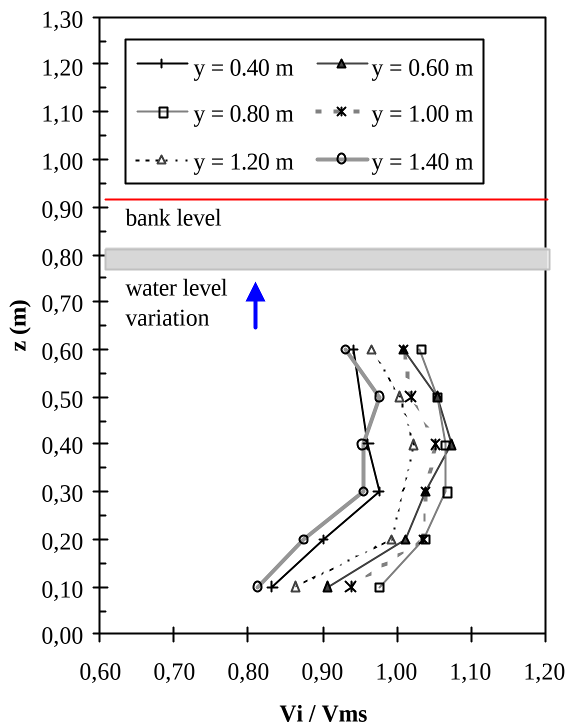
RESULTS AND DISCUSSION

Experimental Results

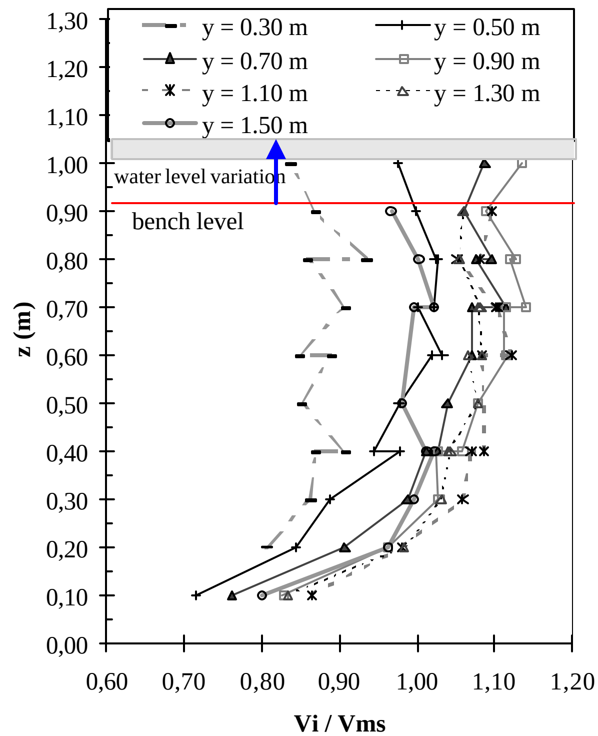
Velocity Profiles over a Cross Section: The sewer used for these experiments has an egg-shaped section with a bank. During most of the dry weather conditions the water level remains under the bank and the section is almost trapezoid, a situation which will be referred to here after as low water situation. At the daily dry weather flow level peak or during wet weather conditions the water level go over the bank and the section becomes compound, a situation referred to as high water situation. Stearns, [16], has already noticed that, in such hydraulic contexts, the maximum velocity is below the free surface unlike the usually accepted situation for river flows. Nezu *et al.,* [17] illustrate when channel is narrow (b/h<5) maximum velocity can be expected below the free surface, as a consequence of the so-called "dip phenomenon".

In all cases the channel is narrow and the ratio of the maximum water level hmax on the width b is between 1.6 and 2.2, which corresponds to a narrow channel. For compound sections Shiono and Feng, [18], show velocity distributions with a maximum velocity located within 0.5 and 0.7 hmax, with strong velocity gradients generated by the step and secondary currents.

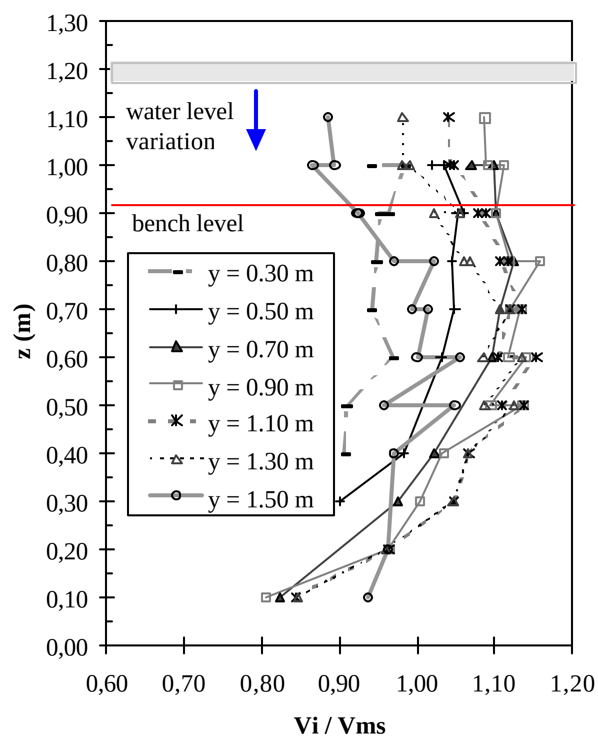
Fig. 4a shows the velocity profiles for a low water day. The profiles present clearly a dip-phenomenon with a maximum velocity located at mid water level. Figure 4b and 4c show high w ater situations with, respectively, an increasing and a decreasing of the water level during the measurements. In both cases the cross section is compound and the water levels change about 5 cm



4a) low water depth conditions

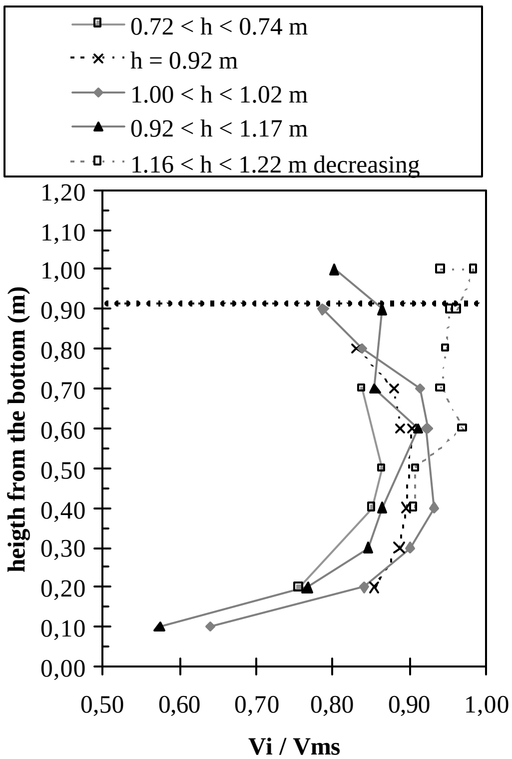


4b) high water depth conditions with compound section and water level increasing

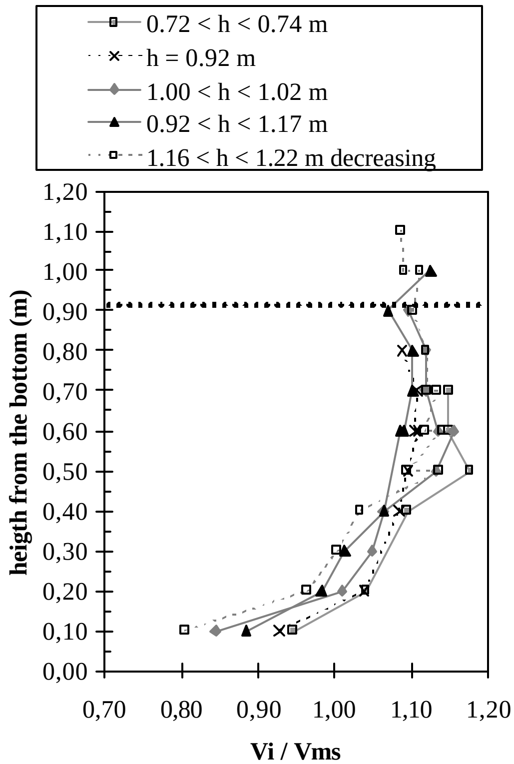


4c) high water depth conditions with compound section and water level decreasing

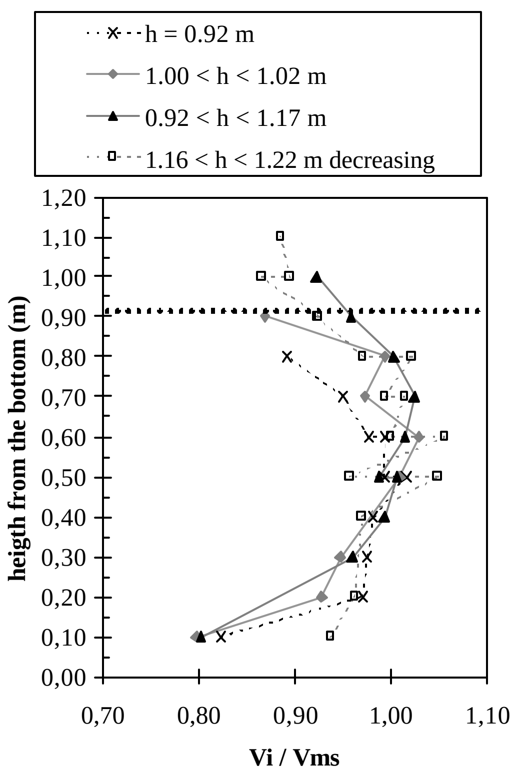
Fig. 4: Velocity profiles in a cross section



5a) y = 0.30 m



5b) y = 0.90 m



5c) y = 1.50 m

Fig. 5: Velocity profiles at a given transversal location

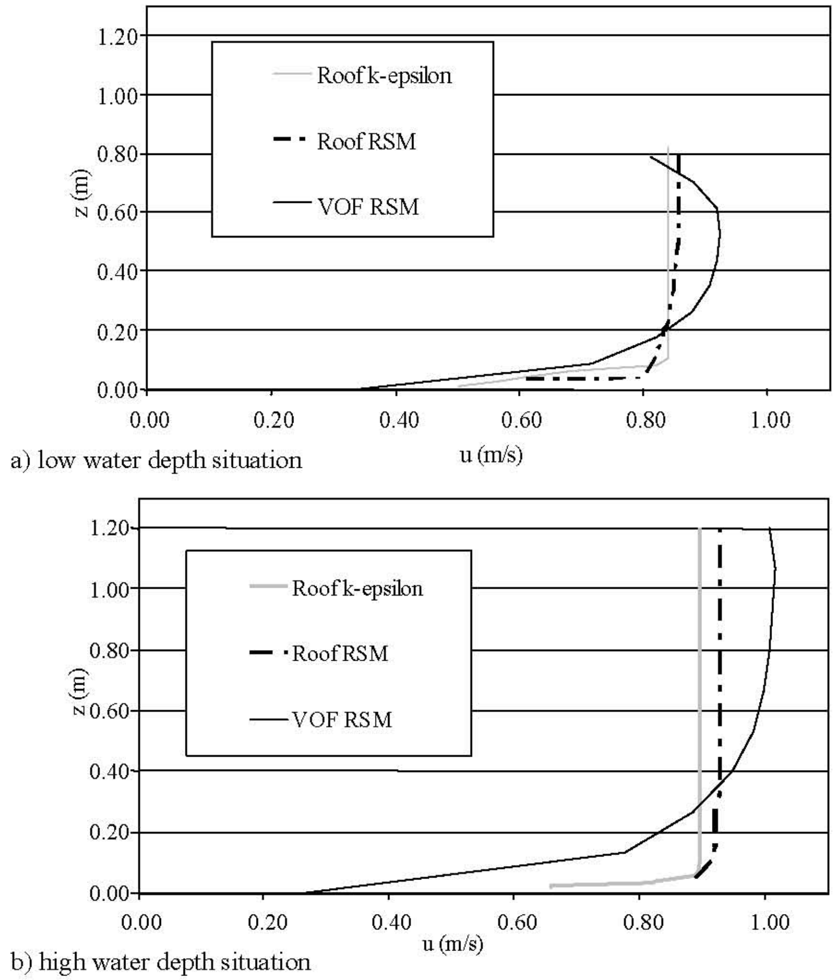


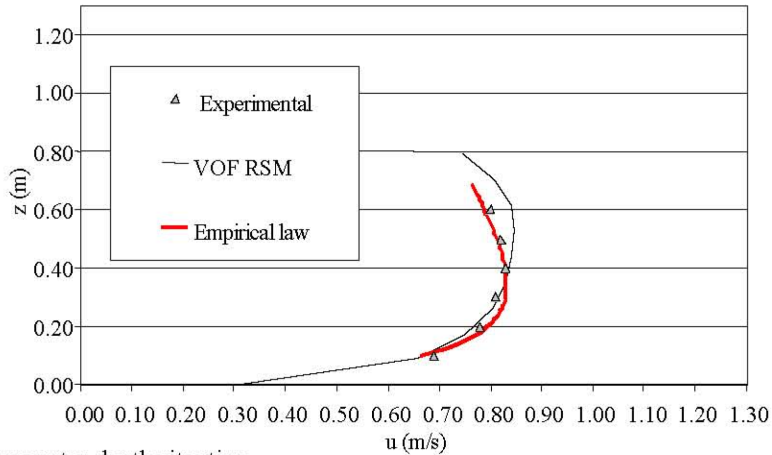
Fig. 6: Influence of the turbulence model and free surface condition

between the first and the last measurements in the whole cross section. It is interesting to notice that the curvature of the profiles changes at the bank level. It was not possible to measure the velocity above the bank but some attempts show a great velocity gradient and great fluctuations close to the wall as can be seen on the profiles at y = 1,5 m. From the manhole it was possible to see big vortices over and near the bank that seems to be similar to the secondary currents presented by [19].

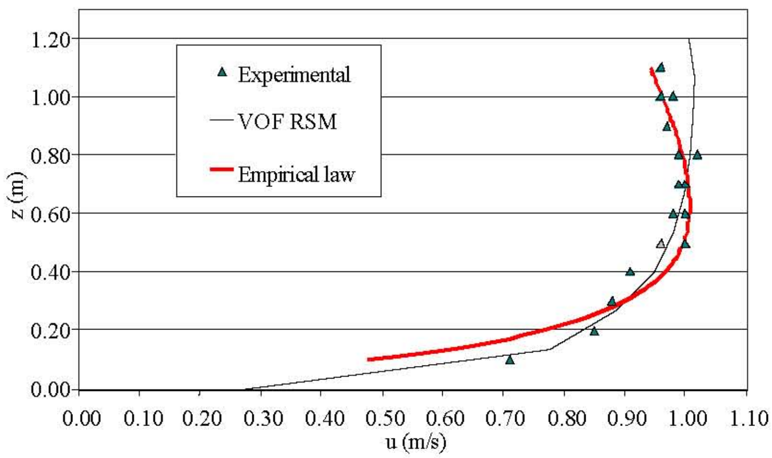
Velocity Profiles on a Vertical: It is interesting to see the evolution of the velocity profiles as a function of the water level hevolution. Very close to the wall at y = 0.30 m (Figure 5a) and close to the bank at y = 1.50 m (Figure 5c), the velocity profiles present a clear dip phenomenon even when the water level increases over the bank level at 0.92 m from the bottom. This phenomenon was not measured when the water level decreases but the velocity measurements were very difficult with strong fluctuations and vortex that could be seen from the manhole. In the middle of the sewer cross section at y = 0.90 m (Figure 5b) it can be seen that the dip phenomenon remains but that it is weaker and the velocity gradient decreases. It is interesting to notice that when the water increases over the bank level the profile presents a curvature inversion whereas when the water level decreases the profiles tend to a logarithmic shape.

Numerical Results

Influences of Turbulence Model and Free Surface: The Reynolds numbers based on the experimental water levels are greater than 10^5 thus we have to take care of turbulent phenomena. Moreover those phenomena are important for suspended solids transport. The numerical tools propose various models of turbulence. In order to show their influences, we used the isotropic k-ε and



a) low water depth situation



b) high water depth situation

Fig. 7: Comparaison of numerical and experimental results.

the anisotropic RSM models. The latter needs much more computational capabilities but it allows to take account the three dimensional aspects of the flow velocity fluctuations. Another issue that should be considered in open channel flows is modelling the free surface.

Even with a stable flow hypothesis, the computations of a real free surface situation (diphase) might be quite time consuming. To avoid that, firstly a roof with free slip condition has been used at the free surface boundary condition. Secondly a free surface condition based on a volume of fluid method (VOF) has been used [20]. Figures 6 shows the influence of the turbulence model and free surface condition on the longitudinal velocity, u, profiles for low (Figure 6a) and the situations with high water depth (Figure 6b) in the middle of the vertical cross section. It can be noticed that the roof condition, whatever the turbulence model is, gives no velocity gradient for z/hmax greater than 0.5, where hmax is the maximum water level and z the height from the bottom. The VOF condition associated with the RSM model computations give more complex profiles with a kind of dip phenomenon with a maximum velocity at 75% of the water level but a 10% over estimation of the velocity. The low water results are not available yet due to convergence problems. The first analysis indicates that this is associated with not fine enough meshes to properly capture the velocity gradient in all the sections.

Comparison of Numerical and Experimental Profile: In order to validate the numerical study, let's compare the numerical results with the experimental ones. For the low water depth situation (Figure 7a) the experimental data can be fitted by the following empirical logarithmic law:

$$\frac{u}{u_*} = aLn\left(z\left(1-\frac{z}{h\max}\right)^b\right)+c \qquad (1)$$

where hmax is the maximum water level during the experiments, z the height from the bottom, $u_* = \sqrt{\frac{\tau}{\rho}}$ is the shear velocity that has been calculated with $\tau = 1$ N/m^2 [21], $\rho = 1000$ kg/m^3, a = 2.8, b = 1.14, c = 33.8 are 3 constants. That law can not be extrapolated to high water depth situations (Figure 7b). The model was well fitted with the experimental data at low water depth where as the data for high water depth situation showed relatively deviation, especially in near free surface. The relative error in high water depth was found to be 5-10%. That confirms the numerical model must be improved with change on meshes, definition of boundary condition and turbulence model.

CONCLUSION

A set of experimental velocity profiles has been collected for an egg-shaped sewer with a bank. It shows the influence of the channel geometry on the velocity with a clear dip phenomenon for low water situations. The first computational results show the influence of the roof condition and turbulence model. But the numerical study gives a quite rough approximation of the experimental results.

That study has to be continued in order to:

- implement a new set-up that allows measurements over the bank,
- pursue measurements under other hydraulic conditions,
- continue computations of the hydraulic field.

REFERENCES

1. AFNOR, 2002. Norme NF EN ISO 748: 50.
2. Schutze, M., D. Butler, M.B. Beck and H.R. Verworn, 2008. Criteria for assessment of the operational potential of the urban wastewater system. Water Sci. and Technol., 45(30): 141-148.
3. Wohrle, C. and H. Brombach, 1991. Probenahme im abwasserkanal, Wasserwirtschaft, 81: 60-65.
4. Ashley, R.M., S. Tait, F. Clemens and R. Veldkamp, 2004. Sewer processes-problems and new knowledge needs, In the Proceeding of the 2004 Urban Drainage Modelling Conference, pp: 195-204.
5. Bradbrook, K.F., P.M. Biron, S.N. Lane, K.S. Richards and A.G. Roy, 1998. Investigations of controls on secondary circulation and mixing processes in a simple confluence geometry using a three-dimensional numerical model. Hydrol. Processes, 12: 371-396.
6. Nicholas, A.P. and G.H. Sambrook Smith, 1999. Numerical simulation of three-dimensional flow hydraulics in a braided channel. Hydrol. Processes, 13: 913-929.
7. Ma, L., P.J. Ashworth, J.L. Best, L. Elliott, D.B. Ingham and L.J. Whitcombe, 2002. Computational fluid dynamics and the physical modelling of an upland urban river. Geomorphol., 44: 375-391.
8. Stovin, V.R., J.P. Grim, A.P. Buxton and S.J. Tait, 2002. Parametric studies on CFD models of sewerage structures. In the Proceeding of the 2002 Urban Drainage Modelling Conference.
9. Lau, S.D., V.R. Stovin and I. Guymer, 2006. The prediction of solute transport in surcharged manholes using CFD. In the Proceeding of the 2006 Urban Drainage Modelling Conference.
10. Bonakdari, H., F. Larrarte and J.B. Bardiaux, 2007. Experimental and computational study of velocity fields in narrow or compound section sewers. Water Practice and Technol., 002: 2.
11. Bonakdari, H., 2006. Modélisation des écoulements en collecteur d'assainissement-Application à la conception de points de mesures. Ph. D. thesis, University of Caen, France.
12. Bonakdari, H., F. Larrarte and C. Joannis, 2007. Effect of a bend on the velocity field in a circular sewer with free surface flow. In the Proceeding of the 2007 International Conference on Sustainable Techniques and Strategies in Urban Water Management, NOVATECH.
13. Bonakdari, H. and A.A.L. Zinatizadeh, 2008, How can computational fluid dynamics improve measuring in real sewers? In the Proceeding of the 2008 International Symposium on Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering, pp: 29-32.
14. Bonakdari, H., 2009. Determination of Doppler flow meters position in sewers using computational fluid dynamics, Flow Measurement and Instumentation. (submitted)

15. Larrarte, F., L.M. Cottineau and D. Bellefleur, 2003. 2D velocity and concentration samplers for particle laden flows. In the Proceeding of the 2003 Eurosensors 17 Conference.
16. Stearns, E.P., 1883. A reason why the maximum velocity of water flowing in open channels is below the surface, Trans. ASCE, 7: 331-338.
17. Nezu, I., A. Tominaga and H. Nakagawa, 1993. Field measurements of secondary currents in straight rivers, J. Hydraulic Engineering, 119(5): 598-614.
18. Shiono, K. and T. Feng, 2003. Turbulent measurements of dye concentration and effects of secondary flow on distribution in open channel flows, J. Hydraulic Engineering, 129(5): 373-384.
19. Bousmar, D. and Y. Zech, 2002. Periodical turbulent structures in compound channels. In the Proceeding of the 2002 River flow Conference: 177-185.
20. Hirt, C.W. and B.D. Nichols, 1981, Volume of fluid (VOF) method for dynamics of free boundaries, Journal Computational Physics, 39: 201-225.
21. Jaumouillié, P., F. Larrarte and V. Milisic, 2002., New devices for 2D sampling of velocities and pollutant concentrations in sewers, In the Proceeding of the 2002 International Conference on sewer process and networks Conference, 171-178.