Conference on Turbulence and Interactions TI2006, May 29 – June 2, 2006, Porquerolles, France

Experimental and numerical study of the wall stress in narrow compound channel

Hossein Bonakdari*,,*** - Frédérique Larrarte*,*****
* Division Eau et environnement - Laboratoire Central des Ponts et Chaussées - route de Bouaye - BP 4129 - 44341 Bouguenais Cedex – France
** Centre de Géomorphologie M2C UMR 6143 CNRS, 24 rue des Tilleuls, 14000 Caen - France
*** Groupement pour l'évaluation des mesures et des composants en eaux et assainissement, 149, rue Gabriel PERI. 54500 Vandoeuvre Lès Nancy,. France.

ABSTRACT

Sewer networks convey waste waters to the treatment plant. The management of deposits in combined sewers is interesting for hydraulic, technical and management reasons. But in practice it is very difficult to measure velocity and wall stress in sewer. Moreover most of the sections are not properly studied geometries such as egg shaped, circular, compound sections. Thus a research program is in progress in LCPC, to study how flows behaviour in such channels and the numerical simulations can be used systematically investigate some parameters. This paper presents the theoretical basis used to calculate the wall stress, the experimental site and the numerical method. For the numerical method, the analysis is based on three-dimensional numerical modelling solving the Navier-Stokes equation extended by different model of turbulence. Then the experimental and numerical results are presented, compared and discussed.

INTRODUCTION

Many sewer situations correspond to free surface flows in narrow compound channels (width/height of water < 5). In such hydraulic situations, side walls produce anisotropic turbulence that provides secondary currents in the cross section, [1] and causes the maximum velocity to appear below the free surface, [2]. Moreover the flows in a compound channel exhibit a complex three-dimensional structure. [3] have highlighted a strong asymmetry of the velocity field in a compound section with bench.

The wall stress is involved in the solid transport. This parameter is related to velocity gradients and Reynolds stresses that are involved in the generation of turbulence, which significantly influence the solid transport. Solid transport has an impact on the capability of the network to carry pollutants to the treatment plant, on sedimentation, on the nature of the pollution being discharged through combined sewer overflow into the natural environment, etc. But, in practice, it is very difficult to measure velocity and wall stress in sewer. Thus, computational fluid dynamics is a useful tool.

This paper presents: (1) the experimental velocity profiles measured in real sewers, (2) the effect of Reynolds stress and turbulence modeling on the numerical results accuracy and (3) the limits of the isotropic turbulence model and the potentialities of the anisotropic turbulence models on modeling the secondary currents in narrow or compound channel.

THEORETICAL BACKGROUND

The wall stress at a solid wall can be calculated by the relation:

$$\tau_w = \rho . u_*^2 \qquad (1),$$

where ρ is the fluid density and u_* the shear velocity.

The shear velocity near the solid wall can be calculated by fitting the vertical velocity profile in the near wall region, (z/h<0.2), by a log-law as follows, [4]:

$$\frac{U_{(z)}}{u_*} = \frac{1}{\kappa} \ln\left(\frac{z}{k_s}\right) + B_s \qquad (2)$$

where u_* is the shear velocity, κ is the constant of Von-Karman, z is the distance from the wall, k_s is the equivalent sand grain roughness and B_s is the constant equal to 8.5 for rough turbulent flow defined as $R_e^* = \frac{u_* k_s}{\nu} > 70$.

The CFD solvers directly calculate the shear velocity using equation (2):

$$-\overline{u_i u_j} + \nu \frac{d\overline{U_i}}{dx_j} = u_*^2 \left(1 - \xi\right) \qquad (3)$$

where i=1, j=3 for the bottom of the channel and i=1, j=2 for the lateral walls, and ξ the relative distance from the solid limit (bottom or lateral wall).

The key term of equation (3) is the first left hand side one that is called Reynolds stress and is related to the field velocity by the Reynolds Averaged Navier-Stokes (RANS):

$$\frac{\partial \overline{U}_i}{\partial t} + \overline{U}_j \frac{\partial \overline{U}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \overline{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \overline{U}_i}{\partial x_j} - \overline{u_i u_j}\right) \qquad (4)$$

where $\overline{U_i}$ are the mean velocity in the x (streamwise), y (lateral) and z (vertical) direction, $\overline{P}$ is the pressure, ν the kinematic viscosity and $\overline{u_i u_j}$ are the components of the Reynolds stress tensor.

The issue for the numerical results is the way the Reynolds stresses ($-\overline{uw}$ for the bottom of channel and $-\overline{uv}$ for the lateral wall) are calculated and that depends on the turbulence model.

The eddy-viscosity turbulence models are based on the assumption of isotropy that means the turbulent Reynolds stresses are related to the mean velocity gradients and eddy-viscosity (ν_t):

$$-\overline{u_i u_j} = \nu_t \left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \qquad (5)$$

where k is the turbulent kinetic energy and δ_{ij} is the symbol of Kronecker (δ_{ij} =1 for $i = j$ and δ_{ij} = 0 for $i \neq j$). The eddy-viscosity is a scalar and hence is the same for all stress components. This latter is not a physical property of the fluid, its value changes from one point to another according to the domain and the type of flow thus it has to be calculated. The two-equations k-epsilon turbulence model has been widely used in numerical simulations due to its simplicity and for various cases the capability of predicting turbulent flows. This model employs two semi-empirical equations for the turbulent kinetic energy (k) and the turbulent dissipation rate (ε). Once the distribution of k and ε are known, the eddy viscosity is calculated from following equation:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \qquad (6)$$

where C_μ is the constant. The Reynolds Stress Models (RSM) provide a more accurate representation of the turbulence because they used anisotropic concept. The unknown Reynolds stress components are obtained directly from the solution of differential transport equations in which they are the dependent variables. Thus these models are more complicated than the eddy-viscosity models. [5] suggested the use of different eddy viscosities in the free surface channel. The gradients of velocities in the vertical and transversal direction are negligible hence the equation (5) can be written:

$$\overline{uw} = -\nu_{tz} \frac{\partial U}{\partial z} \qquad (7)$$

$$\overline{uv} = -\nu_{ty} \frac{\partial U}{\partial y} \qquad (8)$$

where ν_{tz} and ν_{ty} are respectively the vertical and transverse eddy viscosity. Among of different RSM models, our study is based upon the Reynolds stress models of Speziale, Sarkar and Gatski (SSG), [6].

EXPERIMENTAL SITE AND SET-UP

The experimental site used in this study is located on the main sewer line of the city of Nantes (North West France). The cross section corresponds to a narrow and compound section made of an egg shaped channel with a bank Figure 1. The walls are made of good quality concrete and the Manning Strickler's coefficient is evaluated to 70, [7].

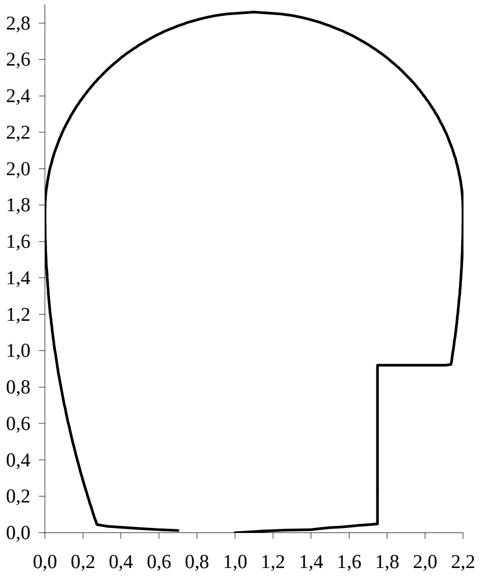


Figure 1: *Cross section of experimental site.*

To investigate the spatial distribution of the velocities in large sewers, a two dimensional remote-controlled device for measuring velocity fields, has been developed. [8] have already presented this set-up that allows to get velocity profiles in a combined channel either for low and high water levels as it is possible to scan all the wet area from the bottom to a maximum level of 1.5 m (that means about 80 % of the annual water levels). 5 to 10 minutes are requested to obtain a vertical velocity profile (depending on water level and clogging, more frequent during high water levels).

RESULTS

Experimental results

Figure 2 shows the vertical velocity profiles in different transverse locations of the cross section (y = 0.5 and 0.9 m) and the fitted curves that have been extrapolated in the near bed zone. It has to be pointed out that, at y = 0.50 the channel bottom is at z = 0.03 m (Figure 1). It can be noticed that a so-called "dip phenomenon" exists as the maximum velocity is below the free surface at a relative level z/h =0.6.

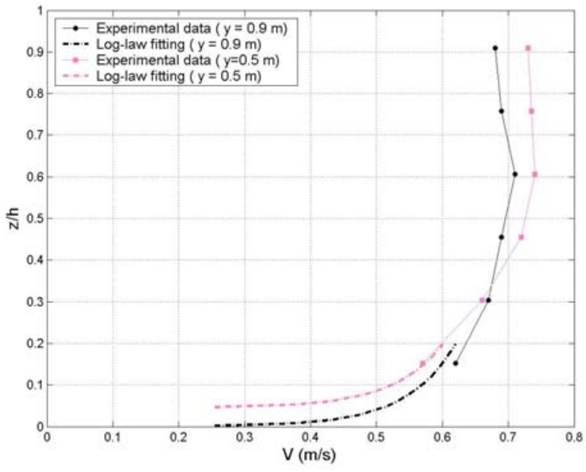


Figure 2: *Distribution vertical of velocity for experimental site in low water level (water level h = 0.66 m).*

For high water condition the water level goes over the walk-way and the section becomes compound. During the measurements, the water level in the sewer decreases by 15 cm. This explains the velocity fluctuations during the measurements . It was not possible to measure the velocity above the walk-way but great velocity fluctuations are notable close to the wall as can be seen on the Figure 3 at y=1.5 m.

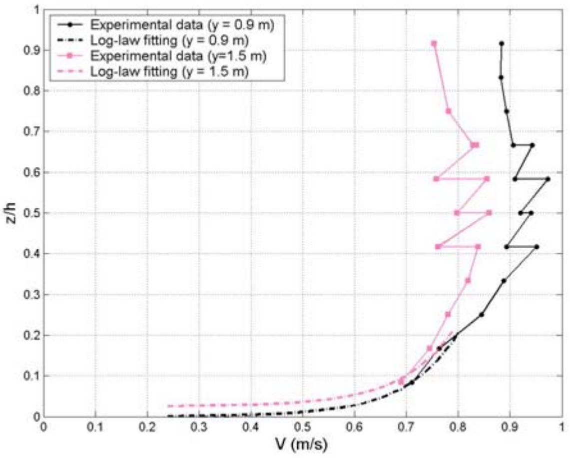


Figure 3: Distribution vertical of velocity for experimental site in storm condition.

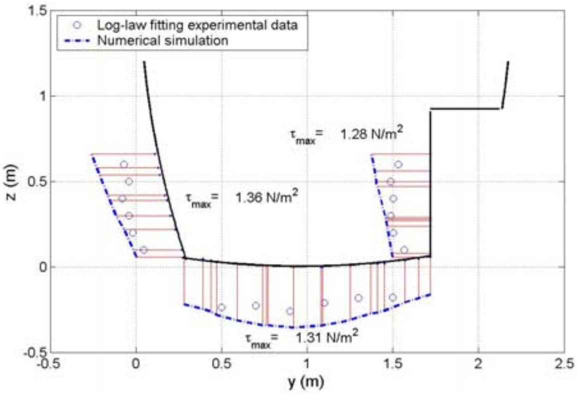
The shear velocity is calculated using equation (2) by fitting a logarithmic law on the measurements. The difficulty is to determine the wall equivalent sand grain roughness k_s that is not exactly equal to the real roughness of the solid surface, [1]. For estimating the equivalent sand grain roughness we have used the two measurements made near the wall and found $1.0*10^{-3}$ and $2.5*10^{-3}$ for respectively low and high water conditions. The shear velocity u_* evaluated with the log-law fitting is 0.03 and 0.04 m/s in respectively low and high water conditions.

Numerical results

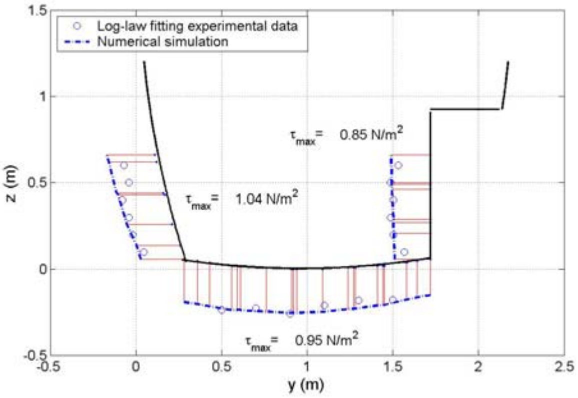
Figure 4 shows the wall stress at the bottom and lateral walls of the channel for low water condition (height water = 0.66 m). It can be seen that the k-epsilon turbulence model over-estimates the wall stress by 30%. On the opposite the RSM presents the same results than the experiments.

Let's make an analysis of the Reynolds stresses components. The computational fluid dynamics with Reynolds stress models show that, for the channel bottom, the secondary currents imply that the maximum value of $\nu_{tz} = 3\nu_{ty}$ hence there is a great anisotropy of turbulence. Therefore, due to this differences, the hypothesis of isotropic turbulence produces the over estimation of shear stress.

Figures 5 and 6 show the ratio $\log\left|\frac{\nu_{tz}}{\nu_{ty}}\right|$ of the vertical and transverse eddy viscosity on the solid wall. For low water, the very strong anisotropy on the vertical wall in the right hand side and on the bottom of the channel are evident. On the opposite this anisotropy is rather small on the curved wall on the left hand side. When the water level increases, the effect of the secondary currents on the bottom weakens. Therefore anisotropy of turbulence in the channel bottom decreases and the two k-epsilon and SSG models show approximately the same results (about 10% difference), Figure 7.



(a) k-epsilon



(b) SSG

Figure 4: *Distribution of shear stress for low water condition.*

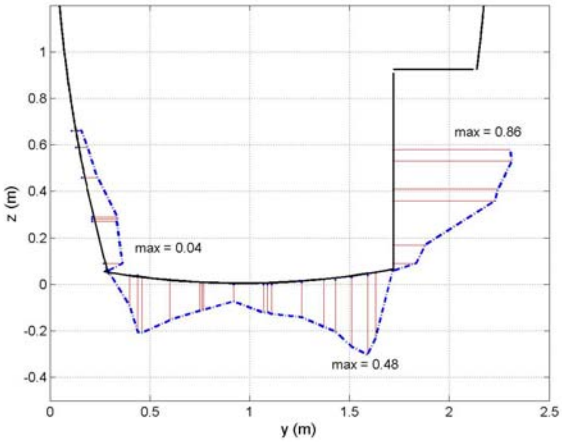


Figure 5 *Distribution of eddy viscosity components ratio (logarithmic scale).*

For high water condition, the section is compound and there is a great velocity gradient at the junction of walk-way with the main channel. This phenomena was shown by measurements of [10] and [11]. In this region there is a great anisotropy of Reynolds stress tensor. This difference explains that k-epsilon turbulence model values are greater than experimental results is this area.

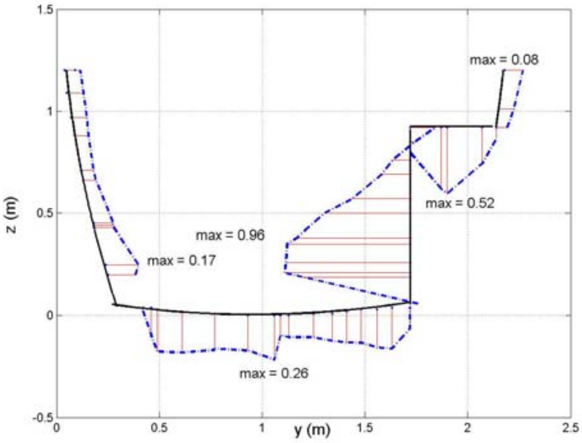


Figure 6: *Distribution of eddy viscosity components ratio (logarithmic scale).*

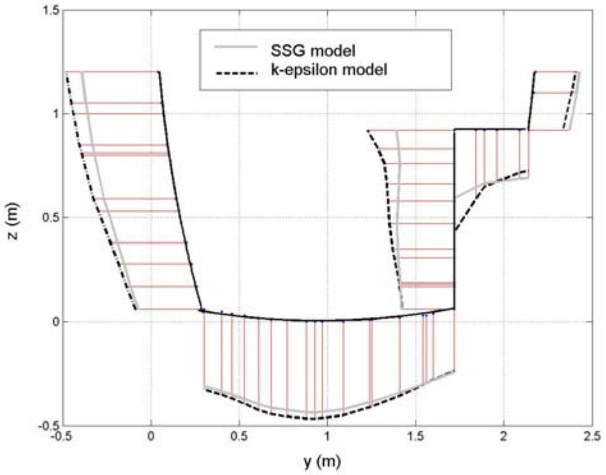


Figure 7: *Distribution of shear stress for high water condition with k-epsilon and SSG models.*

The computational fluid dynamics not only capture the physics of flow (secondary current, gradients) but also it allows to access to parameters that are very difficult to measure such as velocities in the vicinity of the wall, over walk-way in the real site.

CONCLUSION

The shear stress in sewer flow has been studied numerically, with CFD models checked against velocity measurements performed on the field all over the wet section. Calculations show the influence of the turbulence models on the shear stress for low water situations and the effect of the compound section on the velocity gradients. Thus numerical modeling provides some kind of access to data almost impossible to measure in sewer channels. In the future, we will try to investigate the effect of singularity (bend, confluence,…) in the narrow channels and compound sections on the shear stress.

That study has to be continued in order to:

- implement a new set-up that allows measurements over the bank even for high water situations,
- get measurements under other hydraulic conditions or geometry,
- predict the sedimentation in other contexts.

ACKNOWLEDGEMENTS

This work has been supported by GEMCEA, a Public Interest Group for Assessing Measurements for Water and Sewerage. The authors wish to acknowledge the technical staffs of both the Laboratoire Central des Ponts et Chaussées Division Eau & Environnement and the Direction de l’Assainissement de la Communauté Urbaine de Nantes (Nantes metropolis Wastewater Authority) for their valuable contributions to these experiments.

BIBLIOGRAPHY

[1] Nezu I., Nakagawa H., *“Turbulence in open-channel flows”*, IAHR-Monograph, A. A. Balkema Publishers, Rotterdam, The Netherlands, 281 p, 1993.

[2] Naot D., Rodi W., *“Calculation of secondary currents in channel flow”*, Journal of Hydraulic Division, Vol. 108, No.8, pp 948-968,1982.

[3] Bousmar D., Zech Y., *“Periodical turbulent structures in compound channels”*, River flow, pp 177-185, 2002.

[4] Graf W., Altinakar, M. S., “Hydraulique fluviale”, Tome 2, 378 p., PPUR Lausanne, 1996.

[5] Naot D., Rodi W., (1982), Calculation of secondary currents in channel flow, Journal of Hydraulic Division, Vol. 108, No.8, pp 948-968.

[6] Speziale C-G., Sarkar S., Gatski T-B., “Modelling the pressure-strain correlation of turbulence: an invariant dynamical systems approach”, Journal Fluid Mechanics, Vol. 277, pp 245-272, 1991.

[7] Jaumouillié P., Larrarte F., Milisic V., “New devices for 2D sampling of velocities and pollutant concentrations in sewers”, 3rd International Conference on sewer process and networks, Paris, April 2002.

[8] Larrarte F., Bardiaux J.B., Bonakdari H., *“Computational and experimental study of velocity profiles in sewers”*, 10th International Conference on Urban Drainage, Copenhagen/Denmark, 21-26 August 2005, proceedings on CD.

[9] Stovin V. R., Grimm J. P., Buxton A. P., Tait S. J., *“Parametric studies on CFD models of sewerage structures”*, 9rd International Conference on Urban Drainage, Portland, Oregon, U.S.A, 2002.

[10] Tominaga A., Nezu I., *“Turbulent structure in compound open channel flows”*, Journal of Hydraulic Engineering, Vol. 117, No. 1, pp 21-41, 1991.

[11] Shiono K., Scott C. F, Kearney D., *“Prediction of solute transport in compound channel using turbulence models”*, Journal of Hydraulic Research, Vol. 41, No. 3, pp 247-258, 2003.