



HAL
open science

Effect of a bend on the velocity field in a circular sewer with free surface flow

H. Bonakdari, F. Larrarte, C. Joannis

► To cite this version:

H. Bonakdari, F. Larrarte, C. Joannis. Effect of a bend on the velocity field in a circular sewer with free surface flow. Novatech 2007 - 6ème Conférence sur les techniques et stratégies durables pour la gestion des eaux urbaines par temps de pluie / Sixth International Conference on Sustainable Techniques and Strategies in Urban Water Management, Jun 2007, Lyon, France. hal-03248844

HAL Id: hal-03248844

<https://hal.science/hal-03248844>

Submitted on 3 Jun 2021

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Effect of a bend on the velocity field in a circular sewer with free surface flow

Influence d'un coude sur le champ de vitesses dans un collecteur circulaire

H. Bonakdari^{*,***}, F. Larrarte^{*,***}, C. Joannis^{*,***}

* Division Eau et environnement - Laboratoire Central des Ponts et Chaussées route de Bouaye – BP 4129 - 44341 Bouguenais Cedex – France
frederique.larrarte@lcpc.fr

** Département de Génie Civil, Université de Razi, Kermanshah, Iran

*** Groupement pour l'évaluation des mesures et des composants en eaux et assainissement 49, rue Gabriel Péri. 54500 Vandoeuvre Lès Nancy - France

RESUME

Afin de répondre à des objectifs aussi bien techniques que financiers, de plus en plus d'appareils de mesure sont installés en réseau d'assainissement, mais le choix des sections adaptées peut s'avérer délicat car les performances des matériels disponibles sont tributaires des conditions d'écoulements. L'objectif des recherches en cours est de développer une méthodologie d'aide à l'instrumentation des collecteurs d'assainissement. Ces travaux combinent des études numériques et expérimentales permettant de mieux connaître les contextes hydrauliques présents dans les collecteurs. Cet article présente un exemple d'application sous forme d'une étude systématique de l'influence d'un coude sur la répartition des vitesses à l'aval d'un coude à 90°, et ses implications sur une mesure de débit utilisant un capteur de vitesse.

ABSTRACT

More and more measurement devices are implemented in sewer networks for various purposes. Selecting sections for conducting these measurements may prove difficult, as the efficiency of available devices strongly rely on hydraulic conditions. The current research project uses numerical and experimental investigations in order to improve the representativeness of the measurements performed in sewer networks by taking into the specificities. of each particular site Our purpose is to create an aid for designing the instrumentation of sewer sections. This paper presents an application for assessing the influence of a bend on velocity fields and its implications on flow measurements relying on velocity sensors.

MOTS CLES :

Bend, computational fluid dynamics, metrology, velocity field.

1 INTRODUCTION

Flow-rates and pollutant discharge measurements in sewer are presently developing, for a variety of purposes including research, operation, or performance evaluation. If we focus on the topic of flow-rates measurement in large sewers (i.e. with a characteristic size of cross-section larger than one meter), it commonly involves the measurement of velocity, for which several technologies are available. All these technologies imply some kind of spatial sampling of the velocity field. Some constraints must be fulfilled for this sampling to be representative, and able to yield a true mean value over a cross section. Indeed velocity sensors are parts of flowmeters, which embed some software for converting measured velocities into mean velocities. This conversion relies on assumptions on the velocity field, hence the constraints quoted above.

These constraints may be expressed as general recommendations (i.e. straight reach, away from singularities, with uniform gradient and no sediment...), or sometimes as more precise rules (i.e. minimum distance of 20 D downstream of a bend) (Bertrand-Krajewski *et al.*, 2000). They are often hardly compatible with other constraints (purpose of the measuring point, access, security...) and trade offs may lead to a rather loose interpretation of the constraints regarding the sampling of the velocity. Moreover these constraints can in some cases be relaxed, provided a knowledge of the real velocity field is considered for computing the mean velocity.

A methodology for qualifying candidate measuring sections (and designing suitable implementation of sensors, and suitable data processing as well) has been outlined by Larrarte *et al.* (2004). A key point of this methodology is the identification of the status of each candidate measurement site according to three categories: "no problem", "possible problem" and "not suitable". The criterion for this identification is the easiness of prediction of the velocity field for every water depth liable to occur.

This identification could be achieved on the basis of computations, performed on typical configurations, with results expressed as functions of meaningful parameters. This paper is a contribution to this methodology, for the particular case of measurements downstream of a bend, in a circular section.

This paper presents three-dimensional numerical modeling of velocity fields downstream various bends, and investigates the effect of the bend characteristics on the velocity field for various hydraulic situations. It also provides some clues about the distance of a significant downstream influence of a bend according to relevant parameter, as well as guidelines for the implementation of measuring devices.

2 NUMERICAL STUDY

The numerical procedure is using the CFD industrial package CFX for solving the three dimensional Reynolds averaged Navier-Stokes equations to predict the distributions of velocity over a cross-section. It involves a finite volume method with tetrahedric meshes. Simulations have been performed for different values of radius, different diameters D of circular pipes and for several water depths h and velocities (Gradient and roughness appeared as less relevant parameters in preliminary studies).

At the inlet, uniform velocity \bar{U} and water depth h are specified, and sufficient length is provided upstream of the bend to get a fully developed velocity profile. At the outlet, the relative averaged static pressure is maintained equal to zero on all the outlet boundary. At the solid walls, a no slip condition with a 2 mm roughness is applied. In the near wall region, the standard wall function method proposed by (Launder et Spalding, 1974) has been used. That means the velocity components are equal to zero in the walls. Due to the non isotropic behavior of flows in narrow channels where the aspect ratio of the free surface width on the water depth h is smaller than 5 (Nezu et Nakagawa, 1993), our study is based upon the Reynolds stress models of Speziale *et al.* (1991).

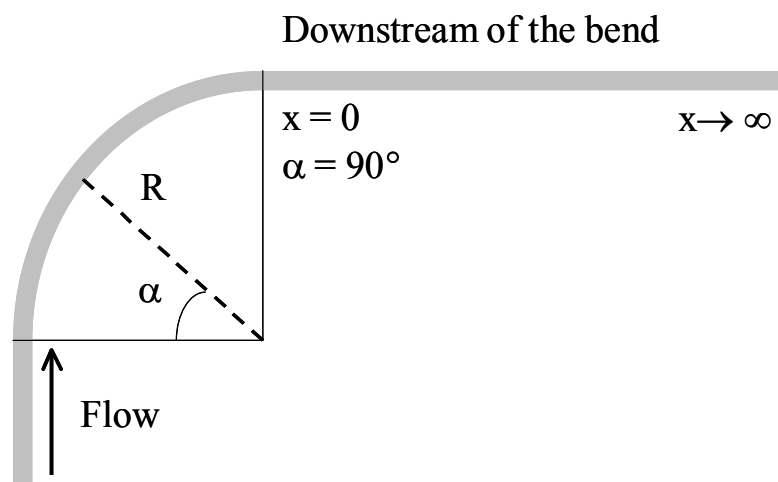


Figure 1 : modelled domain

In order to keep the computational time compatible with systematic investigations, the free surface is modeled as a rigid roof. This simulation does not predict the dissipation term near the free surface and then the dip phenomenon. However that simplification has been used for various reasons:

- the main condition of zero shear stress at the free surface is correctly simulated thus this approximation is used by various authors, Czernusenko and Rylov (2000), Morvan *et al.* (2002), Rameshwaran and Naden (2003) for simulations in open channel flows,
- as long as the flow is not developed, the secondary currents of Prandtl's first kind are predominant and the dip phenomenon does not occur,
- the numerical and experimental results are in good agreement (Larrarte *et al.*, 2004),
- in the developed part of the flow, the difference between the mono and diphasic computations are smaller than 0.5 % for different water depths.

The numerical results show that, downstream of the bend, the flow can be divided in 3 zones:

- a strong heterogeneity zone: a vortex is created by the centrifugal force that generates the secondary currents of Prandtl's first kind. A unique and important vortex exists (Figure 2a). The secondary currents velocity $U_s = \sqrt{v^2 + w^2}$ is about 10% of the maximum velocity in the section upstream of the bend; where v and w are the transversal and vertical components of the velocity,
- a transition zone : the heterogeneity is small but the flow has not recovered is fully developed situation yet. There is still only one vortex but the velocity U_s is about 2 to 3% of the maximum velocity,
- a developed flow zone : the flow has recovered is fully developed behaviour with two vortex turning in opposite directions with a vertical downing component in the middle of the channel (Figure 2b). The velocity U_s is about 2 % of the maximum velocity. The minimum distance necessary for the flow to re-develop is called the re-establishment length L_R is this paper.

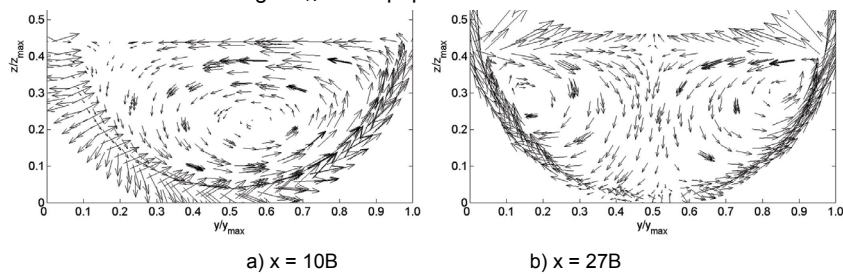


Figure 2 : secondary currents downstream of a bend in a 1.5 m pipe and $h/D = 44\%$

3 SINGULARITY INDEX

To quantify the influence of the bend on the velocity field, a singularity index I_{Sx} has been defined for the S_x section:

$$I_{Sx} = 100 \times \frac{\sqrt{\sum_n (U(x,y,z) - U(\infty,y,z))^2}}{n\bar{U}}$$

where $U(x,y,z)$ is the local velocity in the section located at a distance x downstream of the bend, $U(\infty,y,z)$ is the velocity in the section downstream of the bend where the flow is fully developed, \bar{U} the mean velocity in the section. If this index is calculated between a fully developed flow and a strictly uniform flow, the index I_{Sx} is equal to 3.2% for the water height equal to 0.66 m in the 1.5 m diameter circular collector.

4 RESULTS

Nivus (1999) specifies that in order to obtain reliable measurements downstream of a bend, the sensors have to be implemented at a minimum distance L :

$\alpha \leq 15^\circ$	$L \geq 3*B$
$15 < \alpha \leq 45^\circ$	$L \geq 5*B$
$45 < \alpha \leq 90^\circ$	$L \geq 10*B$

where B is the free surface width.

Figures 3 show the velocity fields at $10 \cdot B$ downstream of a 90° bend for a 1.5 m circular pipe with a given uniform velocity at the inlet:

$$\bar{U} = KR_h^{2/3} i^{1/2}$$

where $K = 70$ is the Manning Strickler coefficient, $i = 4 \cdot 10^{-4}$ m/m the slope, and R_h the hydraulic radius. It can be noticed that the velocity fields are not symmetrical and the maximum velocity U_{max} is shifted towards the outer side of the bend whatever the water depth is. The curved isolines are becoming more and more vertical when the water depth increases.

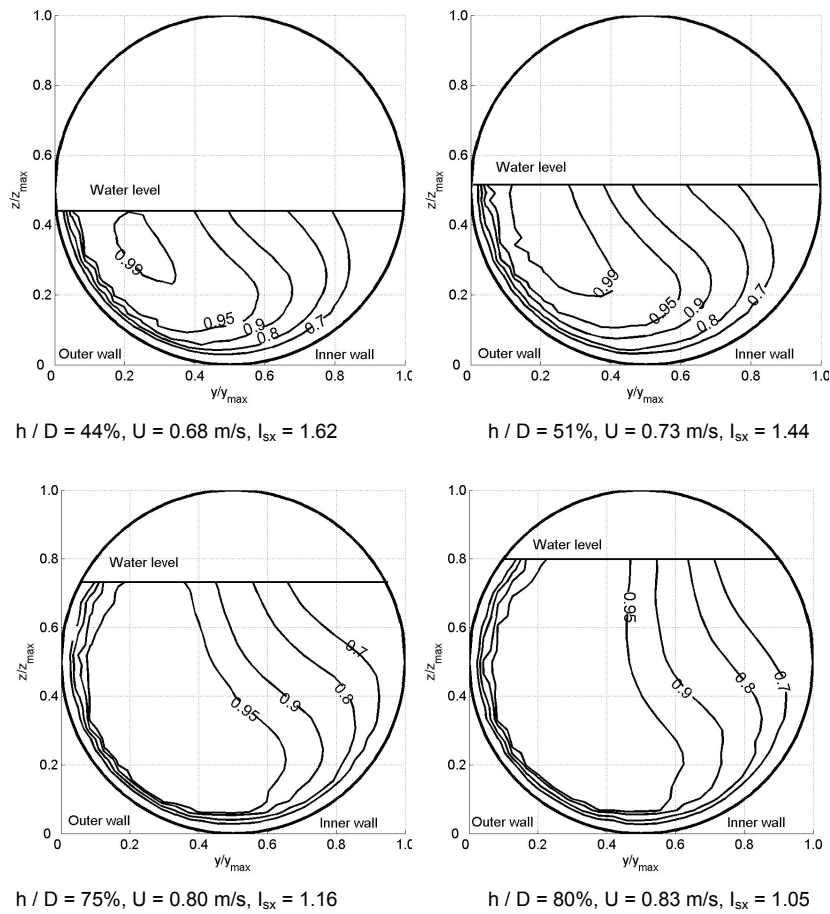


Figure 3 : velocity fields U_i / U_{max} at $10 \cdot B$ downstream of a bend in a 1.5 m diameter pipe

Figures 4 shows that the radius of the bend has a very low influence on the singularity index excepted in a very short distance downstream of the bend. Further downstream, the index tends to a common minimum value of 0.5%. That value should be equal to zero but, due to numerical diffusion, that value is very difficult to obtain. Thus a value of 0.5% has been taken, it correspond to the fact that less than 10% of the cross section has a difference greater than 10%. On the opposite, for a given radius, the diameter of the pipe has a great influence going up to twenty times the free surface width.

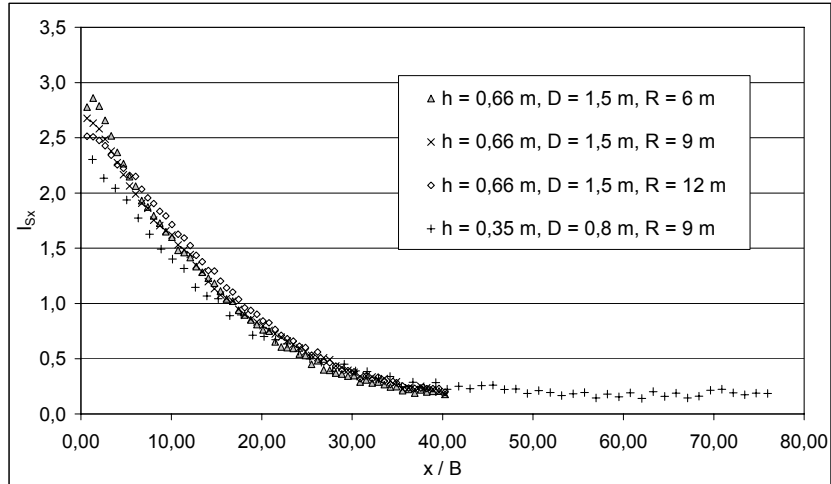


Figure 4 : Influence of the bend radius on the index of singularity for $h/D = 42\%$.

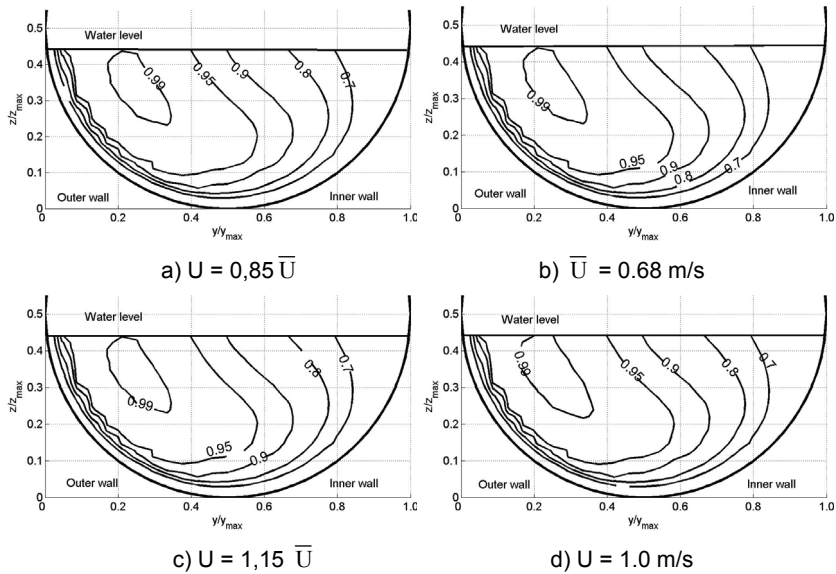


Figure 5 : velocity fields U_i / U_{max} at $10 \cdot B$ downstream of a bend in a 1.5 m diameter pipe

In order to reach some generic rules, it is interesting to investigate the sensitivity of the velocity field to a change in the velocity given at the inlet. Figure 5 shows that there is not any influence on the non dimensional velocity field U / U_{max} . That means that a measurement of the maximum velocity in the cross section and a computation for $\bar{U} = 1 \text{ m/s}$ are able to give the velocity fields for all the pipes as long as the flow is sub-critical (we did not investigate the critical flows yet).

Figure 6 shows that the downstream influence of the water depth increases with the water depth up to a distance of twenty times the free surface width. Beyond this distance, the water depth has only little influence, in the range investigated.

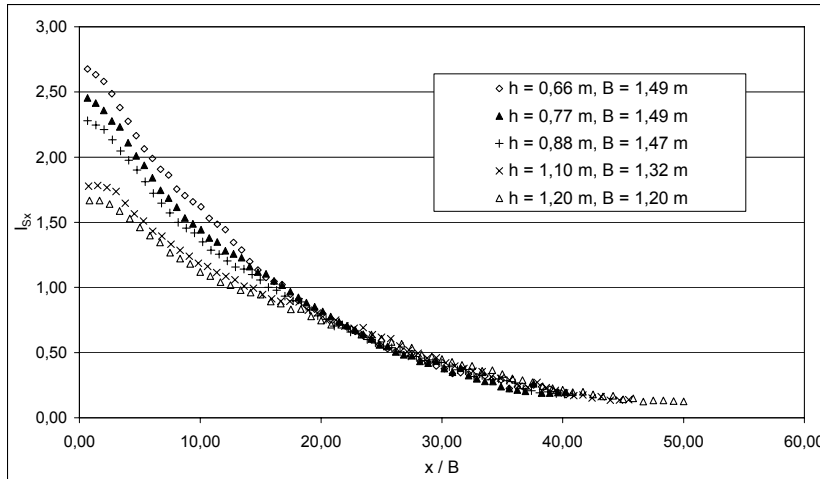


Figure 6 : Influence of the water depth on the index of singularity for 1.5 m diameter pipe and a 9 m bend radius

This can be seen on Figure 7. The ratio of the re-establishment length L_R by the free surface width is a linear function of the of the filling ratio h/D of the water depth by the pipe diameter:

$$L_R / B = 10 (h / D) + 22.5$$

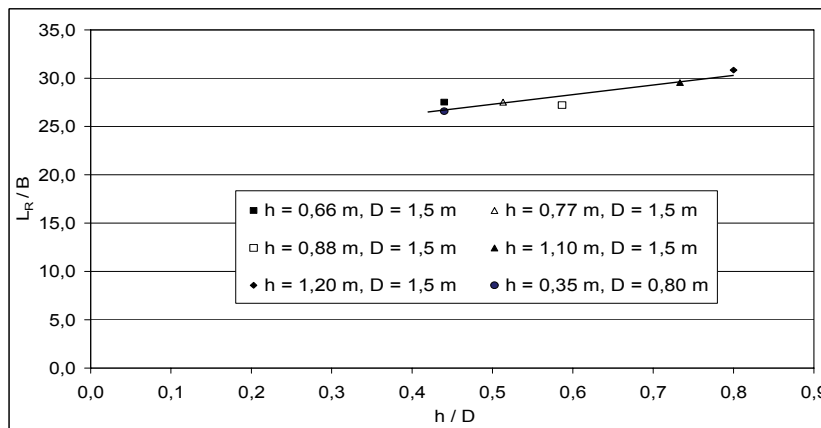


Figure 7 : influence of the filling rate on the influenced length.

5 CONCLUSION

In order to help operators to implement measurement devices in sewers, we develop a methodology based on the potentialities of the computational fluid dynamics. One important step is the modelling of generic situations, both the velocity field in a fully-developed section and the influence of singularities can be assessed in real situations. This information is important for flow-rate measuring devices which rely on velocity sensors, like acoustic Doppler flowmeters.

This paper presents the application to the velocity fields in a circular pipe with free surface flow downstream of a bend. It shows that the radius of the bend and the mean velocity do not have any influence on the velocity field for a particular combination of pipe diameter and water depth. Thus a single computation gives access to a wide range of situations. Moreover, the length influenced by the bend only slightly increases with the filling rate in the range investigated. On the opposite this length is proportional to the width of the free surface.

This study has to be continued to investigate the influence of other singularities and geometries but it already suggests more precise recommendation to implement sensors downstream of a bend.

BIBLIOGRAPHIE

- Bertrand-Krajewski J. L., Laplace D., Joannis C., Chebbo G., (2001), Quelles mesures pour quels objectifs, In : TSM, Vol. 2, pp. 45-56.
- Bradshaw P., (1987), Turbulent secondary flows, In : Ann. Rev. Fluid Mech., Vol. 19, pp. 53-74.
- Czernuszenko W., Rylov A., (2000), A generation of Prandtl's model for 3D open channel flow, Journal of Hydraulic Research, Vol. 38, No. 2, pp. 133-139.
- Larrarte F., Jaumouillié P., Joannis C., (2004), Computational Fluid Dynamics : an aid for designing the instrumentation of sewer sections, Novatech 2004, Tome 1, pp. 729-736.
- Larrarte F., Bardiaux J. B., Battaglia P., Joannis C., (2006), Vélocimétrie Doppler : mise au point d'un protocole d'essai en laboratoire, Techniques, Sciences et Méthodes, génie urbain génie rural, No. 6, pp. 58-65
- Lauder B. E., Spalding D. B., (1974), The numerical computation of turbulent flows, Computational Methods Appl. Mech. Eng., Vol. 3, pp. 269-289.
- Morvan H., Pender G., Wright N. G., Ervine D. A., (2002), Three dimensional hydrodynamics of meandering compound channels, Journal of Hydraulic Engineering, Vol. 128, No. 1, pp. 674-682.
- Naot D., Rodi W., (1982), Calculation of secondary currents in channel flow, In : Journal of Hydraulic Division, Vol. 108, pp. 948-968.
- Nezu I., Nakagawa H., (1993), Turbulence in open-channel flows, IAHR-Monograph, A. A. Balkema Publishers, Rotterdam, The Netherlands.
- Nivus., (1999), Instructions pour le montage de capteurs de vitesse d'écoulement.
- Rameshwaran P., Naden P. S., (2003), Three-dimensional numerical simulation of compound channel flows, Journal of Hydraulic Engineering, Vol. 129, No. 8, pp. 645-652.
- Speziale C. G., Sarkar S., Gatski T. B., (1991), Modelling the pressure-strain correlation of turbulence: an invariant dynamical systems approach, In : Journal Fluid Mechanics, Vol. 277, pp. 245-272.