Conference Paper, Published Version

Bardiaux, Jean-Bernard; Mose, Robert; Vazquez, Jóse; Bonakdari, Hossein; Larrarte, Frederique

Velocity Profiles through a sewer channel: Using CFD to obtain velocity fields

Dresdner Wasserbauliche Mitteilungen

Zur Verfügung gestellt in Kooperation mit/Provided in Cooperation with:
Technische Universität Dresden, Institut für Wasserbau und technische Hydromechanik

Velocity Profiles through a sewer channel:

Using CFD to obtain velocity fields

BARDIAUX Jean-Bernard, BONAKDARI Hossein, LARRARTE Frédérique, MOSE Robert, VAZQUEZ José

Most sewer managers are currently confronted with the evaluation of the water discharges, that flow through their networks or go to the discharge system, i.e. rivers in the majority of cases. For this purpose, we are developing with partners a new sensor, using DOPPLER technology. The apparatus is able to supply a velocity profile and we have to transform this information into a discharge measurement. To obtain this discharge, we have to be able to simulate the velocity field.

The step consists here in determining the modelling method. This paper will present all the results we obtain for these investigations. The use of a particular outfall (which simulates a downstream influence) allows to compare the k-e and RSM turbulent models and also to compare the monophasic approach obtained with the symmetry plane boundary condition and the biphasic one obtained with the VOF (Volume Of Fluid) method. The simulation results will be confronted with data drawn from the literature and also those obtained on our own experimental sites (a real sewer network and a laboratory physical model). In our work, the two discriminating criteria allowing us to evaluate the good approach of modelling are first the characteristic "dip phenomenon" and secondly the representation of the secondary currents.

1 INTRODUCTION

Sewer systems have existed for centuries in European countries, but 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. Moreover the wastewater collection systems have recently been recognised as fully included in the wastewater depollution process. Like any industrial process, wastewater collection networks need measuring means for real–time control of flows, as well as for performance evaluation.

Sound management of these networks and minimization of the pollution 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, 1991) 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, 2004) have pointed out that a large amount of data are needed, covering both low and wet weather situations.

2 NUMERICAL STUDY

2.1 MECHANIC CONTEXT

The numerical study is based on the resolution of the equations of Navier-Stokes through the FLUENT© software. The discretisation takes place while following the finite volume method through a rectangular prism mesh.

The first problem with the open channels remains the free surface. The majority of work carried out in the field of CFD use a symmetry condition to model the behaviour of the medium to the interface water - air. That implies that the free face is not deformed, which is not inevitably the case (Czernuszenko W. and Rylov A., 2002). Some authors also regard the free face as being a wall without friction (Stovin and Saul , 1996). In addition to this way, with an aim of describing the deformation of the free surface (e.g. taking into account of downstream influence), we have to carry out calculations in biphasic mode following VOF method "Volume of Fluid". The field of calculation consists of a water volume and a air volume located in the higher part of the field of study. No constraint will be fixed at the level of the free face which will be able to thus become moved. In this case, the downstream influence will have then to be simulated by the presence of a weir, the height makes it possible to find dimensions of the section of measurement.

The modelling of such a flow implies, in addition, the choice of a model of closing for the equations of Navier-Stokes, i.e. the turbulence model. The scientific community is informed of many models, first or second order, from k-ε to L.E.S.. The softwares propose various models of turbulence, (Stovin et al., 2002) shown their influence on the ability to represent the complexity of turbulent flows. The difficulty is here to translate the strong anisotropy and in order to show their influence, we used the isotropic k-ε and the anisotropic RSM models.

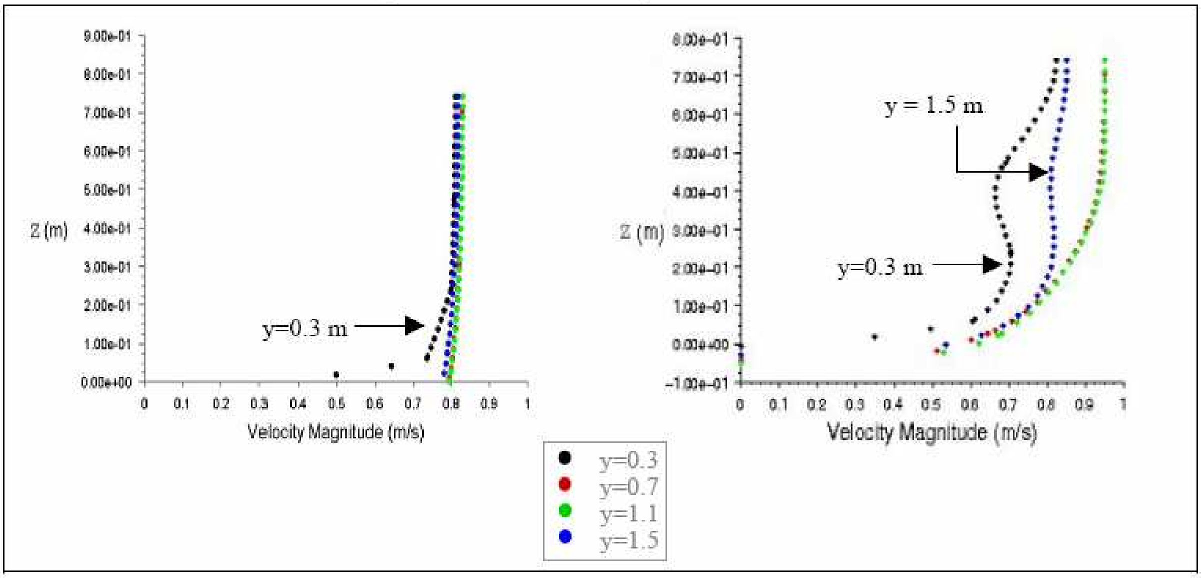
2.2 CROSS COMPARISON AND VELOCITY FIELD

The approach is clearly to cross the interface choice with the turbulence model in order to release a good combination of modelling. All the following simulations correspond to an egg shape with bench geometry (width 1.75 m, height 2.80 m). The wetted cross section has a height of 0.72 m with a mean velocity of 0.80 m/s. The modelled pipe is 50 meters long and the grid counts 200 000 hexahedral cells. The resolution follows a implicit method.

k-ε or RSM

First, we compared the two closure methods through a monophasic approach. The figure 1 shows 4 velocity profiles (from 0.3 m to 1.5 m from the right side of the pipe).

figure 1: velocity field simulated with free surface as symmetry condition with a) k-ε model and b) RSM model

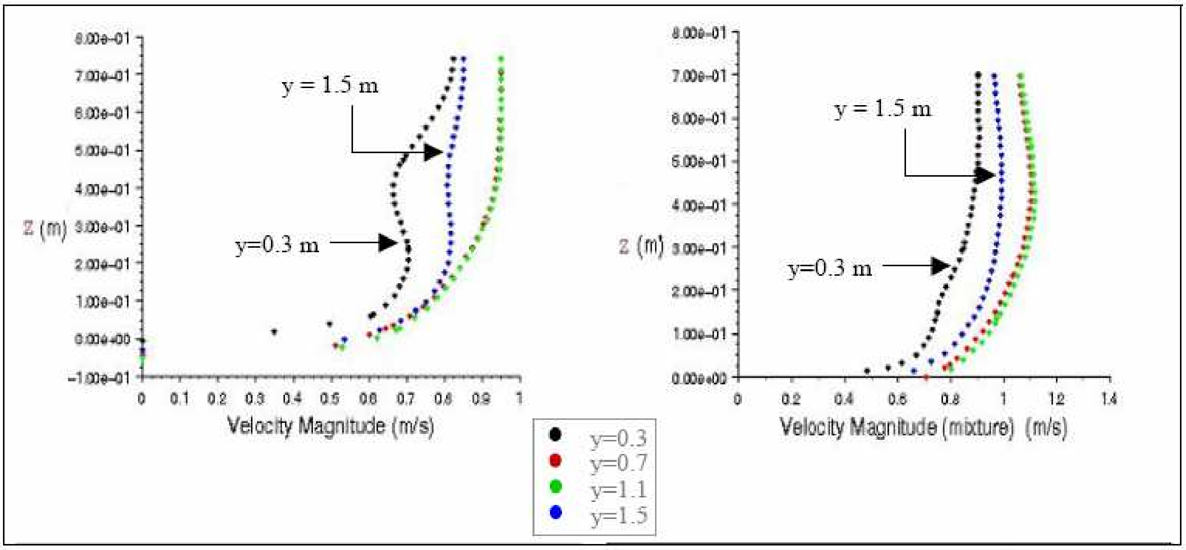


It clearly appears that the RSM model accounts better for the deformations of profile, this is due, obviously, to the anisotropy. But, near the surface, the constraints of calculation (the value of velocity perpendicular to the surface is 0) impose a non-realistic distribution. Indeed, in such narrow channels the dip-phenomenon is expected.

Symmetry or Volume of Fluid

Now, having chosen the way to simulate the turbulence effects, we are going to compare the free surface approaches. Staying in the same package, we used first the symmetry condition and secondly the diphasic method. For the second one, we create a frontal outfall (the weir was 0.45 m high). The figure 2 shows the 4 velocity profiles.

figure 2 : velocity field simulated with RSM model with the free surface as with a) symmetry condition and b) VOF approach



Now, the dip-phenomenon is simulated. This is consistent with the velocity measurements presented by (Nezu and Nakagawa, 1993) and (Naot and Rodi, 1982).

2.3 SECUNDARY CURRENTS

The impact of the turbulence models is also on the ability to represent the secondary currents. In this aim, the works of (Tominaga et al., 1989) have been simulated with FLUENT©.

The experiments were carried out in a 12.5 m length inclined channel with a square cross section of 0.40 m . The bottom as well as the walls of the channel are out of glass ($K_s = 90 m^{1/3} s^{-1}$). A fully developed turbulent flow is established with a section located at 7.5 m of the entry by adjusting in a suitable way the slope and the height of a weir located at the exit of the channel. The geometrical and hydraulic characteristics of the channel are summarized in the following table (Tab. 1)

Table 1 : hydraulic and geometrical characteristic of the open channel

Discharge (l/s)	H_{water} (cm)	width (cm)	$\frac{B}{H_{water}}$	U_{mean} (m/s)	U_{max} (m/s)	Re (*10^4)	Fr	Energy slope (*10^{-3})
7.58	10.15	40	3.94	0.187	0.235	5.07	0.19	0.138

The measurement are shown in (fig. 3); the arrows represent the velocity components perpendicular to the flow direction.

The following pictures [fig.4-a) and 4-b)] present the results extracted from the software, with a k-ε and a RSM model respectively.

figure 3 : secondary currents by (Tominaga et al., 1989)

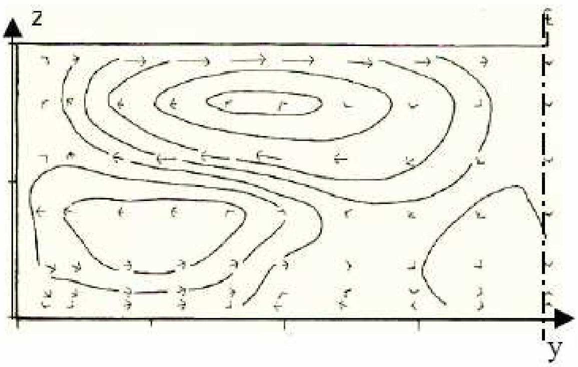
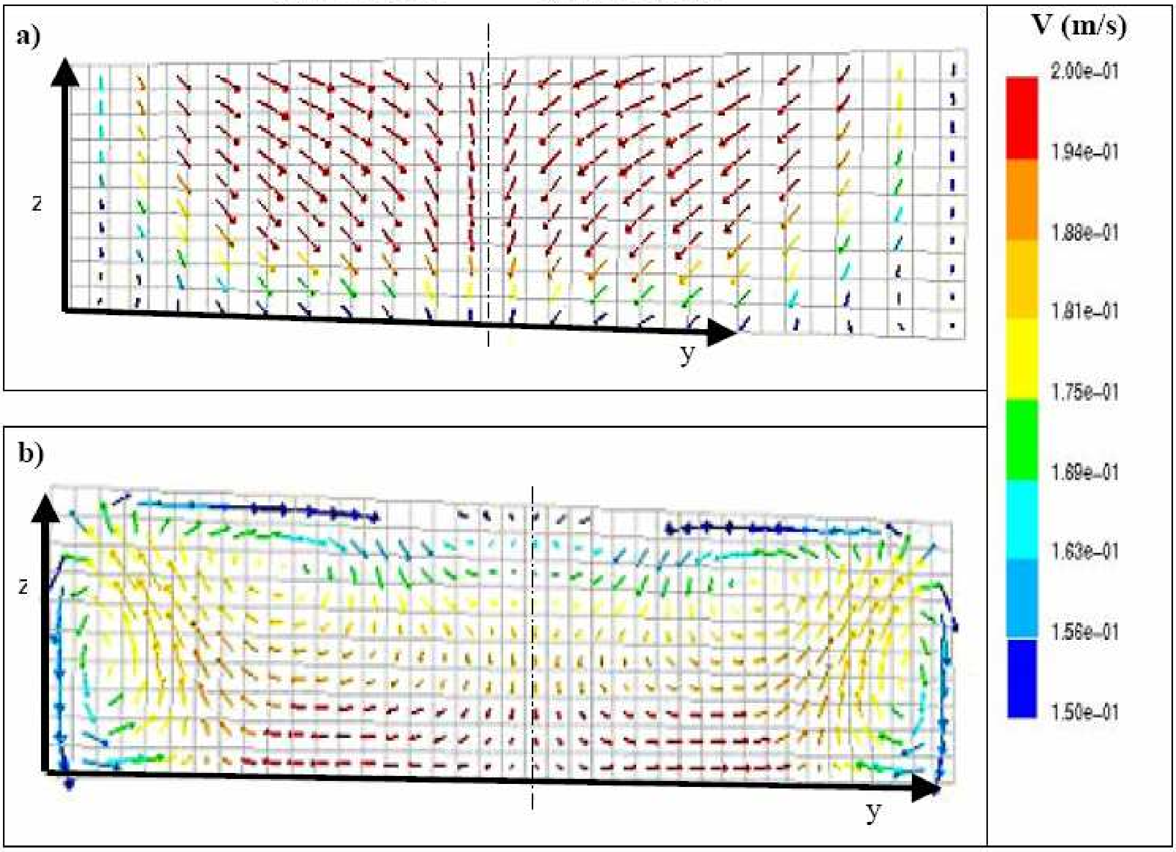


figure 4 : secondary currents calculated with
a) k-ε model b) RSM model



As we did not know all the data, it is difficult to compare the values we obtained. However, it is obvious that a simple isotropic approach cannot generate such currents. The RSM model seems to give the right direction according to the contra rotating re-circulation cells which appear.

3 EXPERIMENTAL VALIDATION

3.1 EXPERIMENTAL SITE

The previous combination (VOF + RSM) had been tested with data coming from an experimental French site.

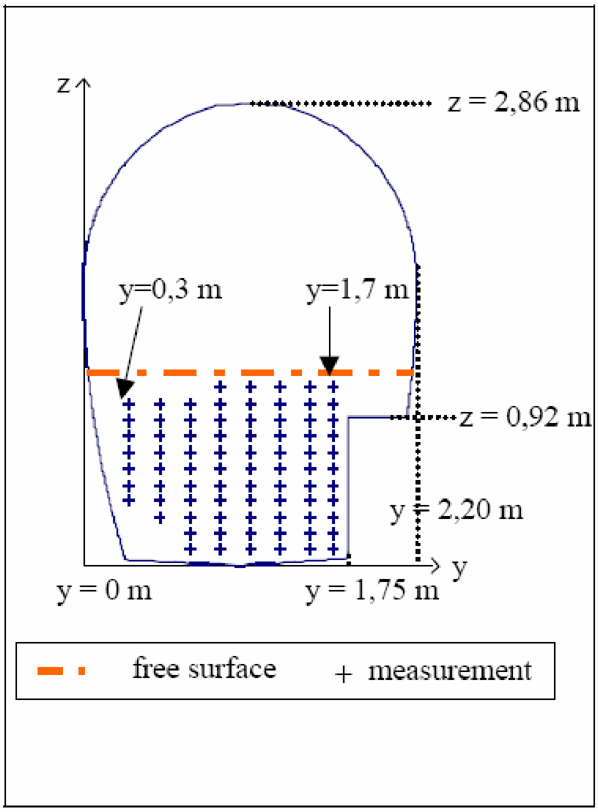
The experimental site (fig. 5) is located in an area called Cordon Bleu, on the main sewer line of the city of Nantes.

To investigate the spatial distribution of the velocities in large sewers, (Larrarte et al., 2005) have developed "Cerbère", a two dimensional remote-controlled device for measuring velocity fields with Doppler effect.

The mean velocity was 0.88 m/s, the water height 1.20 m.

No measurement could be realised over the walk-way.

figure 5 : experimental site characteristics: Cross section of the sewer



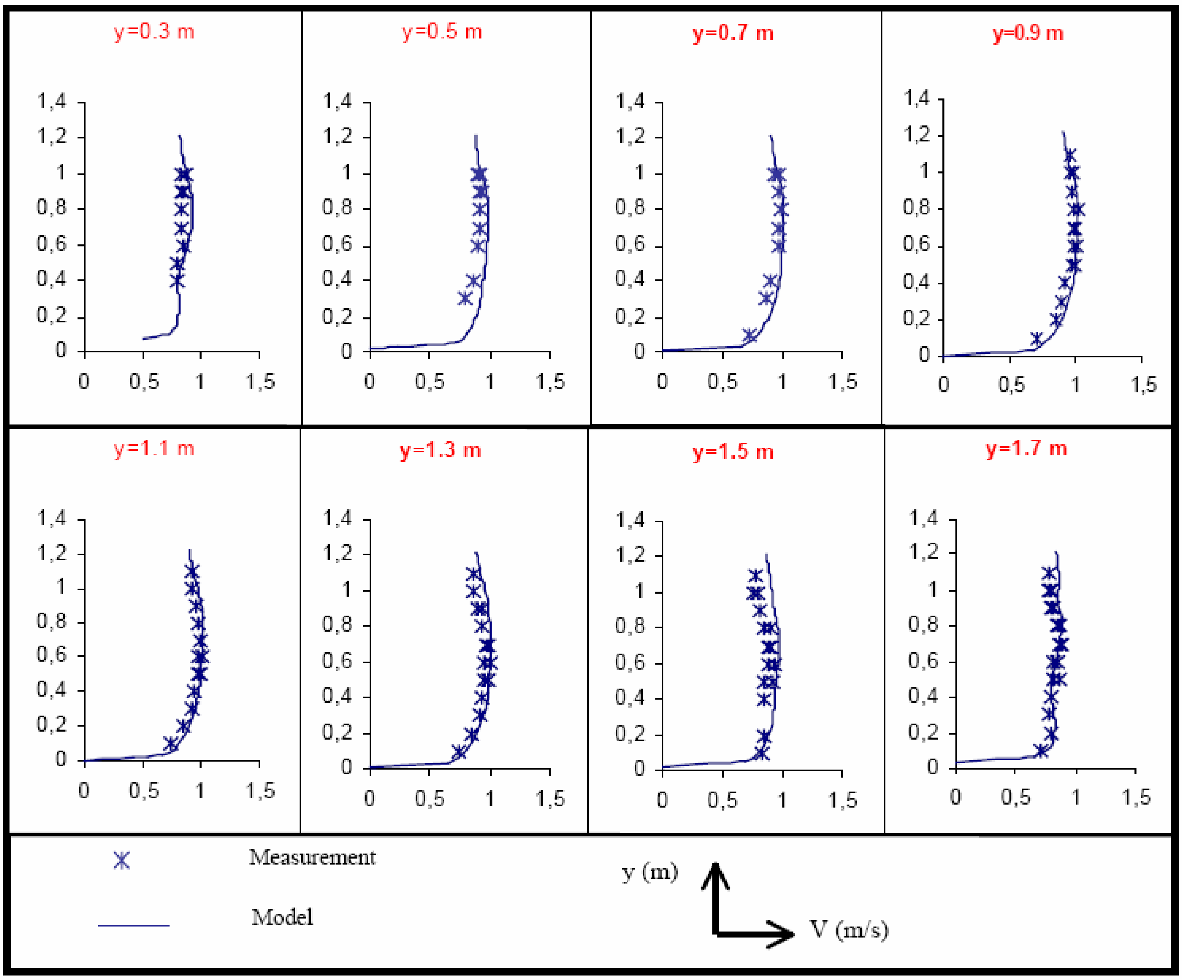
The system had been described through a 250 000 hexahedral cells mesh. The upstream condition was an uniform inlet velocity field (0.88 m/s), the downstream was a free outfall (Atmospheric pressure) over a weir (0.7m high). The roughness was equal to 2.44 mm and the initial turbulence intensity calculated by the relation $I = 0.16\mathrm{Re}^{-1/8}$ where the Reynolds number is defined by $\mathrm{Re} = D_h \,.\, U / \nu$, where D_h is the hydraulic diameter, U the inlet uniform velocity and ν the viscosity.

3.2 RESULTS

The results (fig. 6) display the collation, for each profile, of experimental results (dots) and calculated ones (line).

The dip-phenomenon is represented with a reliable location and the value of the maximal velocity is right. However, it probably would be better to emphasized the mesh in the vicinity of the free-surface and the wall to simulate better the effect of the bench (y=1.5 m and y=1.7m).

figure 6 : velocity profiles experimental and modelled



The authors allow themselves here to note that similar results were obtained by (Larrarte et al., 2005) with CFX software (VOF + RSM). The comparison between experimental data and model results show a good assessment of the isovelocity lines.

4 CONCLUSION

It appears that The anisotropic RSM model and the VOF method give, both at the same time, a good way to simulate water behaviour in open channels. With such results we can consider systematic computation to obtain, for different geometries and flows, velocity fields. This step would allow us to predict the velocity spatial distribution: In a pipe, if we have got one profile (with a sensor) we can know all the field and give the "true discharge".

REFERENCES

Ashley, R. M. Tait, S. Clemens, F. Veldkamp, R. (2004), Sewer processes – problems and new knowledge needs, Int Conf on Urban Drainage Modelling, Dresden, pp. 195-204.

Czernuszenko W. and Rylov A., « Modélisation des champs de vitesses tridimensionnelles en canal à surface libre », Journal of Hydraulic Research, Vol. 40, 2002, No. 2

Larrarte, F. Cottineau, L.-M. (2005), Nouveaux instruments pour connaître les champs de vitesses et de concentrations, Revue Européenne de Génie Civil, Vol. 9/7-8 – 2005, pp.963-978.

Larrarte, F. Bardiaux, J.B. Bonakdari, H. (2005) Computational and experimental study of velocity profiles in sewers, 10th International Conference on Urban Drainage, Copenhagen/Denmark, 21-26 August 2005

Naot, D. Rodi, W. (1982), Calculation of secondary current in channel flow, Journal of Hydraulic Engineering, Vol. 108, No 8, August, pp. 948-968.

Nezu, I. Tominaga, A. Nakagawa, H. (1993), Field measurements of secondary currents in straight rivers, Journal of Hydraulic Engneering, Vol. 119, No 5, May, pp. 598-614.

Stovin V. R. and Saul A. J. « Efficiency prediction for storage chambers using computational fluid dynamics », Water science & Technology Vol. 33, No 9, 1996, 163 – 170

Stovin V. R. Grimm, J.P. Buxton, A.P. Tait, S.J. (2002) Parametric studies on CFD models of sewerage structures, 9th International Conference on Urban drainage, Portland, August 2002.

Tominaga, A., Nezu, I., Ezaki, K, and Nakagawa, H. (1989) « Three - dimensional turbulent structures in straight open channel flows », Journal of Hydraulic Research, 27(1), , 149 – 173

Wohrle, C. Brombach, H. (1991), Probenahme im abwasserkanal, Wasserwirtschaft, Vol. 81, pp. 60-65.

Authors :

Jean-Bernard BARDIAUX
Robert MOSE
José VAZQUEZ
U.P.R. Systèmes Hydrauliques Urbains
Ecole Nationale du Génie de l'Eau et de l'Environnement de Strasbourg
1 quai Koch BP 1039 F, 67070 STRASBOURG CEDEX, France
Tel.: (33) 388248271
Fax: (33) 388248283
bardiaux@engees.u-strasbg.fr

Hossein BONAKDARI
Centre de Géomorphologie M2C UMR 6143 CNRS, 24 rue des Tilleuls, 14000 Caen

Laboratoire Central des Ponts et Chaussées
Division eau et environnement
Route de Bouaye BP 4129
44341 BOUGUENAIS CEDEX FRANCE

LARRARTE Frédérique
Laboratoire Central des Ponts et Chaussées
Division eau et environnement
Route de Bouaye BP 4129
44341 BOUGUENAIS CEDEX FRANCE
Tel : (33) 2.40.84.58.82
Fax : (33) 2.40.84.59.98
frederique.larrarte@lcpc.fr

HYDRAULISCHE MODELLIERUNG DER HOCHWASSER-ENTLASTUNGSANLAGE DES HOCHWASSERRÜCKHALTE-BECKENS I in GLASHÜTTE

Veranlassung der Modellversuche

Das beim Augusthochwasser 2002 gebrochene Absperrbauwerk des Hochwasserrückhaltebeckens I Glashütte wird z. Zt. im Rahmen der Hochwasserschadensbeseitigung von der Landestalsperrenverwaltung des Freistaates Sachsen, Talsperrenmeisterei Gottleuba/Weißeritz, wiedererrichtet.

Da das Hochwasserrückhaltebecken in der bisherigen Form jed… den Anforderungen des Hochwasserschutzkonzeptes für die … genügt, wurde die vom Auftraggeber der Modellversu… GmbH, entworfene Hochwasserentlastungsanla… der Verwendbarkeit in einem größeren Bec…

Die HWE wird als Schachtüberfall … Tosbecken ausgeführt.

Aufgrund des sehr ku… Richtungsänderung … Strömungsverhältnisse … bekannten Berechnungs…

Die Dr. Salveter GmbH … hydraulische Modellversuc… …lage nachzuweisen und ggf. A… …keiten zu machen. Ein Student am M… …durchführung im Rahmen einer Belegarbeit.

Berechnungsgrundlagen

Bezeichnung	Formelzeichen	Wert	Bemerkung
Rückhalteraum	I_{GHR} [hm³]	0,05	kleines Becken
OK Sohle Einlaufbauwerk	H_S [m ü HN]	379,95	-
Stauziel Vollstau	ZV [m ü HN]	387,85	mittleres B…
Stauziel 1 (mittleres Becken)	$ZH_{1,m}$ [m ü HN]	388,55	mitt…
Stauziel 1 (großes Becken)	$ZH_{1,g}$ [m ü HN]	389,10	
Stauziel 2 (mittleres Becken)	$ZH_{2,m}$ [m ü HN]	3…	
OK Dammkrone	H_{DK} [m ü HN]		
BHQ_1 (mittleres Becken)	HQ_{500} [m³/s]		
BHQ_1 (großes Becken)	$HQ_{1.000}$ [m³/s]		
BHQ_2 (mittleres Becken)	$HQ_{5.000}$ [m³/s]		
BHQ_2 (großes Becken)	$HQ_{10.000}$ [m³/s]	7…	

Versuchsdurchführung

Das Modell wur… Maßstab 1:20 von der DESIGNPROJEKT GmbH Dresden im Hubert… …or errichtet. Im Modell wurde das gesamte hydraulische Sys… …erentlastung mit Einlauftrichter, Krümmer, Ablaufstollen …ildet.

…Krümmer und der gesamte Ablaufstollen wurden aus … (Acrylglas bzw. glasklares Polysterol) hergestellt, …en Bereichen zu gewährleisten.

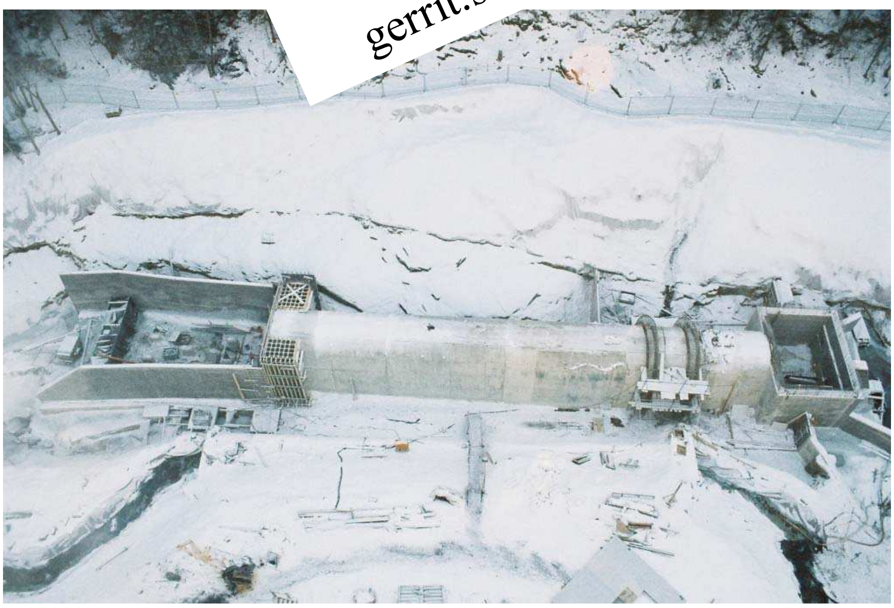
…dablasseinlauf und die Belüftungseinrichtungen …en Verhältnisse bei allen relevanten Abfluss-…

…bfluss- und Geschwindigkeitsmessungen sollen …wurfsverfasser berechneten Überfallhöhen-Abfluss- … auch der Überdeckungsabfluss und die Verhältnisse im …berprüft werden.

…d von Messwerten und visueller Beobachtungen sollen die Leistungs-…ähigkeit der Anlage nachgewiesen und ggf. Optimierungsvorschläge gemacht werden.

Ausgewählte Abmessungen der Anlage

Bauteil	Bezeichnung	Einheit	Natur	Modell
Einlauftrichter	Außendurch…	m	6,00	0,30
	Durc…	m	2,68	0,13
		m	3,70	0,19
Krümmer		m	3,50	0,18
		m	2,23	0,11
		m	1,00	0,05
		m	2,50	0,13
			33,65	1,68
		m	3,80	0,19
		m	4,30	0,22
		m	18,50	0,93
		m	8,72	0,44
		m	1,20	0,06
	…länge	m	61,45	3,07
	…esamtbreite	m	8,72	0,44
	Gesamthöhe	m	10,86	0,54



Dr. Salveter GmbH
Ingenieurbüro für Bauwesen
Beraten - Planen - Überwachen
www.dr-salveter.de

Auftraggeber: Dr. Salveter GmbH
Projektleiter: Dr.-Ing. Gerrit Salveter
Betreuer: Dipl.-Ing. Holger Haufe
cand.-Ing. Daniel Dommaschk

Zeitraum: Oktober bis Dezember 2005
Kontakt: Dr.-Ing. Gerrit Salveter
Tel.: ++49 3529 50 10 0
Fax: ++49 3529 50 10 10
gerrit.salveter@dr-salveter.de

IWD