EXPERIMENTAL INVESTIGATIONS OF 2D STATIONARY MIXED FLOWS AND NUMERICAL COMPARISON

N. Van Nam[1], S. Erpicum[1], B. Dewals[1], M. Pirotton[1] & P. Archambeau[1]

[1]Hydraulics in Environmental and Civil Engineering – HECE, ArGEnCo Department, Liege University, Belgium

e-mail: vannam.nguyen@doct.ulg.ac.be, s.erpicum@ulg.ac.be,

Abstract

Experimental investigations have been carried out to observe the mechanisms of stationary mixed flows in a flume combined with a conduit. The tests have been performed for varied discharges, considering two 4.2m long rectangular channels 0.985m wide and 0.50m deep, linked by a 2m long closed conduit with a 0.20m wide and 0.15m high rectangular cross-section, located at the channels bottom along the right bank. The experimental results have been compared with numerical modeling performed using an original 2D numerical model, dealing with mixed flows using a single set of equations.

Introduction

Mixed flows are known as the simultaneous occurrence of free-surface and pressurized flows. These phenomena are often encountered in water supply systems, sewer systems, storm-water storage pipes, flushing galleries, water conservancy projects, hydraulic structures, etc.. (Erpicum et al., 2008; Kerger et al., 2009).

Mixed flows have been presented in many studies from both experimental and numerical point of views. The first studies of mixed flows regimes, conducted in the decades before and after WW2, were hydraulic scale models that looked at the design of particular structures, (Djordjevic & Walters, 2004). These studies mainly focused on cases where the one-dimensional approximation is valid: Wiggert (1972) produced one of the first experimental investigations of the surge wave propagation in tunnel; Valentin (1981) investigated experimentally the hydraulic phenomena that occurs during the transition between free surface and pressurized flow in a sewer system. Sundquist and Papadakis (1982) performed experimental studies to simulate the flow conditions in a circulating water system of a thermal plant; Capart et al. (1997) performed an experimental study of the transient transcritical flow in a closed sewer pipe; Li & McCorquodale (1999) performed experimental studies to observe the pressure transients in a sewer system; Gómez & Achiaga (2001) performed a study to analyze the transition from free-surface to pressure flow at both ends of a pipeline; Vasconcelos and Wright (2005) performed an experimental work to observe the nature of flow regime transition in pipes; Wright et al. (2008) presented a study to demonstrate both types of flow regime transition in pipes also; Erpicum et al. (2008) carried out an experimental and numerical investigation of mixed flow in a gallery and Kerger (2009) considered this flow with the air/water interaction on numerical simulation point of view.

2D shallow flows, where the lateral velocity is not negligible regarding the main direction one, are common in hydraulic engineering. They have been extensively studied and modelled for years. However, such flows in mixed configurations have not been studied thoroughly to date, neither experimentaly nor numerically.

With the objective to begin to fill this gap, this paper presents the first results of an experimental study considering stationary mixed flows taking place in two free-surface channel reaches linked by a closed conduit. These experimental studies have been carried on with the main objective to validate a numerical model, which is also presented in the paper. In this scope, the distribution of velocity and pressure fields in both free-surface channels and pressurized conduit portions have been characterized in a wide range of flow conditions.

For the purpose of comparison, numerical modelling has been carried out with an original 2D mathematical model for mixed shallow flows implemented in the academic software WOLF. WOLF is a finite volume scheme developed within the Research Group of Hydraulics in Environmental and Civil Engineering (HECE) of Liege University. Using unified pressure gradients, the shallow water equations applicability is extended to pressurized flow. The mathematical model is based on an extenion of the Preissmann slot concept, valid for Saint-Venant equations.

Comparison between numerical and experimental data shows good agreement and proves the validity of the numerical approach.

The experimental results provided in this paper constitutes a reliable benchmark for validating 2D mixed flows numerical models. In addition, they provide insight into the mechanism of transition and into the flow characteristics in both free surface and under pressure flows.

Experimental Setup

Physical model

Experimental investigations have been performed in a system made of two 4.2m long reaches with a 0.985x0.50m constant rectangular cross-section linked by a 2m long close conduit with a 0.20x0.15m uniform rectangular cross-section (Figure 1). The closed conduit is connected to the reaches along their right bank, at the flume bottom. The flume is horizontal and made of clear glass. The conduit is made of exterior type wood on the left and upper faces. The lower and the right ones are of clear glass (flume sides).

A pipe network and automated pumps are used to feed water into the usptream alimentation basin, equipped with a permeable screen to distribute uniformly the water into the model. At the downstream extremity of the physical model, a thin steel plate 0.27m high is used to adjust the downstream water level and discharge release. This gate is used as a free weir or a rising gate, depending on the discharge configuration and wished usptream water level. Some photos of the model are given in Figure 2.

Measurement Systems

The model is equipped with the following measurement systems:

- The upstream discharge (inflow) is measured with an electro-magnetic discharge meter (accuracy of 0.5l/s) and adjusted with a frequency regulator on the pumping system.
- The velocity measurement system consists in an electro-magnetic (EM) probe manufactured by Valeport, model 802 OEM, placed on a mobile beam above the channels.
- The water level measurement system (free surface channels) includes 8 ultrasound sensors from Microsonic ranging from 350 to 60mm in height from the water surface.
- The pressure measurement system (closed conduit) is made of 8 Keller piezoresistive pressure transducers, connected to a LabView software for signal treatment.

It is not economical nor interesting to measure the flow parameters at every location on the model. Therefore, specific location have been selected and used for measurements. Details of these gauges positions and definition of dimensions are shown in Figure 3.

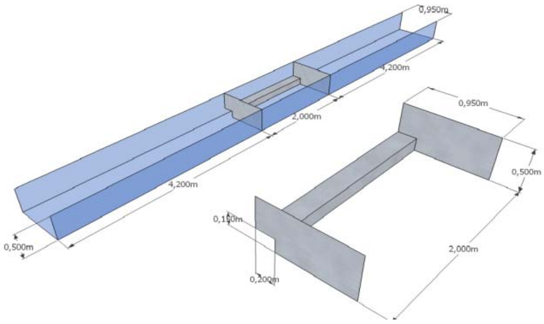

Figure 1: Sketch of the experimental model

a) General view b) Closed conduit

Figure 2: Photos of experimental model

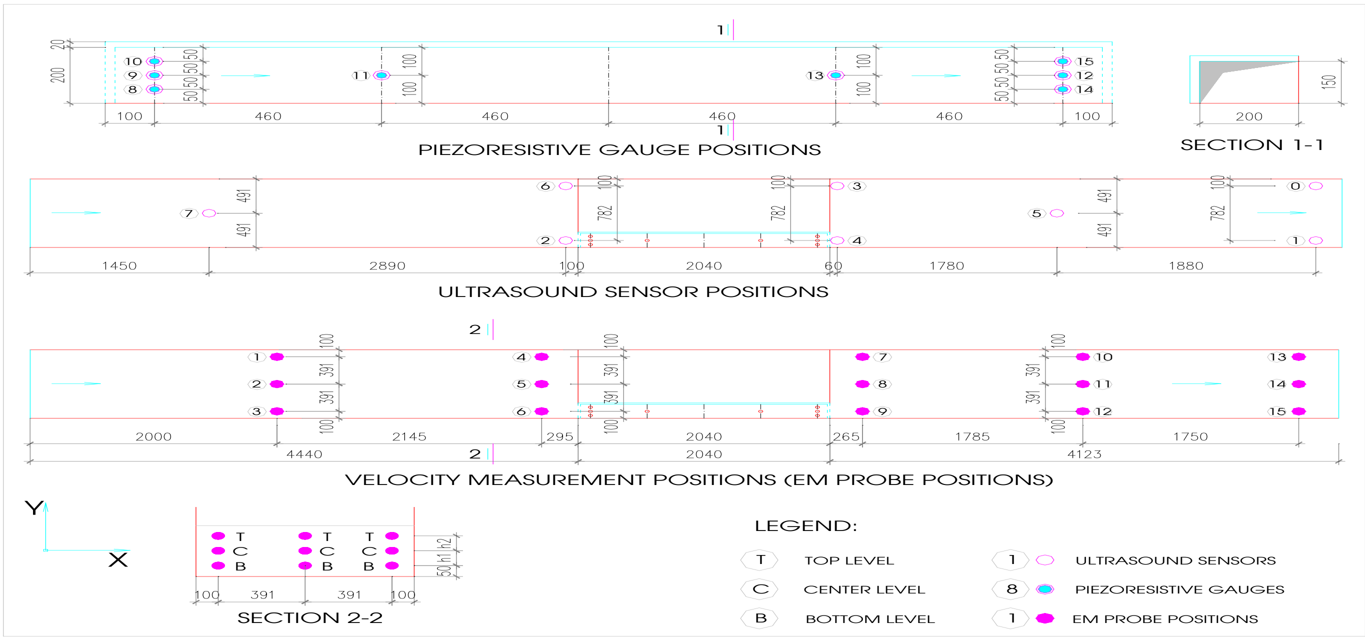

Figure 3: Measurement positions of flow parameters

Numerical Model

The 2D multiblock flow solver WOLF2D, part of the modelling system WOLF, is based on the conservative form of the so-called shallow water equations (Erpicum et al., 2009). This set of equations is usually used to model two-dimensionnal unsteady open channel flows, i.e. natural flows where the vertical velocity component is small compare to both horizontal components (Erpicum et al., 2010a). It is derived by depth-integrating the Navier Stoke equations. It counts for hydrostatic pressure distribution and uniform velocity components along the water depth.

Using the Preissmann slot model (Preismann, 1961), pressurized flow can equally be calculated by means of the Saint-Venant equations by adding a conceptual slot on the top of a closed pipe. When the water level is above the maximum level of the cross-section, it provides a conceptual free surface flow, for which the gravity wave speed is related to the slot geometry (Kerger et al, 2009).

To deal with steady pressurized flows, the Saint Venant equations writes as in Eq 1-3. The Preismann slot dimensions are the mesh size as in steady flow, pressure is not related to the slot characteristics.

$$\frac{\partial h}{\partial t}+\frac{\partial u}{\partial x}+\frac{\partial vb}{\partial y}=0 \qquad (1)$$

$$\frac{\partial}{\partial t}(bu)+\frac{\partial}{\partial x}\left(bu^2+\frac{g(2h-b)b}{2}\right)+\frac{\partial}{\partial y}(buv)= \qquad (2)$$

$$-gh_b\frac{\partial z_b}{\partial x}+gh_r\frac{\partial z_r}{\partial x}+gh_J J_x$$

$$\frac{\partial}{\partial t}(bv)+\frac{\partial}{\partial y}\left(bv^2+\frac{g(2h-b)b}{2}\right)+\frac{\partial}{\partial x}(buv)= \qquad (3)$$

$$-gh_b\frac{\partial z_b}{\partial y}+gh_r\frac{\partial z_r}{\partial y}+gh_J J_y$$

In Eqs 1 to 3, *u* and *v* are the velocity components along *x* and *y* axis respectively, *h* is the water depth, *b* is the conduit heigth, z_b and z_r are the bottom and roof elevations, h_b, h_r, and h_J are equivalent pressure terms and J_x and J_y the components along axis of the energy slope. The bottom friction is conventionally modelled with empirical laws, such as the Manning formula. To deal with both free surface and pressurized flows, *b* is computed as the minimum of the conduit elevation (infinity in case of free surface reach) and the water depth *h*. (Figure 4)

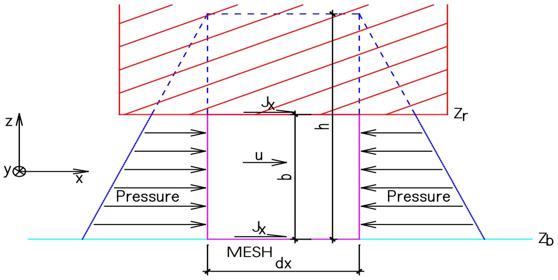

Figure 4: Sketch of the mathematical model variables

The space discretization of the conservative equations is performed by means of a finite volume scheme. This ensures a proper mass and momentum conservation, which is a prerequisite for handling reliably discontinuous solutions. As a consequence, no assumption is required as regards the smoothness of the solution. Variable reconstruction at cells interfaces is performed by constant or linear extrapolation, in conjunction with slope limiting, leading in the later case to a second-order spatial accuracy. Flux treatment is based on an original flux-vector splitting technique (Erpicum et al., 2010a). The hydrodynamic fluxes are split and evaluated partly downstream and partly upstream according to the requirements of a Von Neumann stability analysis. Optimal agreement with non-conservative and source terms as well as low computational cost are the main advantages of this original scheme (Erpicum et al., 2010b). Explicit Runge-Kutta schemes are used for time integration.

Results

Investigations focused on stationary mixed flows and aimed at determining the velocity and pressure fields distribution in both directions of the main flow plane. Table 1 summarizes the range of discharges considered in the experimental and numerical investigations depending on the downstream boundary conditions (free weir or raising gate).

Each measurement on the physical model has been done several times to ensure the consistency of the results.

Table 1: Range of discharges studied in the experimental and numerical investigations

Discharge [l/s]	
Free weir	Raising gate
5.0 10.1 15.1	20.0; (a=16mm) 30.0; (a=26mm) 40.0; (a=35mm)

(a: openning gate rate [mm])

Pressure field

The pressure fields have been measured in both of free surface and pressurized flows with an elevation reference at the channel bottom.

The experimental and numerical results are representend on Figures 5 and 6 along a profile defined along the test facility (following the section 7-2-9-11-13-12-5-1 of gauges locations on Figure 3). Figure 5 shows the results of small discharges and a free weir downstream of the channel while Figure 6 shows the results for higher discharges and a downstream raising gate.

Velocity field

Velocity measurement have been conducted in the open channel flow reaches. The EM probe measures the two velocity components Vx and Vy in the sensor plane, placed

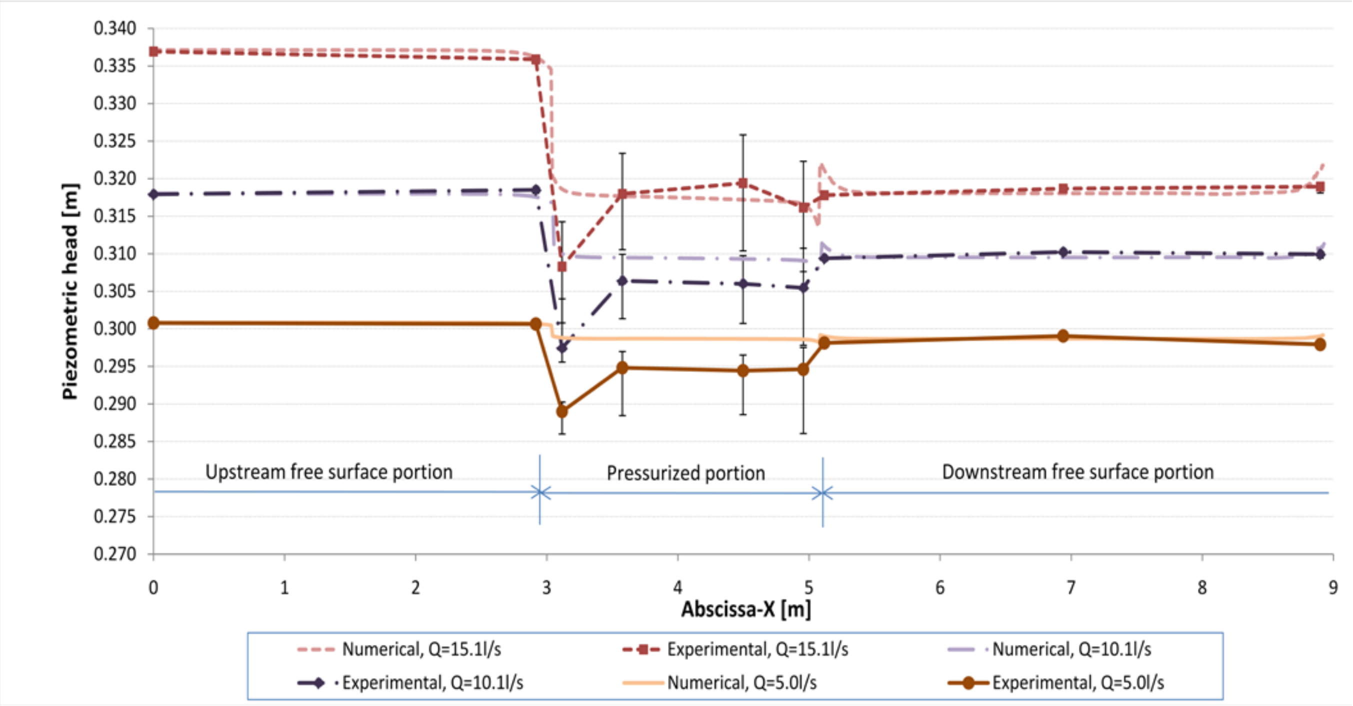

Figure 5: Piezometric head versus distance along the channel (section 7-2-9-11-13-12-5-1 on Figure 3), free weir, Q=5.0÷15.1 [l/s]. The error bars represent the variation of the measurement on the physical model or of the numerical results

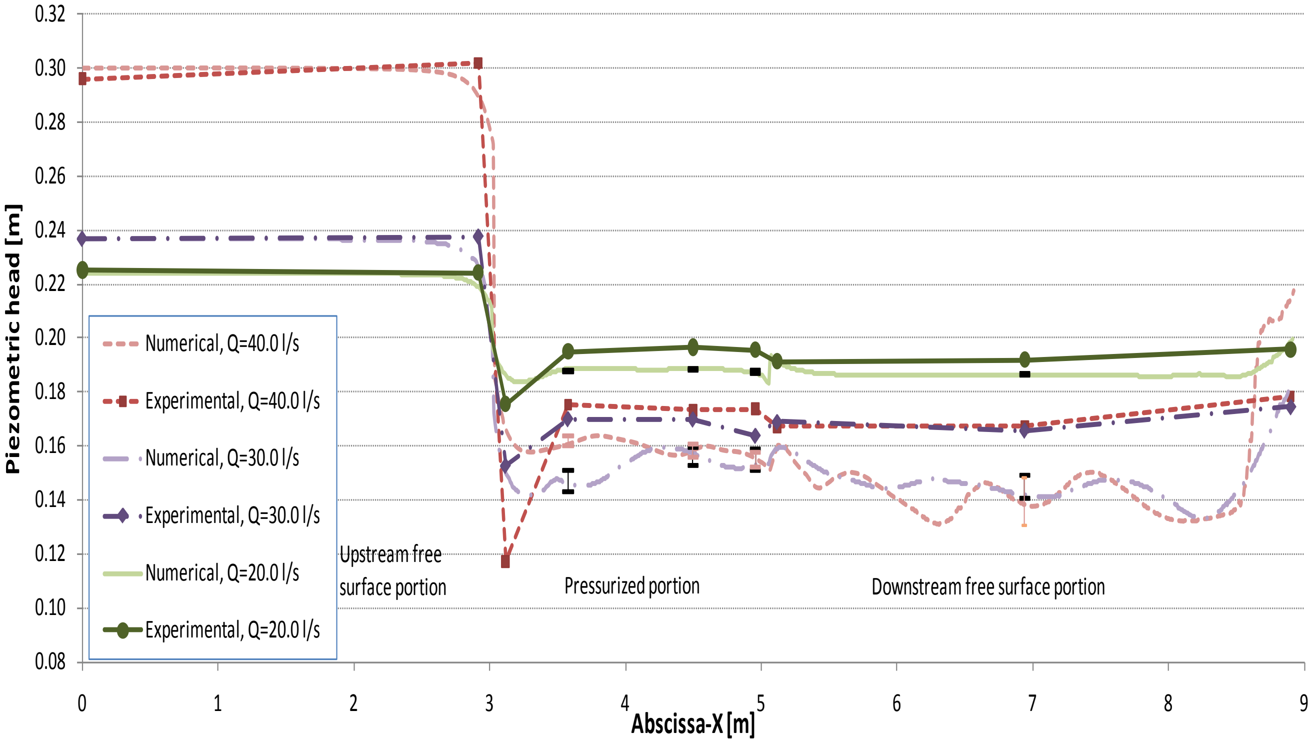

Figure 6: Piezometric head versus distance along the channel (section 7-2-9-11-13-12-5-1 on Figure 3), raising gate, Q=20.0÷40.0 [l/s]. The error bars represent the variation of the measurement on the physical model or of the numerical results

parallel to the channel bottom. Vx>0 is a flow direction from upstream to downstream of the flume; Vy>0 is a flow direction from the right wall to the left side of the flume. The magnitude and direction of total velocity are computed from Vx and Vy values. Corresponding to each discharge value, the velocity has been measured at 2 to 3 levels depending on the water depth (see Figure 3).

An example of the distribution of the mean velocity components Vx and Vy in channel cross-sections is represented on Figure 7 for the discharge of 10.1l/s.

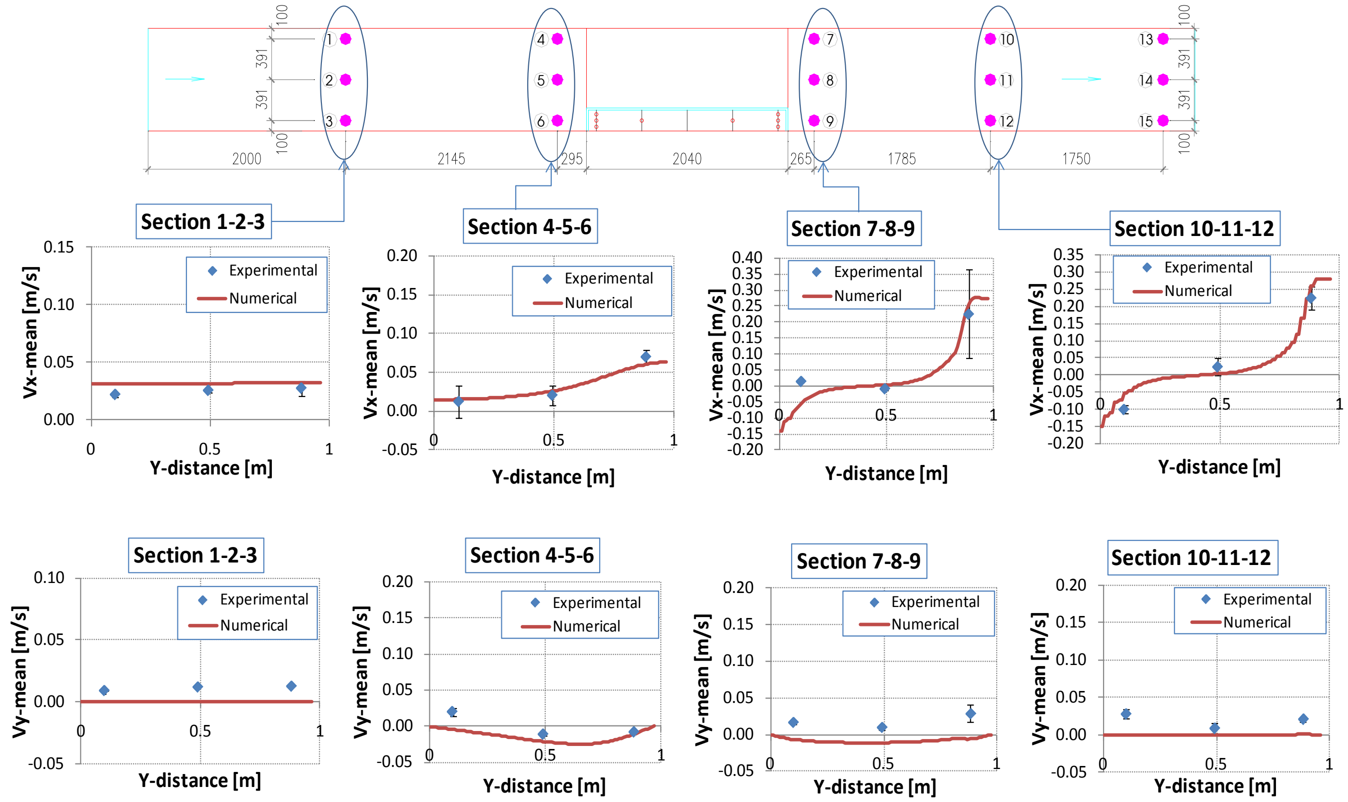

Figure 7: Mean velocity components Vx, Vy versus distance of the channel cross section, Q=10.1 l/s and position of the cross sections. Vx-mean and Vy-mean are the averaged values of Vx and Vy values at levels B, C…, respectively)

During the tests, vortex with air entrainment has been observed at front of gate, especially in case of raising gate (Figure 8). The amplitude of the phenomenon increases with the discharge.

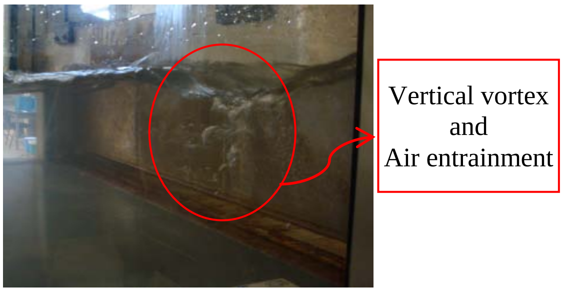

Figure 8: Vortex and air entrainment at front of gate (raising gate, Q=20.0÷40.0 [l/s])

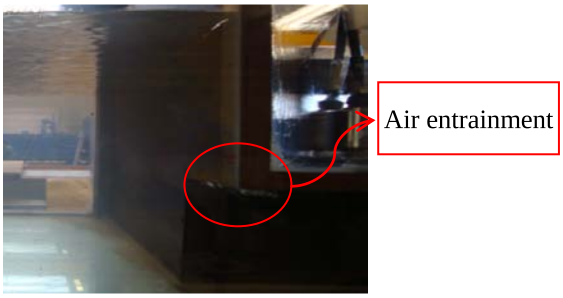

Figure 9: Conduit inlet with air bubbles (raising gate, Q=20.0÷40.0 [l/s])

Discussion

In this section, we discuss the experimental data first and then we analyse the comparison with numerical results.

The specific layout of the physical model induces specific flow characteristics in each section of the flume:

1. Pressure and velocity fields in the upstream free surface portion of the system are uniform and constant (Figure 5, Figure 6 and Figure 7) whatever the discharge; except at the cross section close to the conduit inlet because of the concentration of flow into the conduit. At that place, the flow velocity increases in front of the conduit inlet with locally non negligible Vy components variation.
2. In the downstream free surface portion, a recirculation area develops because of the high velocity of the flow at the conduit outlet along the right bank (positive and negative Vx components on cross sections 7-8-9 and 10-11-12 on Figure 7). This phenomenon is increased by the downstream weir or gate which bounds the channel reach.
3. In the closed conduit, the longitudinal pressure distribution is relatively uniform, except near the conduit inlet where a significant drop is measure. This can be explained by a recirculation area at the conduit top and the air entrainment (Figure 9).
4. In all cases, the head loss between upstream and downstream reach is significant.

The comparison of experimental data and numerical results has been performed considering a longitudinal profile along both channel reaches and the conduit. In addition, some typical cross sections have also been considered.

In general, the global head loss from side to side of the conduit is well reproduced.

Numerical results are in good accordance with experimental data on the upstream free surface channel whatever the discharge. In the downstream reach, the recirculating flow induces periodic oscillations of the numerical results, especially for higher discharges. These oscillations are identified by the error bars, and reproduced on Figure 6.

In the pressurized section, the experimental pressure data and the computation results are in good accordance for small discharge configurations (Figure 5) as the variation of the physical results includes the respective numerical results. For higher discharges (Figure 6), agreement is not so good, especially at the conduit inlet.

Regarding velocity fields, the physical measurement tendancy is generally well reproduced by the numerical model (see for instance results for a 10.1 l/s discharge on Figure 7).

Conclusions

Experimental investigations have been carried out to observe the mechanisms of stationary 2D mixed flows in a flume combined with a conduit. A wide range of discharges and downstream boundary conditions has been carefully considered to measure velocity and pressure fields distribution, providing a large set of data to characterize the flow. These data have been used in the purpose of comparison with numerical results provided by a 2D flow solver developed to model mixed shallow flows.

The experimental results underlined the non uniformity on transverse sections of velocity and pressure fields. In addition, they provide insights into the mechanisms of transition from free surface to pressurized flow.

A relatively good agreement between measured and computed results has been obtained. In particular, the global head loss form side to side of the conduit is well reproduced. Numerical periodic oscillations are observed in the dowsnstream reach, where a large recirculation area takes place.

In the next steps, a detailed analysis of both physical and numerical results will be performed in order to identify the possible causes of discrepancy. In addition, similar works will be done for other conduit geometries to enlarge the importance of 2D effects and thus the complexity of the flow.

Ackownledgements

The authors would like to acknowledge the support of the Ministry of Education and Training of Vietnam, the VIED, which has provided a fellowship to support the PhD studies of the first author.

References

Capart, H., X. Sillen, and Y. Zech (1997), Numerical and experimental water transients in sewer pipes*. Journal of Hydraulic Research, 35(5): p. 659 - 672.*

Djordjevic, S. and G.A. Walters (2004). Mixed free-surface/pressurized flows in sewers. *in WaPUG Meeting from Scotland and Northern Ireland. Dunblane.*

Erpicum, S., F. Kerger, P. Archambeau, B.J. Dewals, and M. Pirotton. (2008) Experimental and numerical investigation of mixed flow in a Gallery. *Engineering Sciences, volume 1, Computational Methods in Multiphase Flow V.*

Erpicum, S., Meile, T., Dewals, B. J., Pirotton, M., Schleiss, A. J. (2009). 2D numerical flow modeling in a macro-rough channel. *Int. J. Numer. Methods Fluids*, 61(11), 1227-1246.

Erpicum, S., Dewals, B.J., Archambeau, P., & Pirotton, M. (2010a). Dam-break flow computation based on an efficient flux-vector splitting. *Journal of Computational and Applied Mathematics.***234**, 2143-2151.

Erpicum, S., Dewals, B. J., Archambeau, P., Detrembleur, S. & Pirotton, M. (2010b). Detailed inundation modelling using high resolution DEMs. *Engineering Applications of Computational Fluid Mechanics,* 4(2)**,** 196-208.

Gomez, M. and V. Achiaga (2001), Mixed Flow Modelling Produced by Pressure Fronts from Upstream and Dowstream Extremes*. In Brashear, R.B., Maksimovic, C. (Eds), Urban Drainage modeling-Proceedings of the Specialty symposium Held in conjunction with World Water and Environmental Resources Congress. ASCE, Orlando, FL.*

Kerger, F. (2009), *Numerical Simulation of 1D Mixed Flow with Air/Water Interaction.* PhD Thesis, HECE, University of Liege, Belgium

Kerger, F., P. Archambeau, S. Erpicum, B.J. Dewals, and M. Pirotton (2009), A Fast Universal Solver for 1D Continuous and Discontinuous Steady Flows in Rivers and Pipes*. International Journal for Numerical Methods in Fluids, 66(1); pp33-43.*

Li, J. and A. McCorquodale (1999), Modeling Mixed Flow in Storm Sewers*. Journal of Hydraulic Engineering, 125(11): p. 1170-1180.*

Preissmann, A. (1961), Propagation des intumescences dans les canaux et rivieres*. In First Congress of the French Association for Computation. 1961. Grenoble, France.*

Sundquist, M.J. and C.N. Papadakis (1982), Surging in Combined Free Surface-Pressurized Systems*. Journal of Transportation Engineering, 109(2): p. 232-245.*

Valentin, F. (1981), Continuous discharge measurement for the transition between partly filled and pressurized conduit flow in sewerage system.
Water Sci. and Technol, 13(8), 81–97.

Vasconcelos, J. and S. Wright (2005), Experimental Investigation of Surges in a Stormwater Storage Tunnel*. Journal of Hydraulic Engineering, 131(10): p. 853-861.*

Wiggert, D. (1972), Transient Flow in Free-Surface, Pressurized systems. *Journal of the Hydraulics Division, Proceedings of the American Society of Civil Engineers, 98(1): p. 11-26.*

Wright, S., Vasconcelos, J., Creech, C., Lewis, J. (2008), Flow regime transition mechanisms in rapidly filling stormwater storage tunnels. Environmental Fluid Mechanics, 8(5): p. 605-616.