Experimental and numerical modelling of the velocity distribution in the dissipation chamber of a navigation lock with loop culvert filling

CHRISTIAENS Damien[1], CONTINO Francesco [1], DUBOIS Nicolas [1], SEYNHAEVE Jean-Marie [1], JEANMART Hervé [1], ZECH Yves [1], SAVARY Céline [2], BOUSMAR Didier [1,2], SOARES-FRAZAO Sandra [1]

[1]Institute of Mechanics, Materials and Civil Engineering, Université catholique de Louvain, Belgium,

Place du Levant, 1, B-1348 Louvain-la-Neuve

[2]Hydraulic Research Laboratory, Service Public de Wallonie,
Belgium, Rue de l'Abattoir, 164, B-6200 Châtelet

email: damien.christiaens@uclouvain.be

Scientific Category: hydraulics, lock design

Abstract

This paper presents preliminary results of a composite modelling approach aimed at optimising the design of the dissipation chamber in a navigation lock with loop culvert filling system. The considered lock is to be built on the River Meuse at Ampsin-Neuville (Belgium) within the frame of the TEN-T 18 project. The composite modelling approach consists in combining experimental observations on a scale model of the lock filling system including the dissipation chamber with two-dimensional numerical modelling of the flow in the system. The paper first presents the experimental model and the 2D numerical solver. Then, experimental measurements at the outlet of the dissipation chamber are compared to the results of the numerical simulations.

Introduction

In order to promote the waterway transport, public agencies support many projects in Europe to modernize inland waterways. In Wallonia (Belgium), two priority projects are supported by the Trans-European Transport Network (TEN-T): the Rhine / Meuse-Main-Danube (TEN-T 18) project that crosses Europe transversally from the North Sea at Rotterdam to the Black Sea in Romania and the Seine-Scheldt (TEN-T 30) project that links Paris, Antwerp and Rotterdam. These projects imply an increase of the lock capacity of the Belgian waterways to reach ECMT class Vb allowing vessels carrying 4500 tonnes for Seine-Scheldt project and ECMT class VIb allowing the traffic of boats carrying 9,000 tonnes for Rhine / Meuse-Main-Danube project.

Following PIANC (1986), navigation locks are hydraulic structures separating water surfaces with different water levels that enable vessels to pass from one side to the other by operating mobile elements. A navigation lock is composed of doors, a filling / emptying system and a lock chamber. The lock chamber is filled by transferring water by gravity from the upstream reach to the chamber.

Different types of filling / emptying systems exist, depending on the required lift speed, the building costs or issues related to ship motion (PIANC, 2009) such as the lock capacity. The lock capacity is defined as the average number of vessels that can be locked within a certain period of time, given sufficient traffic intensity (PIANC, 2009). It depends on several parameters, one of them being the duration of a lock cycle, including lock operations, vessel operations and vessel waiting time.

Within the frame of the TEN-T projects, to achieve the objective of an increased capacity of the Rhine / Meuse-Main-Danube, a new lock of 225×25 m has to be built at Ampsin-Neuville on the River Meuse (Belgium). In order to guide the design of this new lock, a scale model of a lock head has been set-up at the Hydraulics Research Laboratory of Walloon waterways administration (Figure 1). The specific aim of this model is to investigate the possibility to use a short loop culvert filling system. This system has the advantage to limit the required space to build the lock, which is one of the major constraints at the Ampsin site. Key issues in the design of such a short loop-culvert system concern (1) the dissipation chamber layout, to obtain sufficient uniformity in the velocity distribution at the outlet; and (2) the estimation of the head losses of this chamber. Indeed, a good estimate of the head losses is required to determine the lock filling hydrograph (depending on the valve opening law) and from there the evolution of the free surface in the lock chamber. The free-surface slope should be limited to limit the hawser forces encountered by vessels in the lock chamber.

The paper focuses on the velocity distribution at the outlet of the dissipation chamber, for two possible configurations of this chamber. The composite modelling approach where experiments and numerical modelling are combined is first presented. Then, experimental measurements at the outlet of the dissipation chamber are compared to the results of the numerical simulations and preliminary conclusions about the optimal design are drawn.

Composite modelling approach

By definition, a composite modelling approach combines experimental and numerical techniques, but also field measurements and analytical analysis, to take benefit of the advantages of each approach in order to optimize the results. These two techniques can be coupled and integrated notably to calibrate and validate the numerical models or to treat the boundary conditions (PIANC, 2009).

In this research, the different types of approaches are combined as follows:

- A 1D numerical model called ALFREDO (Vancaster et al., 1999) was developed to simulate in a simplified way the whole lock filling or emptying process. The flow through the culvert system is calculated by a finite difference method solving the pipe-network equations. In these equations, the head losses induced by the different elements of the culvert system, including the valve and the dissipation chamber, are estimated empirically, either from the literature (Idel'Cik, 1960) or from the experiments. These calculations are coupled to a 1D finite-volume scheme solving the shallow-water equations in the lock chamber to obtain the free-surface evolution.
- Then, the scale model is run for some discharges selected from the 1D computation, and the velocity distribution at the outlet of the dissipation chamber is measured. These measurements are further described in this paper.
- Finally, a 2D numerical model of the pressurized flow in the dissipation chamber is used to compute the detailed velocity field in the chamber. This 2D model is calibrated and validated by means of the scale-model measurements.

Once validated, the 2D model can be used to guide the design of the layout of the dissipation chamber, in order to obtain the best possible velocity profile at its outlet.

Experimental set-up

Description of the scale model

The set-up consists of a 1:12 scale model of the upstream part of a 12.5 m wide lock (Figure 1). The model is located at the Hydraulics Research Laboratory of Walloon waterways administration (Belgium). This 1.042 m wide model includes a short part of the upstream reach, the culvert system including intakes, butterfly valves and dissipation chamber, and a part of the lock chamber.

Figure 1 : scale model of the upstream head

The model is operated in steady state, with a prescribed valve opening and keeping the water level in the upstream reach and in the lock chamber constant Figure 2 gives a sketch indicating the flow direction through the system. The model is operated either in symmetric conditions, i.e. with the two valves working synchronously, or in asymmetric conditions with a single valve open, corresponding e.g. to maintenance operations.

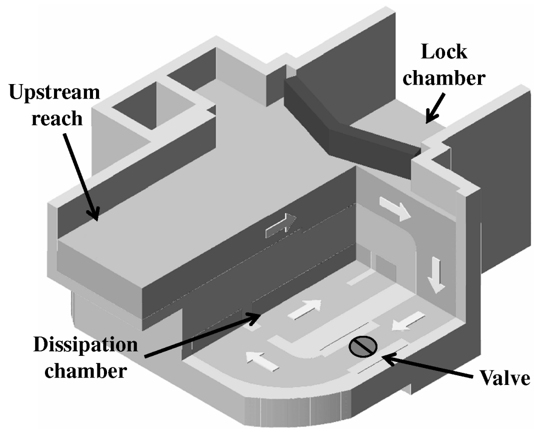


Figure 2 : Flow through the lock filling system

Two different configurations of the dissipation chamber are considered (Figure 3). The configurations only differ by the addition of a 555 mm by 278 mm triangle at the entrance of the dissipation chamber. This triangle was intended to better control the flow distribution in asymmetric condition.

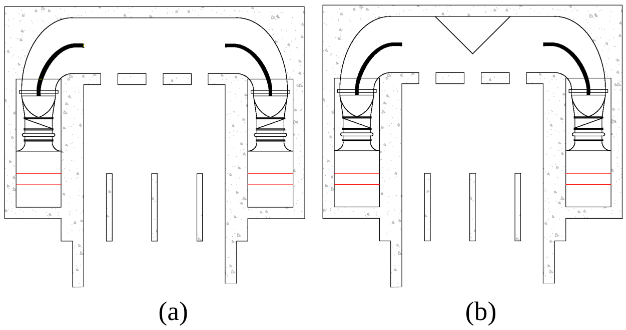


Figure 3 : experimental configurations of the dissipation chamber: (a) configuration 1; and (b) configuration 2

Measurement devices

The model is equipped for the following measurements (Figure 4):

- *Discharge*: the discharge is evaluated with a V-notch weir, located at the outlet of the model. The water level on the weir is recorded using an ultrasonic sensor (Baumer UNAM18U6903)
- *Water level*: the water level in the upstream reach z_{up} and in the lock chamber level z_{lc} are recorded using similar ultrasonic sensors.
- *Head losses*: 20 piezometric taps are placed at significant points in the culverts and in the dissipation chamber. Taps can be connected 2 by 2 to differential pressure transducers (Druck LPM 9381) to record the static head loss.
- *Velocity*: a series of 5 electromagnetic velocity probes (WL|Delft Hydraulics PEMS E30) are mounted on a movable device at the outlet of the dissipation chamber, to obtain the velocity field.
- *Flow pattern*: flow visualisation is obtained by video recording through the Perspex walls of the model.
- *Gate and valves positions*: the tailgate level z_{tg} is recorded by means of a cable-extension transducer. The valves are servo-controlled, and equipped with an angular position transducer.

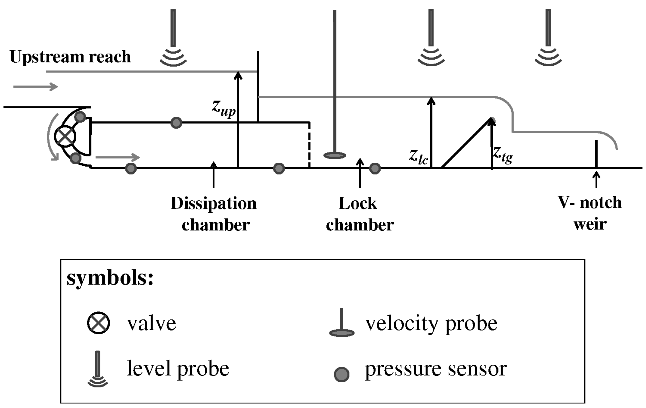


Figure 4 : Sketch of the measurement devices

Selected flow situations

The steady-state experiments are run for 4 different flow situations, each corresponding to a given water level z_{lc} in the lock chamber and a given discharge Q. The values of Q and z_{cl} are determined based on the discharge hydrograph computed by means of the 1D lock-filling model. The linear valve opening, the resulting discharge into the chamber and the water-level evolution in the chamber are illustrated in Figure 5 for a symmetric filling. From there, the 4 flow cases to be simulated by steady-state experiments are chosen: the selected points are indicated on the discharge curve. Each of these points thus corresponds to a given moment in the transient lock-filling process. In the present case, the valve is opened linearly in 6 minutes, for an initial level difference of 0.625 m. The different parameters prescribed on the scale model to achieve the selected flow situations are given in Table 1 for the symmetric filling and in Table 2 for the non-symmetric filling.

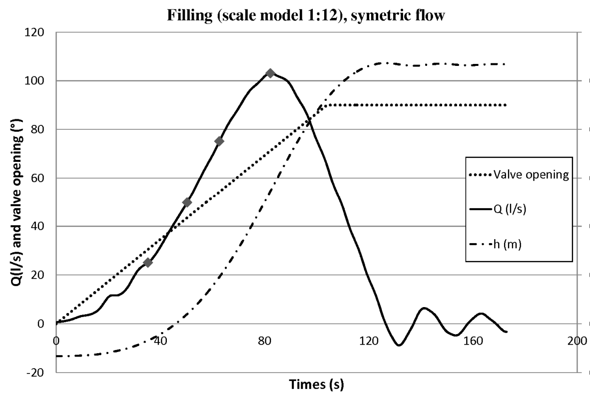


Figure 5 : hydrograph of the filling of the lock with symmetric flow

Table 1: Experimental values of the parameters of the 4 selected steady flow cases (symmetric flow)

Parameter	Case 1	Case 2	Case 3	Case 4
Q (l/s)	27.5	55.5	75.5	101.4
Valve (°)	29	43	53	73
Z_{cl} (mm)	334	359	413	547
Z_{up} (mm)	930	927	925	921
Z_L (mm)	358	415	487	654

Table 2: Experimental values of the parameters of the 4 selected flow cases (asymmetric flow)

Parameter	Case 1	Case 2	Case 3	Case 4
Q (l/s)	21.8	40.6	60.8	86.6
Valve (°)	37	50	61	81
Zcl (mm)	296	296	288	331
Zup (mm)	931	929	926	923
ZL (mm)	308	333	345	409

Velocity distribution measurement

For each of the two configurations of the dissipation chamber, measurements are performed according to the flow situations of Table 1 and Table 2. In a first series of experiments (M1), velocity measurements were performed on a vertical cross section located 0.2 m downstream of the outlet of the dissipation chamber, with a horizontal density of 10 points over the chamber width and a vertical spacing of 0.03 m. In a second series (M2), the number of measurement points was increased to 15 points over the chamber width, in a cross-section located 0.01 m downstream of the chamber outlet, the vertical spacing was kept to 0.03 m. Figure 6 shows the position of the two series and the measurement position.

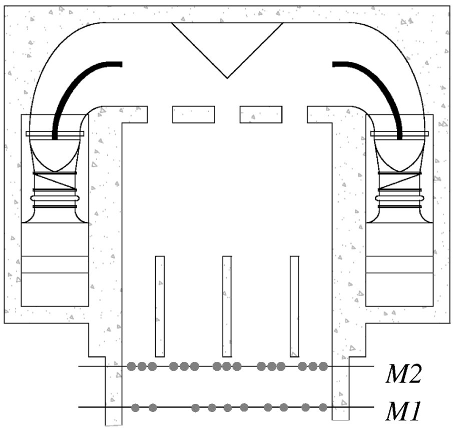
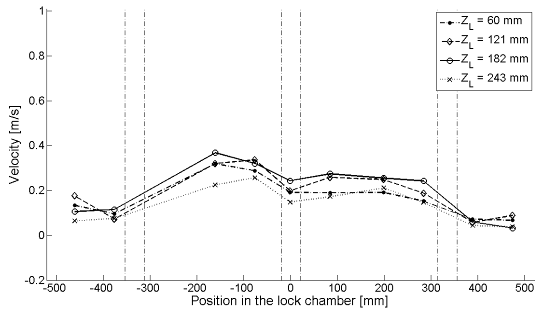


Figure 6 : Horizontal position of the velocity measurement.

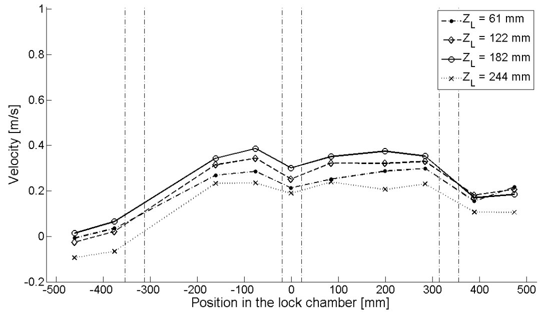
Influence of the dissipation chamber layout

A comparison between the two configurations is shown in figure 7 for a symmetric flow at $Q = 55$ l/s. The velocity profiles present significant similarities. Velocities are in the same order of magnitude with a maximum close to 0.4 m/s. The highest velocities are measured between 100 and 150 mm above the bottom of the lock chamber. The asymmetry that can be observed in both configurations is due to some valve positioning inaccuracy. However, from this first comparison, it can be concluded that the addition of a triangle does not affect significantly the flow distribution at the outlet of the dissipating chamber for symmetric conditions.

Similar results for asymmetric flow conditions at $Q = 60.8$ l/s are shown in Figure 8. Here, the influence of the triangular element (Figure 3b) can be clearly observed. In configuration 1 (figure 8a), the major part of the flow is concentrated on the right part of the lock chamber, opposite to the side where the valve is opened. The velocity is almost constant on a half lock width, with an average value of 0.45 m/s. On the other half lock, the velocity is close to 0 m/s. In configuration 2 (figure 8b), while the major part of the flow is still concentrated on the right part of the lock chamber, with a peak value at 200 mm on the right part, the velocity distribution is low but non zero on the other half-lock. Whereas the presence of the triangle in the dissipating chamber did not affect the flow in symmetric condition significantly, it slightly improves the flow in asymmetric condition by forcing a fraction of the water through the left part of the lock chamber.

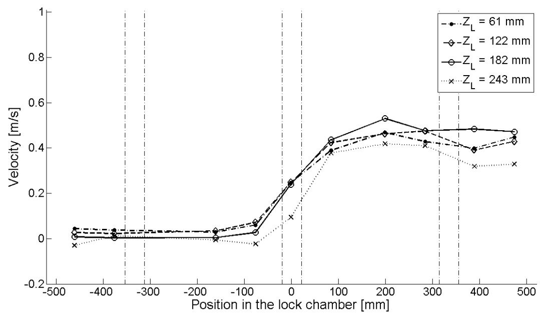


(a)

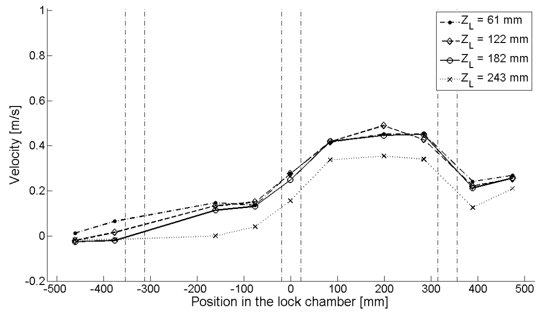


(b)

Figure 7 : Velocity distribution for symmetric flow (55 l/s). Configuration 1 (a) and 2 (b)



(a)

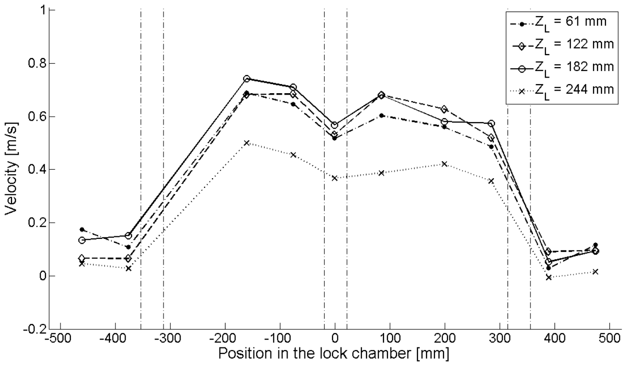


(b)

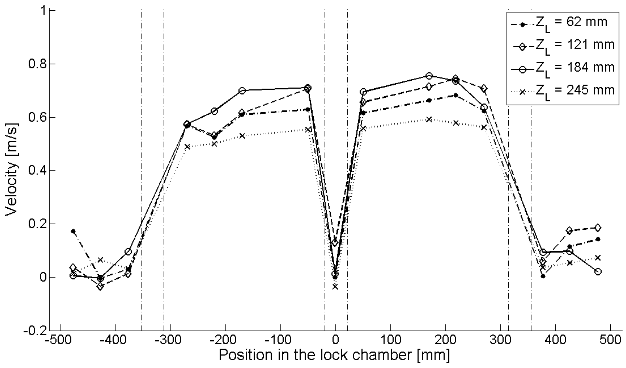
Figure 8 : Velocity distribution for asymmetric flow (60.8 l/s). Configuration 1 (a) and 2 (b)

Comparison between M1 and M2

Comparisons between the measurements for configuration 2 and for the two locations of the measurement grid show that the velocity distributions are very similar (figure 9), excepted immediately behind the guiding walls where the values measured according to M2 (1 cm downstream of the chamber outlet) are higher than the ones measured according to M1, i.e. further downstream, indicating the effect of vertical jet spreading in the lock chamber.



(a)



(b)

Figure 9 : Velocity distribution for symmetric flow (101.4 l/s). Measurements at 20 cm from the outlet of the dissipating chamber (a) and at 1 cm from the outlet (b)

Numerical simulations

Discretization of the dissipation chamber

A 2D numerical model of the dissipation chamber is made with the OpenFOAM® software. A depth-averaged model was chosen rather than a full 3D simulation at this first stage of the research to limit the computational time. The computational domain is represented in Figure 10. It is divided into 10,000 hexahedral cells, corresponding to an optimized mesh as it was observed that a higher level of refinement did not bring any significant improvement.

The software solves the incompressible RANS equations with a standard κ-ε turbulence model. The boundary conditions are a uniform velocity at the entrance of the dissipation chamber and a pressure condition at the outlet of the dissipation chamber, corresponding to the hydrostatic pressure in the lock chamber.

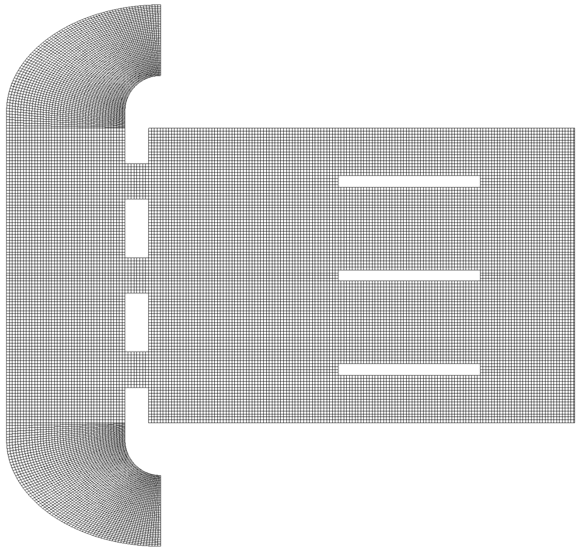
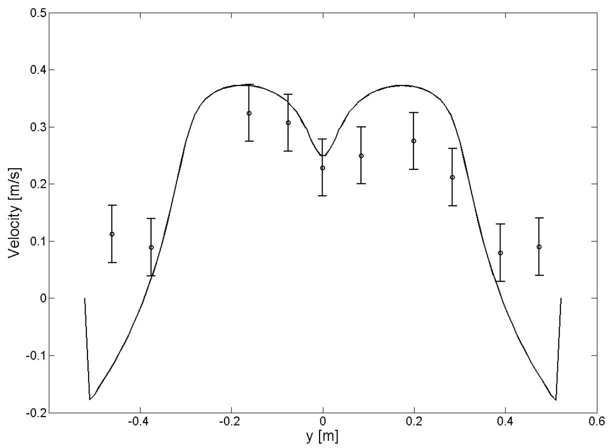


Figure 10 : Computational domain used for the 2D numerical model

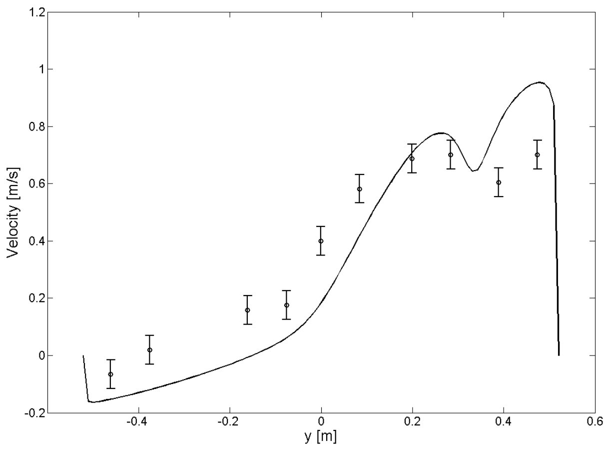
Comparison with the experiments

A comparison between the numerical and experimental results for configuration 1 is shown in Figure 11a for a symmetric flow with $Q = 55$ l/s and in figure 11b for an asymmetric flow with $Q = 86$ l/s. Overall, numerical and experimental results are in good agreement. Only in the region near the walls, velocities present more discrepancies. This might be due to the presence of a guide vane in the culvert elbows (Figure 3) that were not represented in the numerical model.

Computed results for configuration 2 are presented in Figure 12. As in the experiments (Figures 7 and 8), no significant effect of the triangular element can be observed for symmetric condition (figure 12a) while a clear improvement of the flow distribution is observed in asymmetric condition (figure 12b).

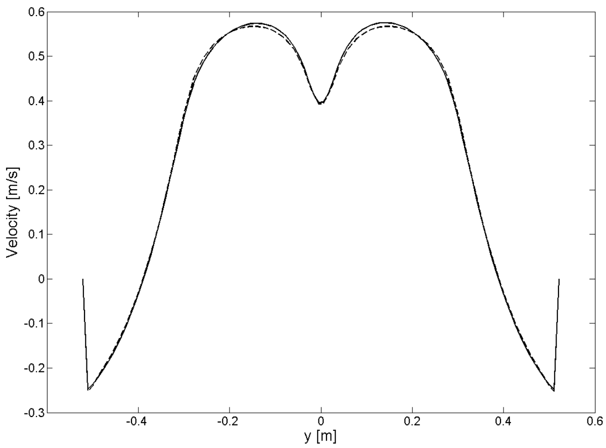


(a)

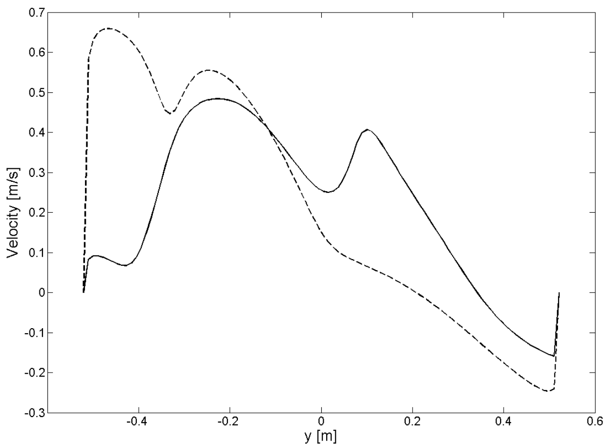


(b)

Figure 11: Comparison with the numerical model (line) and experimental measurement (symbols) for a symmetric flow with 55 l/s (a) and asymmetric flow with 86 l/s (b)



(a)



(b)

Figure 12: Comparison between configuration 1 (dot line) and configuration 2 (line) for symmetric (a) and asymmetric flow (b)

Conclusion

This paper presented some first results of the composite modelling approach used in the design of the Ampsin-Neuville lock. The key parameter is the design of the dissipation chamber in order to obtain a velocity distribution at the outlet of this chamber that is as uniform as possible.

The composite approach presented in this paper is a combination of 3 studies. A simplified 1D numerical model for the filling of a lock with loop culvert filling has been developed. This model allows determining operating points representative of the filling process. These points are used for experimental measurement on the scale model in steady-flow conditions.

The experimental measurements are compared to the predictions by a 2D depth-averaged numerical model. After validation, this model will be used to investigate further possible configurations of the dissipation chamber, in order to optimize to velocity distribution at the outlet.

Finally, full 3D numerical simulations are foreseen in future work to further validate the final design of the chamber by taking into account effects that are not represented in the 2D model.

Acknowledgement

The studies reported in this paper are co-financed by the European Union' Trans-European Network of Transport program (TEN-T), 2007-EU-300010-P, decision C (2008) 8141 of December 15, 2008. The sole responsibility of this publication lies with the authors. The European Union is not responsible for any use that may be made of the information contained therein.

References

Idel'Cik I.E. (1979). Memento des pertes de charge (*Spravotchnik po guidravlitcheskim soprotivleniam).* Paris, Eyrolle.

Vancaster P.H., Escarmelle A., Hiver J.M., Zech Y. (1999). *Coupled 1D and 2D modelling of a navigation-lock filling and emptying*. Proceedings 28th IAHR Congress, Graz, Austria.

PIANC (1986). *Final Report of the International Commission for the study of Locks*, pp. 23, 283-287.

Inland navigation commission (2009). *PIANC report N°106: innovations in navigation lock design*, pp. 36-43, 54, 104-105.