



HAL
open science

Investigation of the hydraulics in flooded housing estate

Augustin Doumic, Frédérique Larrarte, Rajae Rtimi, Nicole Goutal

► To cite this version:

Augustin Doumic, Frédérique Larrarte, Rajae Rtimi, Nicole Goutal. Investigation of the hydraulics in flooded housing estate. SimHydro 2021, 6th Conference Models for complex and global water issues - Practices and expectations, Jun 2021, Sophia Antipolis (en ligne), France. 9 p., graph., photos. hal-03264420

HAL Id: hal-03264420

<https://hal.science/hal-03264420v1>

Submitted on 18 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.

INVESTIGATION OF THE HYDRAULICS IN FLOODED HOUSING ESTATE

Augustin Doumic,
Ecole Nationale des Ponts et Chaussées & Saint-Venant Laboratory for Hydraulics (LHSV), 78400
Chatou, France
augustin.doumic@eleves.enpc.fr

Frédérique Larrarte¹
Université Gustave Eiffel & Saint-Venant Laboratory for Hydraulics (LHSV), 78400 Chatou, France
frederique.larrarte@univ-eiffel.fr

Rajae Rtimi
EDF R&D Laboratoire National d'Hydraulique Environnementale, 78400 Chatou, France
Rajae.rtimi@edf.fr

Nicole Goutal
EDF R&D Laboratoire National d'Hydraulique Environnementale & Saint-Venant Laboratory for
Hydraulics (LHSV), 78400 Chatou, France
Nicole.goutal@edf.fr

KEY WORDS

velocity distribution, emerged and submerged obstacles, square cylinders, flooded housing, experiments, numerical modelling

ABSTRACT

Climate change is currently taking place at the scale of our planet, leading to a violent response from the environment, particularly with regard to the water cycle. One of the risks is already floods. As an example, in France, the average annual cost of damages caused by floods (and insured under the Natural Disaster plan) amounts to 520 million euros. In metropolitan France, nearly 17 million people, about a quarter of the national population, are exposed to the risk of flooding by overflow according to the preliminary flood risk assessment carried out by the State in 2011. And with climate evolution, floods will become more frequent and more intense. One challenge during floods is to be able to access to people refuge over buildings. In order to investigate the hydraulic context generated by the grouped obstacles representative of a submerged housing estate, both an experimental and numerical study is being conducted. We experimentally study the flow in the presence of rows of macro-roughnesses representative of a slightly emerged and of a submerged housing estate, and this for situations representing both current and more intense floods. Measurements of the velocity field will provide information on currents distribution that might affect an evacuation process. In parallel a numerical study is also conducted with the open source code Saturne. The numerical results are compared to experimental results and this preliminary study opens the way to a systematic investigation.

1. INTRODUCTION

Water flows are interacting with the bottom of the water course, either natural or artificial, so the influence of the bottom and walls roughness have been investigated for long time. To give just some examples, [1] proposes formula to determine the flow resistance, [2] used laboratory experiments to investigate the influence of composite roughness on flow resistance, [3] investigate the wide range of context made of the British rivers. More recent works exist on such a topic with emergent applications ranging from fish passes [4] to urban floods [5-8]. The present paper deals with this context. Indeed,

¹ corresponding author

according to the Ministry of Ecological Transition (2019 data), 17.1 million permanent residents are exposed to flooding by river overflow, including 16.8 million in metropolitan France. In addition, 1.4 million residents are exposed to the risk of marine submersion. Thus knowing how the flow behaves withing the housing settlement in relation to the submergence rate can be or practical consequence for safety services [8] have focused on high submergences, [9] on smaller ones. However experimental studies remain scarce. In this preliminary study we focus on the transition between emerged and slightly submerged situations. This experimental study will also provide benchmark data. The paper will first describe the experimental set-up and methodology. The numerial modeling is pesented before the results are compared.

2. EXPERIMENTAL SET-UP

The experiments aim at reproducing the context of human settlement in recently urbanized areas on plain areas (figure 1) such as main French rivers catchments. We also consider oceanic climate floods rising quite slowly and producing pseudo static situations such as the famous Paris 1910 flood that is a centenal one. In such context, the private houses are flooded but their roof remains out of water. With the climate evolution, thoses building could be submerged, modifying the way the inhabitants can be rescued soi t is of great interest to investigate how the flow behaves in this intermediate context.

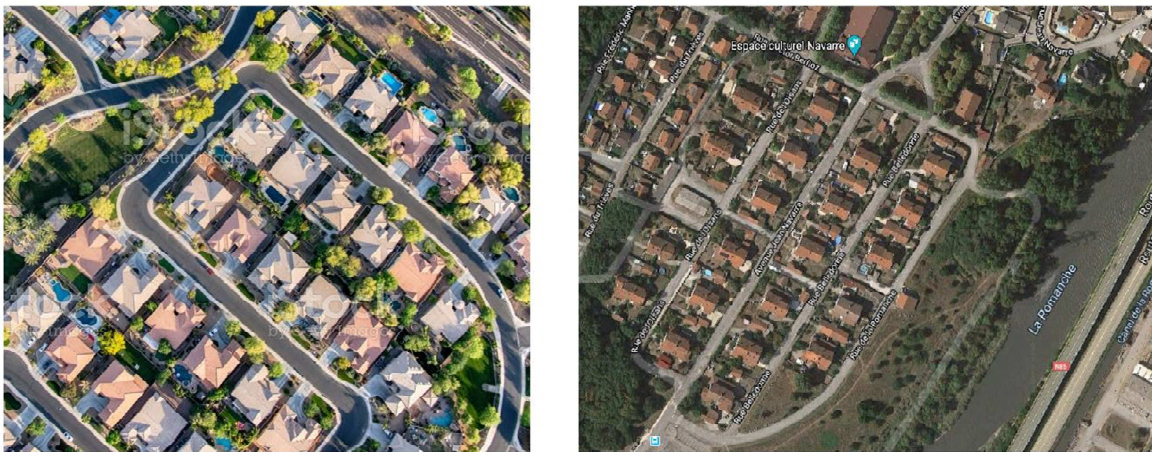
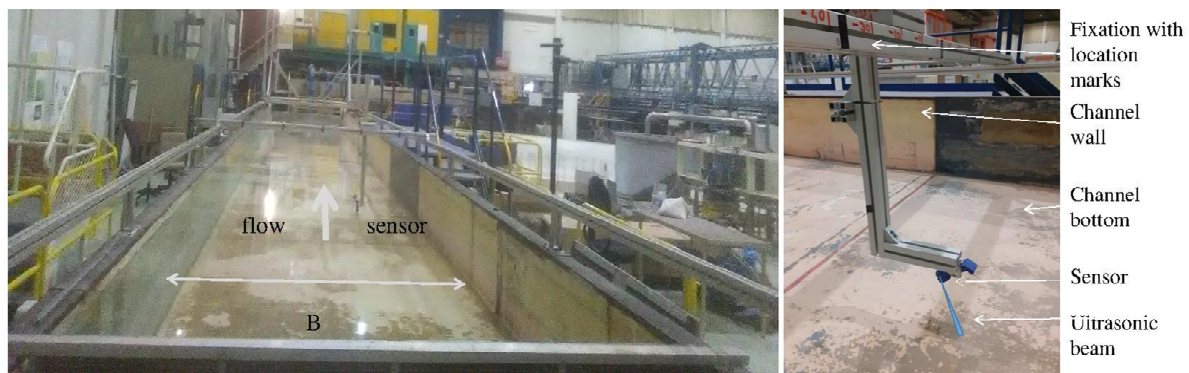


Figure 1: Exemples of aligned housings (photographs from internet)

The experiments have been carried out at the Laboratoire d'Hydraulique Saint Venant in an open channel which is 16 m long, 2 m wide and 0.5 m high (figure 2a). A honeycomb has been placed at the pump outlet to calm the flow. Velocity measurements were performed with a Ubertone UBLab ultrasonic profiler (Figure 2b) fixed at the free surface.



a) General view of the test facility

b) ultrasonic profiler

Figure 2: Experimental channel and instrumentation.

The planners propose of course a very wide range of subdivision layouts. In the littérature the experiments are mainly made with aligned buildings [10-12] or staggered ones [13]. In this study, we take 2 configurations, one with aligned houses (figure 3a) and the other with staggered buildings (Figure 3b). The distance between the buildings is taken equal to the building size in both l_x and l_y steamwise and transverse direction. This may be assimilated to a settlement with one level houses for a family with a road or a peace of garden between two houses. The pattern is made of wooden cubes carefully glued on the channel bottom. Each cube has a square section with a side length $L_x=L_y= 0.1$ m.

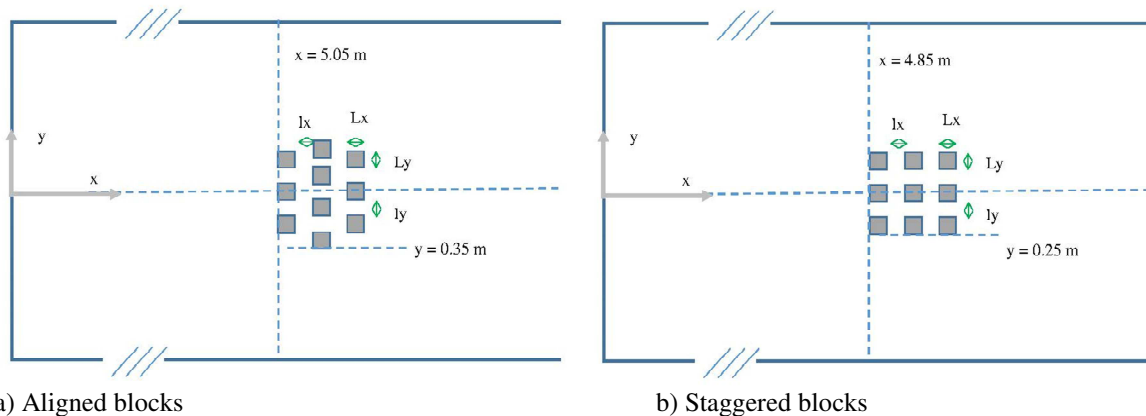


Figure 3: Experimental configurations.

In order to investigate the effect of the emergence or submergence, the flow depth h varies from 95 to 150% of the building height D . The inflow velocity is taken equal to 0.23 m/s. The Froude number Fr ranges between 0.16 and 0.24 and the Reynolds number between 21850 and 34500, meaning a turbulent flow, with $Fr=Ud/(gD)^{1/2}$ and $Re = Ud*h/\nu$, where g is the gravitationnal acceleration. The block coefficient C_b ranges between 0.08 and 0.16 with C_b being the ratio of the surface of the first line of blocks on the wet area of the experimental flume. Table 1 presents all the experimental conditions.

h	Ud	Q	h/D	Fr	Re	C_b	
0.095	0.23	0.044	0.95	0.24	21850	0.16	aligned
0.105	0.23	0.048	1.05	0.23	24150	0.14	aligned
0.130	0.23	0.060	1.30	0.20	29900	0.12	aligned
0.150	0.23	0.069	1.50	0.19	34500	0.10	aligned
0.095	0.23	0.044	0.95	0.24	21850	0.16	stagered
0.150	0.23	0.069	1.50	0.19	34500	0.10	stagered

Table 1 : experimental cases

3. NUMERICAL STUDY

The numerical study was made using Code_Saturne. that is a free software for numerical simulation in fluid mechanics developed by EDF's Research and Development division. Based on a co-located finite volume approach that accepts meshes of any size and any type, Code_Saturne allows to model incompressible or dilatible flows, with or without turbulence or heat transfer.

The Navier-Stokes equations form a mathematical model derived from the laws of conservation of mass and conservation of the quantity of movement applied to a volume. These describe three-

dimensional flows and within the framework of our study, we will take the case of a fluid incompressible real subject to the only gravity field. The equations of Navier-Stokes are reduce to :

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

$$\rho \left(\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} \right) = -\frac{\partial P}{\partial x_i} + \frac{\partial D_{ij}}{\partial x_j} + \rho g_i \quad (2)$$

with ρ , the density of the fluid, u_i components of the velocity vector, P the pressure and D_{ij} is the shear stress tensor. By definition, D_{ij} is expressed as :

$$D_{ij} = \mu \left[\frac{\partial \rho u_i}{\partial x_j} + \frac{\partial \rho u_j}{\partial x_i} \right] \quad (3)$$

Where μ is the dynamic viscosity considered constant in this study

For this preliminary study, several hypotheses were chosen to simplify the numerical modeling of this complex flow without influencing the preponderant physical phenomena. The first hypothesis considers the free surface as a fixed surface modelled as a symmetry boundary condition. So the height of water D is constant throughout the field. Indeed, this hypothesis is justified by the fact that the experiments were performed on a uniform flow at a macroscopic level. The second hypothesis is that the slope of the canal is not taken into account.

In order to carry out a fine modeling of the turbulence with the available calculation means and optimize the computing and memory time, it is interesting to reduce the geometry of the hydraulic channel in a two-row pattern of two houses, with periodicity conditions in the longitudinal direction Ox and transverse Oy . The condition of periodicity consists in reinjecting all variables values from the output limit face into the input face of the domain. It thus allows to obtain an infinite configuration in length and width. Indeed, the established regime of experience justifies the condition of periodicity in the Ox direction.

Large Eddy Simulation (LES) solves the filtered Navier-Stokes equations: only the large-scale vortex structures, where most of the turbulent energy is produced, are computed explicitly, and the small scales, where most of the turbulent energy is dissipated, are modeled. This method has been chosen because it allows to directly access to the instantaneous fields and turbulent fluxes. It offers a very good compromise between computational times and accuracy of the turbulent flows.

As shown on figure 4, the numerical profile in the middle of the fast vein is consistant relatively to the experimental profile of Oukacine et al (2020).

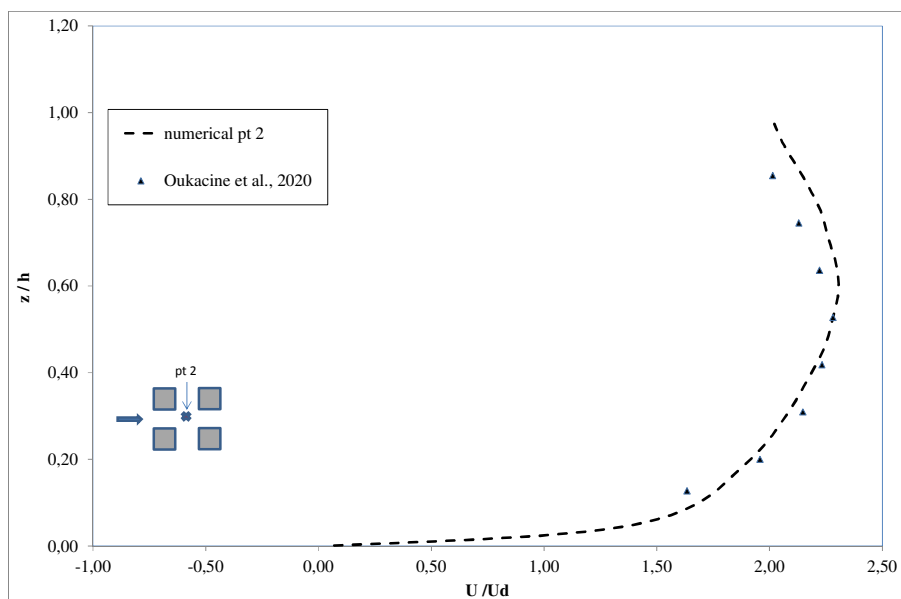


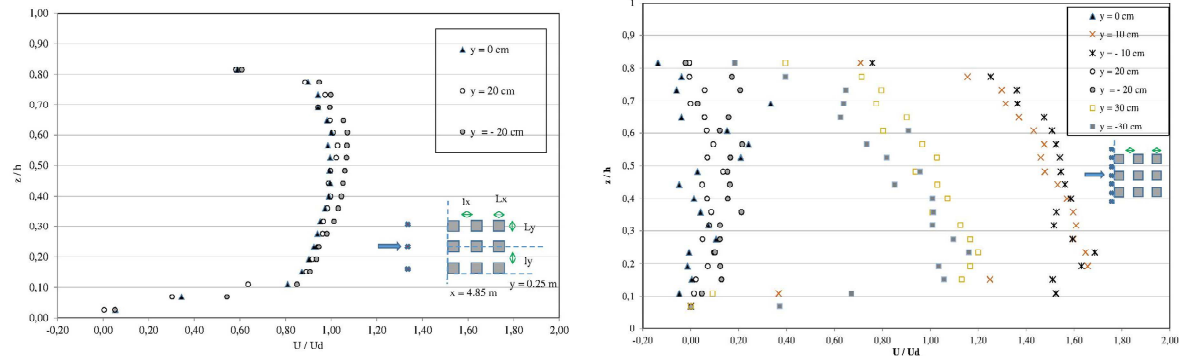
Figure 4: Comparison of Large Eddy Simulation (LES) with the experimental results of Oukacine et al (2020).

4 RESULTS

The velocity profiles have been measured for both aligned and staggered configurations and various flow rates corresponding to an emerged and 3 submerged

4.1 Aligned configurations

Far upstream of the blocks the flow is symmetrical and it is not affected by the array (Figure 5a). Once is the array area, if the symmetry remains, the flow is highly disturbed and the velocity profiles are rather complex.

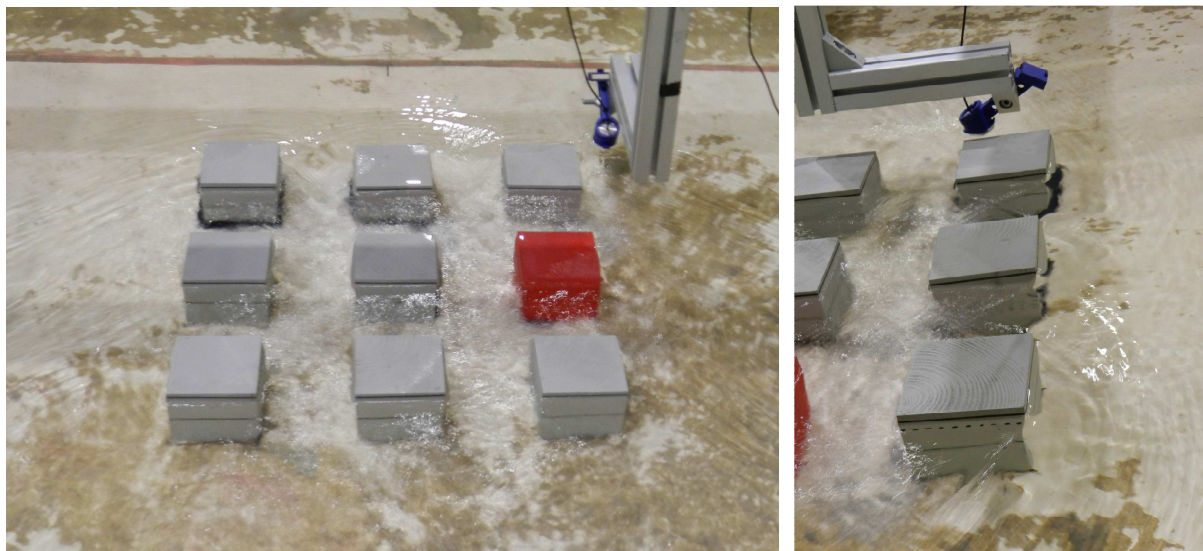


a) upstream of the flow

b) Inside the array

Figure 5: Checking of the flow symmetry (the cross indicate the measurement points) at $h/D = 95\%$

It is interesting to notice that each lateral block has a bow wave that contribute to a global one located at about $1.2 \cdot l_x$ upstream of the central block and making a pattern (Figure 6a) similar to a bow wave with a stepped vertical variation (Figure 6b) similar to the free surface shock waves discussed by [14].



a) View of the upstream extension of the wave development

b) View of the vertical

Figure 6: Upstream influence of the blocks for $h/D = 95\%$.

Figures 7 to 10 present the velocity profiles for emerged and slightly submerged situations. The velocity profiles are affected by the block even between blocks and upstream of the first one but this influence decreases with the water level. Within the array, the velocity between blocks line (points at $y = 10$ cm) is accelerated, making a fast vein. The profiles present a dip phenomenon for the lower submersion rate and become more complex when the water level increases. This influence decreases immediately outside of the array ($y = -30$ cm) and the profiles are less regular. In front of one of the

block, the profiles are highly distorted and become more regular when the water goes over the blocks as for the submersion rates of 130 and 150%.

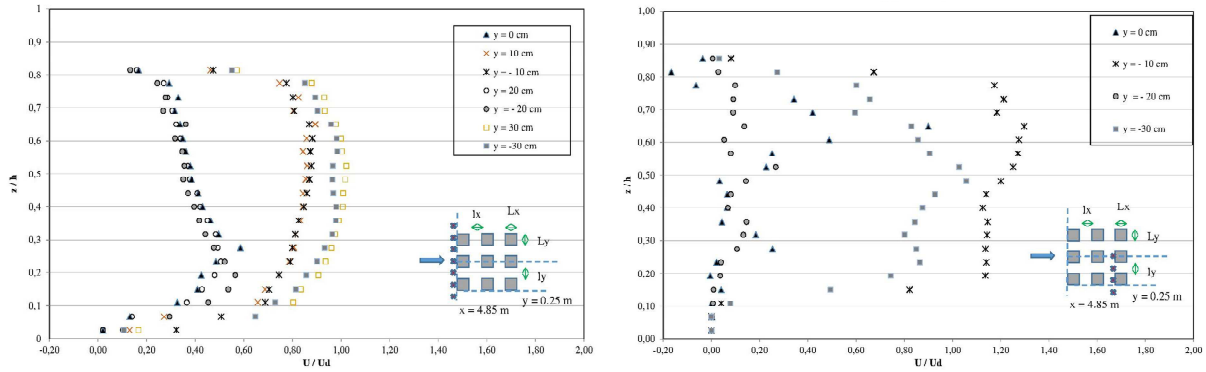


Figure 7: Influence of the blocks on the velocity profiles for $h / D = 95\%$

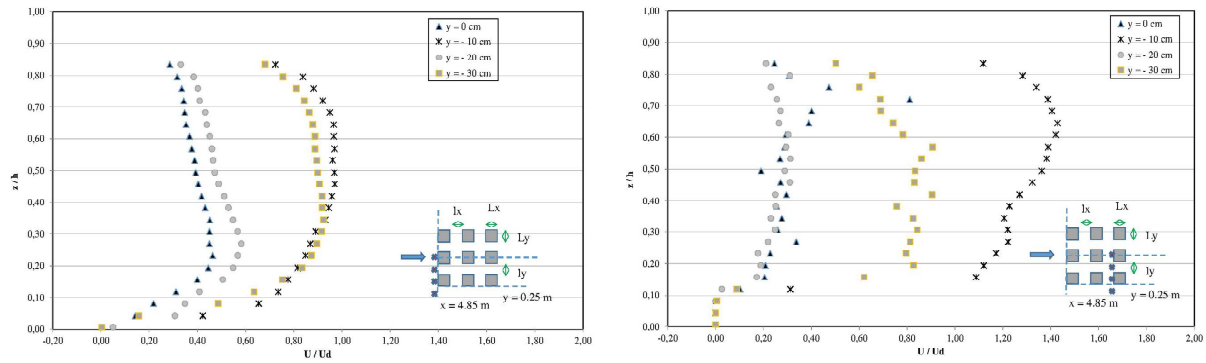


Figure 8: Influence of the blocks on the velocity profiles for $h / D = 105\%$

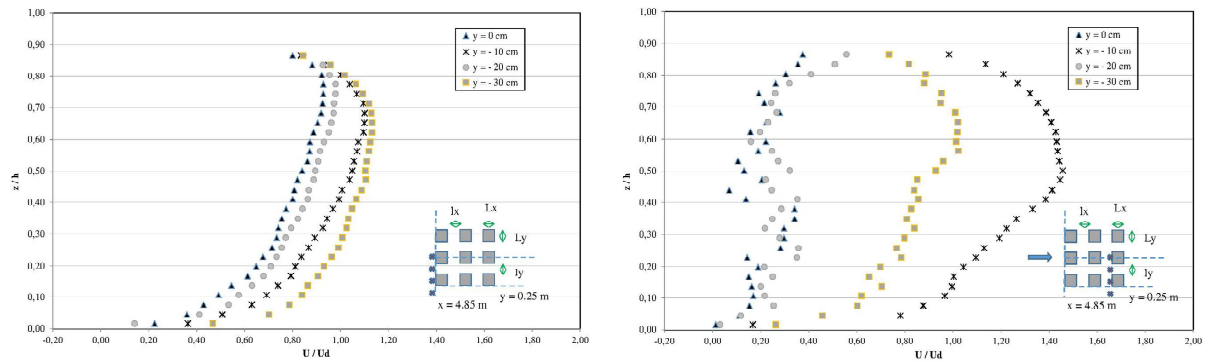


Figure 9: Influence of the blocks on the velocity profiles for $h / D = 130\%$

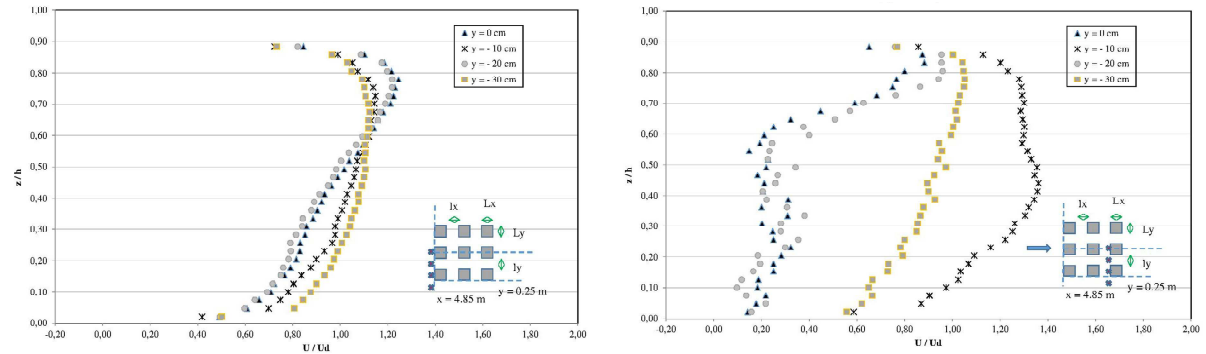


Figure 10: Influence of the blocks on the velocity profiles for $h / D = 150\%$

Figures 11 present the evolution of the velocity profiles in the fast vein with increasing water levels. It is interesting to notice that at a given location the profiles are almost the same with a maximum velocity 50% higher than the mean velocity U_d and located at 60% of the water level. Farther downstream, the profiles remain very similar with a maximum velocity about 40% higher than the mean velocity U_d and located at 55% of the water level.

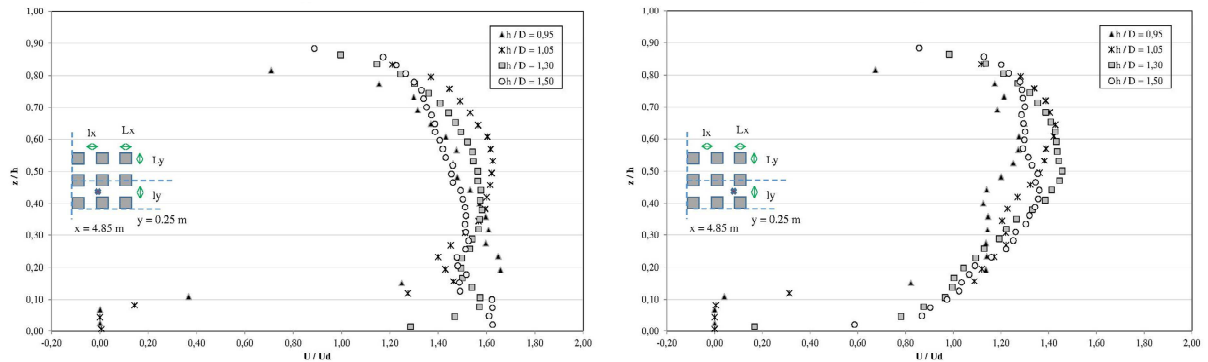
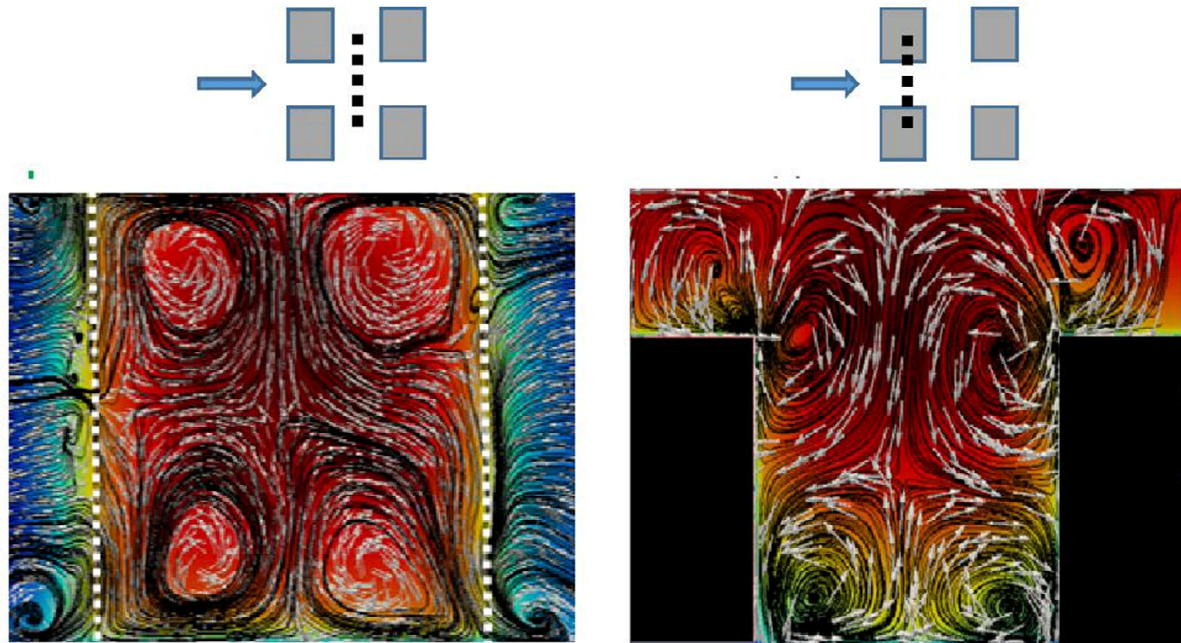


Figure 11: Influence of water level on the velocity profiles in the fast vein.

The profiler used for this preliminary study gives access to the streamwise component of the velocity along a profile that is not strictly vertical. In the same time, visual observations during the experiments have shown that the flow is rather complex. The numerical simulations have shown that strong secondary currents exist in the fast vein for the emerged case (Figure 12a) but also for the submerged case with the existence of another set of vortices above the blocks (figure 12b).



a) For $h / D = 93\%$

b) for $h / D = 148\%$

Figure 12: Secondary currents.

4.2 Staggered configurations

The staggered configuration is made on two lines of 3 blocks and a line of 4 blocks between them. Figures 13 and 14 present the velocity profiles for emerged and slightly submerged situations. The flow is canalized between the upstream blocks, forming a fast vein as for the aligned configuration (Figures 13b and 14b). But the blocks of the second line make as a barrier to the flow and then, downstream, the flow is highly complex as presented in Figures 13b and 14b.

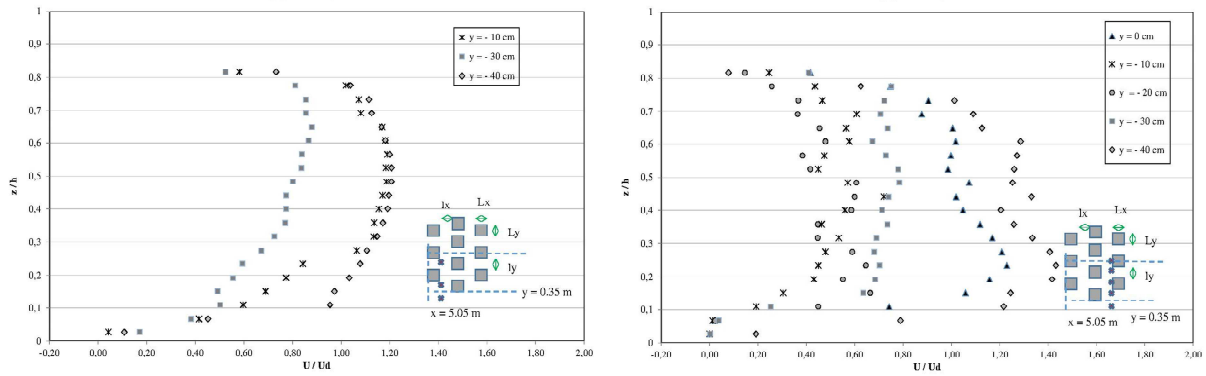


Figure 13: Influence of the blocks on the velocity profiles for $h / D = 95\%$

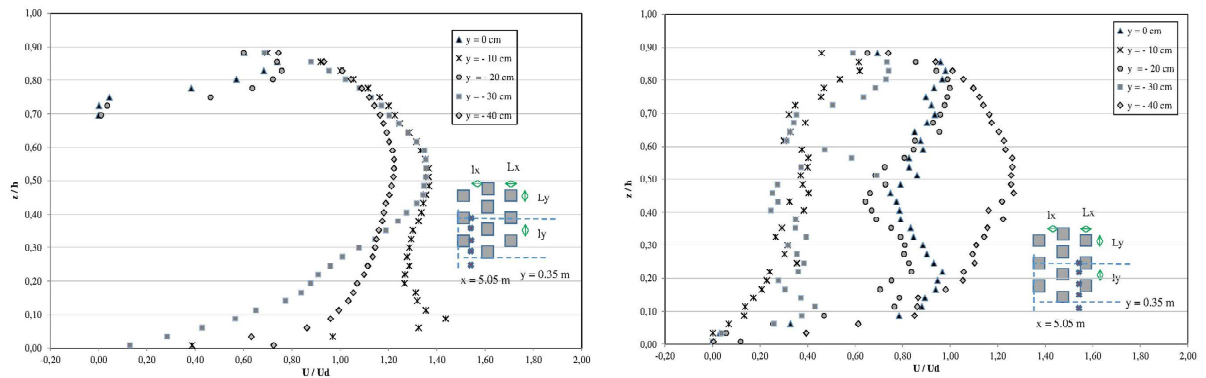


Figure 14: Influence of the blocks on the velocity profiles for $h / D = 150\%$

5. CONCLUSIONS

The present communication deals with the settlements exposed to flooding situations from emerged to slightly submerged situations. First the experimental configurations corresponding to an array of aligned blocs and an array with staggered blocks have been described. The main elements of the numerical study are summarized and the results are positively compared with experimental results from the littérature.

The experiments have shown the influence of the blocks on the flow at the vicinity of the arrays and in particular the existence of a fast vein between the lines of aligned block.. Visual observations during the experiments have shown that the flow is rather complex but the profiler used is not convenient for measuring the flow in a vertical plane perpendicular to the main streamwise velocity.

This preliminary study will be continued to improve the experimental set-up for measuring the 3 components of the velocity and their turbulent fluctuations. The influence of the distance between blocks but also their square or rectangular geometries will be investigated.

NOMENCLATURE

$Ar = B/h$	Aspect ratio in the transverse direction	(-)
B	channel width	(m)
$D = 0.10$	Obstacles height	(m)
D_{ij}	the shear stress	($kg \cdot m^{-1} \cdot s^{-2}$)
$Fr(h) = Ud / (gh)^{1/2}$	Froude number	(-)
h	Water level	(-)
$Lx = Ly = 0.10$	Distance centre to centre from two adjacent square cylinders	(m)
$lx=ly = 0.10$	Width of the square base obstacle	(m)

P	pressure	(Pa)
Q	discharge	(m ³ .s ⁻¹)
ρ	the density of the fluid	(kg.m ⁻³)
Ud	Q / (B*h)	(m.s ⁻¹)
u _i	components of the velocity vector	(m.s ⁻¹)

ACKNOWLEDGEMENTS

The authors thank EDF R&D for given them access to the experimental facilities.

REFERENCES AND CITATIONS

- [1] Keulegan, G. H. (1938). Laws of turbulent flow in open channels. *J. Res. Nat. Bur. Stand.*, **21**(6), 707–741.
- [2] Einstein, H. A. & Banks, R. B. (1950). Fluid Resistance Of Composite Roughness. *Transactions , American Geophysical Union*, **31**, Number 4, 603 - 610
- [3] Bathurst, J. (1985). Flow resistance estimation in mountain rivers. *Journal of Hydraulic Engineering*, **111** (4):625-6430
- [4] Cassan, L.; Tien, T. D.; Courret, D.; Laurens, P.; Dartus, D. (2014). Hydraulic Resistance of Emergent Macroroughness at Large Froude Numbers: Design of Nature-Like Fishpasses, *Journal of Hydraulic Engineering*, **140** (9): 04014043
- [5] Velickovic, M., Zech, Y. & Soares-Frazão, S. (2017). Steady-flow experiments in urban areas and anisotropic porosity model. *Journal of Hydraulic Research* **55**(1), 85–100.
- [6] Guillen-Ludena, S., Lopez, D., Mignot, E. & Rivière, N. (2020). Flow resistance for varying density of obstacles on smooth and rough bed. *Journal of Hydraulic Engineering* **146**(2), 04019059.
- [7] Akutina, Y, Eiff, O., Moulin, F.Y., Rouzes M. (2019). Lateral bed-roughness variation in shallow open-channel flow with very low submergence. *Env Fluid Mech*, 2019:1–23
- [8] Chagot, L., Moulin, F. Y., Eiff, O. (2020). Towards converged statistics in three-dimensional canopy-dominated flows, *Experiments in Fluids* 61:24
- [9] Rouzès M, Moulin FY, Florens E, Eiff O (2018) Low relative-submergence effects in a rough-bed open-channel flow. *J Hydraul Res* 57:139–166
- [10] Macdonald, R.W. (2000). Modelling the mean velocity profile in the urban canopy layer. *Bound Layer Meteorol*, **97**(1):25–45
- [11] Yen, S. C. & Liu, J. H. (2011). Wake flow behind two side-by-side square cylinders. *International Journal of Heat and Fluid Flow*, 32(1):4
- [12] . Oukacine, M., Rtimi, R., Goutal, N., Loizeau, V., Benhamadouche, S., Proust, S., Larrarte, F. (2020). Large Eddy Simulation for flows through emerged or slightly submerged square cylinders, *River Flow 2020*, Delft, 7-10 juillet
- [13] Herbich, J. B. & Shulits, S. (1964). Large-scale roughness in open-channel flow. *Journal of the Hydraulics Division* **90**(6), 203–230
- [14] Miyata, H., Inui, T. (1984). Non linear ship waves, *Advances in Applied Mechanics*, **24**