Benchmark of hydrodynamic models for urban flooding modelisation

G. Dellinger, Associate Professor[1*], P. A.Garambois, A.P.[2], G. Isenmann A.P. [1] , P. Finaud-Guyot A.P. [3], F. Lawniczak A.P., C. Choley PhD. [1], P. François A.P.[3], J. Vazquez, Full Professor[1]

[1]Université de Strasbourg, CNRS, ENGEES, ICube UMR 7357, Strasbourg, France
[2]INRAE, Aix-en-Provence, France
[3]Hydrosciences Montpellier, Polytech Montpellier, Montpellier, France
[4] Université de Strasbourg, CNRS, ICube UMR 7357, Strasbourg, France
**Corresponding author email: guilhem.dellinger@engees.unistra.fr*

Highlights

- Hydrodynamic data (h,V,Q) from an experimental rig (5m x 5m) representing a typical urban geometry are presented.
- 2D and 3D numerical simulations are confronted to the experimental data.

Introduction

The construction of urban areas in floodplains makes these areas highly vulnerable to flooding. The consequences can be particularly severe with significant material damage and human losses. To predict the potential impacts of a flood, in particular for extreme events, numerical modelling tools are widely used nowadays. The software traditionally used to study floods are mainly based on river modelling (1D or 2D flows). However, in highly urbanized areas, many parameters can influence the flows and lead to 3D hydrodynamics. We can mention, for example, the exchanges with the sewerage networks, the infiltration in the buildings or a branched street network. This last point will be studied in this article. The difficulty lies especially in the interconnection of these junctions: the flow pattern in a junction can influence the hydraulic behavior of the other junctions. The aim of this work is to carry out a comparative study of modeling software regarding their capacity to reproduce the flow in an urban aera (water heights, velocity fields and flow discharges). The software used in this study are the following: *Dassflow-2D*, *Telemac2D* and *OpenFOAM*. The numerical results are compared with experimental data collected on the ICube laboratory pilot consisting of a street network.

Methodology

Experimental rig

An experimental rig was built in ICube laboratory (Strasbourg, France) to study the hydrodynamics of urban flood propagation. This experimental device is composed of a street network that corresponds to the district geometry proposed during the Rives and Hyville projects (Lipeme Kouyi et al. 2009). This district is assumed to be representative of a typical urban geometry. The device is then composed of streets with different widths (5 to 12 cm) and of crossroads with various angles (cf. figure 1). The device is built on a 5m x 5m plan and corresponds to a 1km x 1km real district. The details of the scaling can be shown in (Finaud Guyot et al. 2018 and Araud 2012). The device has been built such that water flows in a closed circuit. The water is pumped from a tank and injected into the different inlets of the device. It is possible to impose the upstream flow discharge of each street (between 0,5 l/s and 4 l/s). Each outlet of the device is connected to a channel equipped with a calibrated weir. The downstream flow discharge of each street can then be measured with an accuracy lower than 3%. A scheme of the global circuit is exposed in figure 2. An automatically moving camera films the free surface, and the gathered images are then processed to determine the water depth profile. To determine instantaneous vertical velocity profiles, an ultrasonic sensor is used. The latter can be placed automatically at any location of the device.

The flowrate can then be determined in each street of the device by integration of the profiles: three profiles are used in the narrowest street and five in the largest.

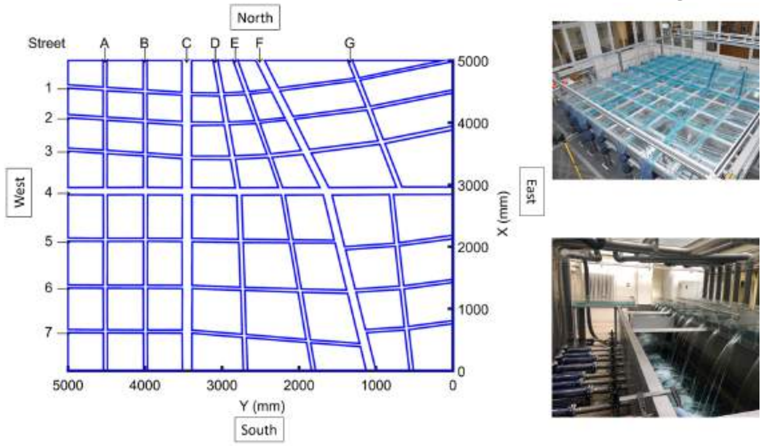

Figure 1 Plan and photos of the experimental rig

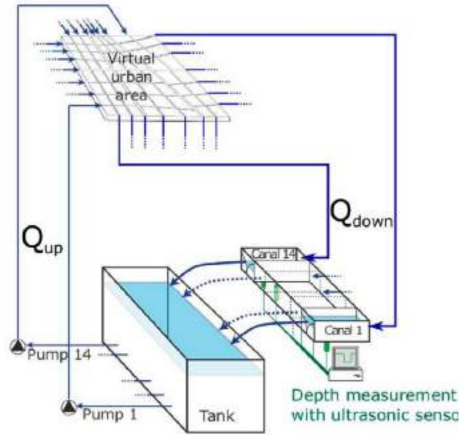

Figure 2 Scheme of the water cycle in the experimental model

Numerical modelling

The 2D calculation codes solve the 2D shallow Water equations. It can be noticed that, whereas the simulations with *DassFlow* are achieved without any turbulent model, the modelling with *Telemac2D* are performed with the $k - \epsilon$ turbulent model. A flow discharge is imposed at each inlet of the hydraulic system. A rating curve is given for each outlet. These rating curves were measured experimentally. Eventually, sensitivity analysis were done in order to determine the mesh size and the time steps.

The *OpenFOAM* calculation code is used to achieve the 3D simulations. This code was used to solve the Reynolds Averaged Navier-Stokes (RANS) equations with the *interFOAM* solver. The $k - \omega\ SST$ turbulent model was chosen to close the system of equations. The inlet flow discharges were imposed but the outlets were simulated numerically (no rating curves imposed). As previously, a sensitivity analysis was done to determine the mesh size. An adaptative time step was used with a maximum value of Courant Number equal to one.

Results and discussion

The figure 3 exposes the relative error in discharge for the three numerical models. It can be shown that the relative error is generally low with values under 10%. The numerical results are especially accurate for the widest streets where the flow rates are the highest. The error can be higher than 25% in the narrow streets where the flow discharges are low. As expected, it can be noticed that the difference between numerical and experimental values increase as we move downstream. It can be explained by the fact that the errors propagate from upstream to downstream as the upstream flow discharges are imposed. Eventually, the sum of relative errors is equal to 9.91 for *DassFlow*, 8.25 for *Telemac2D* and 7.74 for *OpenFOAM*. It shows that the 3D simulation allows to be more accurate in the prediction of the flow distribution. The difference between the two 2D models can be explained by the fact that *Telemac2D* includes a turbulent model and not *DassFlow*.

The figure 4 exposes the numerical and experimental horizontal velocity profiles. It should be noticed that the velocities are averaged over the vertical axis for the values coming from the 3D model and from the experiments. The values from *Telemac2D* and *OpenFOAM* are very close, and they both reproduce well the experimental profiles. While the experimental profiles become more and more symmetrical as we move downstream, the profiles from *DassFlow* keep its asymmetrical shape. It is explained by the fact that the latter does not have a turbulence model. These results highlight the need for a turbulence model to best reproduce the velocity fields in this kind of hydraulic system.

Eventually, the numerical and experimental water dephts were confronted. Once again, *Telemac2D* and *OpenFOAM* reproduce well the experimental values with a relative error lower than 5%. For *DassFlow*, the water depths are generally lower than the experimental ones and the relative error can reach 10%. Once again, it is supposed that this difference is due to the absence of turbulence model.

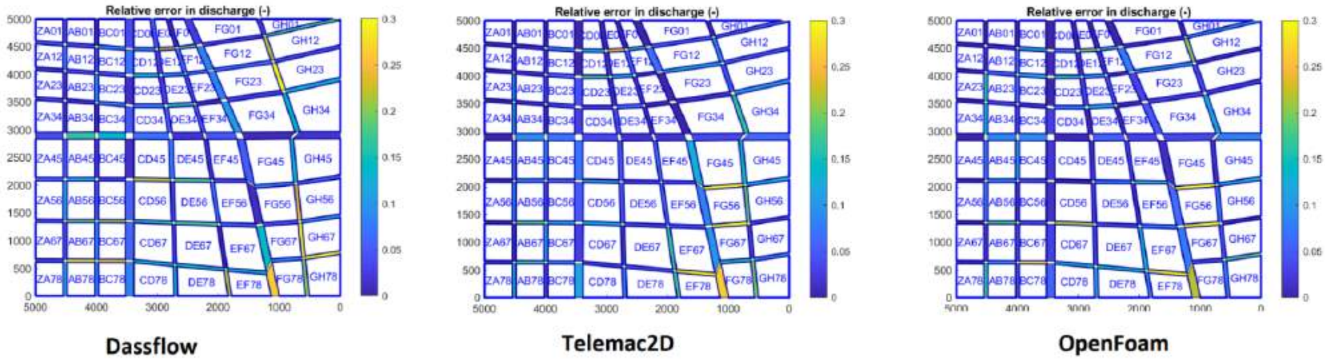

Figure 3 Map of the relative error in discharge

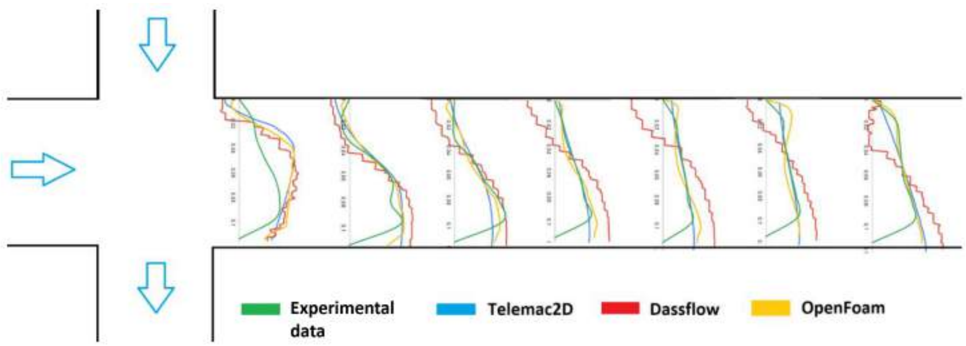

Figure 4 Horizontal velocity profiles downstream to the junction C-4

Conclusions and future work

An experimental rig that represents a district of a typical city is installed in the ICube laboratory of Strasbourg. The experimental data measured in the device were used to compare the performance of 2D (*DassFlow* and *Telemac2D*) and 3D (*OpenFOAM)* numerical models in reproducing the hydrodynamic of urban floods. Firstly, it appears that all the models were able to well reproduce the flow distribution. The modelled values of velocities from *Telemac2D* and *OpenFOAM* are very close to the experimental ones. Nevertheless, the heights and velocities given by *DassFlow* differ from the experiments. It is assumed that it is due to the absence of turbulence model. Eventually, this work has shown that 2D shallow water models coupled with a turbulent model is efficient enough to model well flooding in urban areas. Future work will focus on studying the ability of these models to reproduce urban flooding in transient configurations.

References

Lipeme Kouyi, G., D. Fraisse, N. Rivière, V. Guinot, and B. Chocat. 2009. “Onedimensional Modelling of the Interactions between Heavy Rainfall-runoff in an Urban Area and Flooding Flows from Sewer Networks and Rivers.” Water Science and Technology : A Journal of the International Association on Water Pollution Research 60 (4): 927-34.

Finaud-Guyot, P., P.-A. Garambois, Q. Araud, F. Lawniczak, P. François, J. Vazque and R. Mosé. 2018. «Experimental Insight for Flood Flow Repartition in Urban Areas ». *Urban Water Journal* 15 (3): 242-250.

Araud, Q. 2012.”Simulations des écoulements en milieu urbain lors d’un évènement pluvieux extrême. ”. Thèse de doctorat, Université de Strasbourg