SIMULATING THE HYDRODYNAMICS OF SEWER-GRATES USING A 2D-HYDRAULIC MODEL

Marcos Sanz-Ramos[1a]
Mailing address
marcos.sanz-ramos@upc.edu

Jackson Téllez-Álvarez[a], Ernest Bladé[a], Manuel Gómez-Valentín[a]
Mailing address
jackson.david.tellez@upc.edu, ernest.blade@upc.edu, manuel.gomez@upc.edu

[a]Institut Flumen (UPC-CIMNE). C/ Jordi Girona 1-3, 08034, Barcelona (Spain)

KEY WORDS

Urban drainage, Inlet grates, Infiltration method, 2D modelling

ABSTRACT

Sewer inlets are one of the most important elements for avoiding floods in urban areas, capturing runoff and introducing into the sewer system. Traditionally the inlets have been designed by resistance criteria, without including or analysing their hydraulic efficiency. Currently CFD techniques plays an important role in order to assess more efficient inlets, minimizing the need for flume experiment and taking into account the hydraulics for their design. 2D models are good enough to represents the hydrodynamics, which is mainly 1D on the street and 2D on the surroundings of the inlet. These models cannot properly capture 3D phenomena over the inlet, the discharge through the inlet is not well represented. This work presents different techniques, based on the similarity between the hydrological infiltration models and the discharge captured by the inlet, aiming to enhance the 2D model Iber to adequately reproduce the discharges captured by the grates, even without representing the complex three-dimensional hydrodynamics around and over the inlet. The results fit well with the observed data, properly reproducing the hydraulic structures as hydraulic jumps, recirculation and cross waves.

1. INTRODUCTION

Sewer inlets are drainage elements generally used in urban areas to capture the runoff and introduce it into the underground drainage network in safety conditions. Their location and frequency (e.g. inlets per ha or per 100 m of street) must be adequate enough to avoid or minimize floods [1]. However, the most important point is how much discharge are they able to capture in relative and absolute terms (not only in terms of flood so to dimension the sewer system).

Traditionally, the inlets have been designed to resist the street loads (road traffic) and in order to warrant the pedestrians' safety (hole sizes). These designs are, in many occasions, far from being useful for what they really have to do: capture water. The size and the distribution of the inlet holes are not enough to achieve a proper efficiency [1–5], and sometimes they promote clogging effects [6].

In this sense, laboratory experiments have been traditionally carried out to determine the efficiency of inlets (quotient between intercepted and street flow) under different hydraulic

---

1 Corresponding author: Marcos Sanz-Ramos (marcos.sanz-ramos@upc.edu)

(flow) and geometric (slope of the street) conditions [2–5]. In the abovementioned studies the authors proposed an empirical relationship to obtain the efficiency of the inlet in terms of hydraulics (e.g. flow and depth) and geometry (e.g. number of holes, gap area, etc.). However, the design a new inlet means a high number of flume experiments (number of tests: inlet type per number of slopes, discharges, etc.), which are economically very expensive.

Currently, CFD numerical modelling techniques allow to minimize the number of laboratory experiments, in general, and thus reduce costs, in particular. Regarding to the hydraulics, and in order to properly choose the model (1D, 2D or 3D), notice that the runoff follows 1D patterns which are on the streets according to its alienation and slope (X-direction). Near the inlet, the flow acquires a 2D component, the hydrodynamics being affected by the inlet (Y-direction must not be neglected). Finally, 3D phenomena over the inlet are produced (Z-direction should be important).

Some studies have been carried out using 3D models [7–9] focussing in reproducing the hydraulics. However, to obtain satisfactory results it is necessary to use a very detailed mesh (high density of elements), which implies longer calculation times (at least 2 or 3 days) [9].

With the aim to reduce the computational time, this document shows the development and test of different techniques on the basis of the 2D model Iber [10]. The model was enhanced to represent 3D-phenoma effects using the similarity between the hydrological infiltration models and the discharge captured by the inlet. The numerical results were compared with the experimental data published in [2,3] and with flow patterns recorded in the framework of the 2017 study [11].

2. PHYSICAL MODEL

The experiment set up consisted in a 1:1-scale platform of 5.5 m-length and 3 m-width that represents a street (Figure 1). The platform can change its longitudinal and transverse slope from 0 to 10 % and 0 to 4 % respectively. For the transverse inlet experiments, the flume width was reduced to 1.5 m. The inlet used was “Barcelona1”, one of the most common inlets in Barcelona, and it was located at the end part of the flume.

Figure 1: Physical model. Flume facilities (left) and inlet “Barcelona1” (right). Source: [11].

The intercepted flow (Q_{in}) was measured for different geometric and discharge configurations (25, 50, 100 and 200 l/s), resulting 27 combinations. A detailed description of the flume configuration can be found in [2,3].

3. NUMERICAL MODEL

3.1 2D model Iber

Iber is a two-dimensional hydraulic model based on the Finite Volume Method [12,13]. It solves the 2D Saint Venant equations (2D-SWE) using 1st order Roe or 2nd order Minmod schemes [14]. Since 2011, Iber was enhanced to simulate hydrological process [15–17] including a source term in the mass conservation equation (1).

$$\frac{\delta h}{\delta t} + \frac{\delta q_x}{\delta x} + \frac{\delta q_y}{\delta y} = R - f \quad (1)$$

were h is the water depth, q_x and q_y are the X- and Y-direction fluxes and R and f represent the source term (R: rainfall; f: losses).

This is a common solution to include hydrological process into 2D-SWE models, because the gain or loss of Momentum affected by 3D-processes infiltration is negligible. However, when the work-scale is reduced, as in the analysis of the inlets, the 3D phenomena affects the momentum equation and must be considered.

3.2 Numerical solver

The solver proposed is based on the similarity between the captured flow by the inlets and the infiltration process. In both cases, the water intercepted can be interpreted as a subtraction from the 2D model.

Figure 2 shows how the infiltration process is performed in a 2D hydraulic model. Taking into account that the holes of the inlet are areas of infiltration (element), if an initial volume (h_1) exists over the element, at the next time step a volume is infiltrated (h_3) remaining the final volume (h_2). This mass subtraction implies, under this scheme, a flow acceleration because the 2D-SWE are not in equilibrium. Thus, a proper scheme must be necessary.

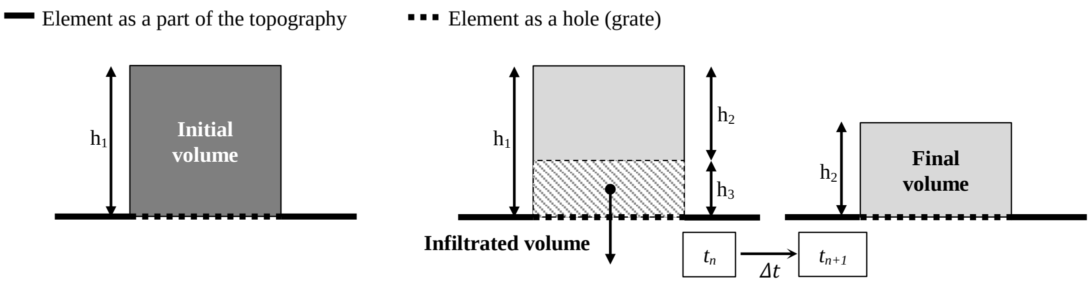

Figure 2: Description of the infiltration process in a hydraulic model.

Therefore, three different strategies were considered on the bases of Iber: NS-0 (official version); NS-1 (modification of Momentum); NS-2 (simplified modification of Momentum). In the first, no-modification was done, so the official version of Iber was used. The second one derives from an analysis of the Momentum equations, resulting that the fluxes are proportional to ($q[t_{n+1}] \propto q[t_n]$) the root-squared quotient of the final (h_2) and the initial volume (h_1). Third one was a simplification of the previous, where the relation between the fluxes is directly proportional to the quotient between the final (h_2) and the initial volume (h_1).

3.3 Model set-up

The flume was discretized by means of a mixt mesh (triangular and rectangular elements), coarse (0.2 m) on the exterior part and becoming finer (0.005 m) when approaching the inlet (Figure 3). The holes of the inlet (plotted in black, Figure 3) were also meshed (0.01 m). The same configuration was used for the transversal inlet.

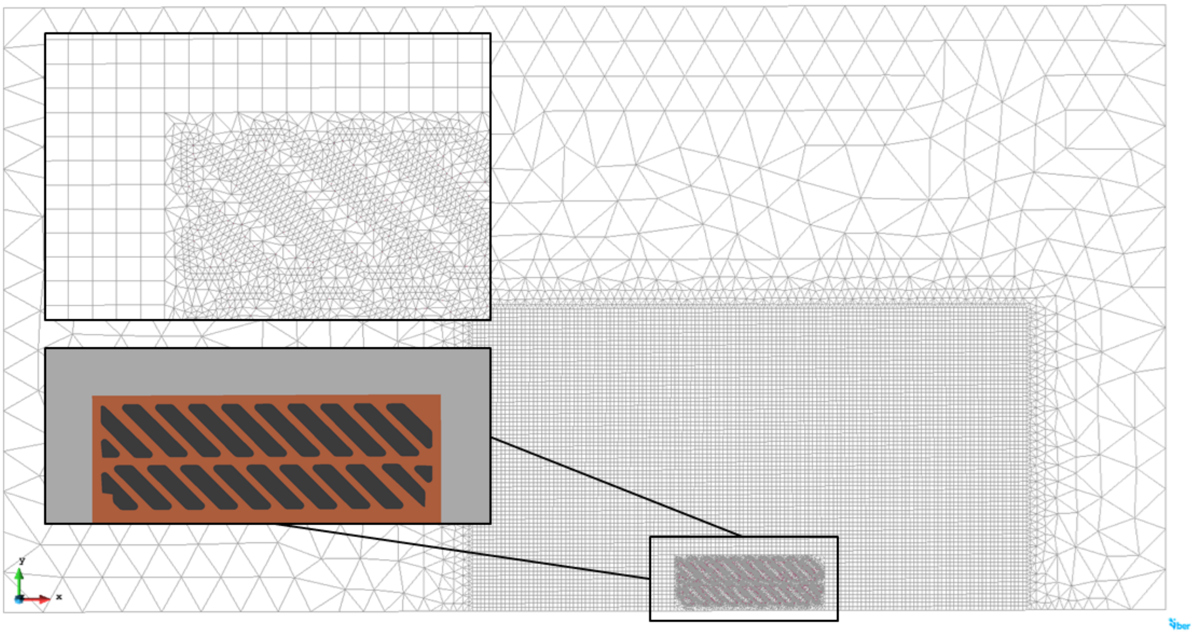

Figure 3: Flume discretization and mesh accuracy for gutter inlet. The different parts of the model were coloured (grey: flume platform; orange: inlet grate; black: inlet holes).

The boundary conditions were a constant discharge (25, 100 and 200 l/s for gutter inlet; 25, 50 and 100 l/s for transversal inlet) upstream and weir condition downstream. Constant infiltration was imposed on the inlet holes calculated as abovementioned. The roughness coefficient was 0.015 and 0.020 $s \cdot m^{-1/3}$ for the platform and the grate respectively. Moreover, k-ε turbulent model [18] was used.

The simulation time was 100 s, ensuring permanent flow, and the results were analysed at the end of the simulation. Wet-dry threshold of 0.0001 m and 2nd order scheme was chosen. A total of 288 simulations were carried out (216 for the lateral inlet and 72 for the transversal inlet), combination of the different flows, slopes and grates type tested.

4. RESULTS

4.1 Intercepted discharge

For all three numerical strategies performed the computed discharges intercepted by the inlet ($Q_{in,Iber}$) were checked with the experimental data (Q_{in}). No remarkable differences were presented in exception of 25 l/s gutter inlet (longitudinal and transversal slopes of 0 and 1 %

respectively), where the relative error was systematically over 10 % for the three numerical strategies (NS-0, NS-1 and NS-2). In general the mean relative error was under 1.25 % for gutter inlet and less than 0.2 % for transversal inlet.

4.2 Hydrodynamic patterns

A hydrodynamic patterns comparison was done by means of the representation of hydraulic jumps, recirculation areas and cross waves. Note that particular 3D phenomena as splash and vertical recirculation cannot be simulated by 2D-SWE model.

In general, as mentioned above, excessive flow acceleration was observed for NS-0 due to no corrections on the momentum equation was done. Thus, this numerical strategy overestimates the velocity field over the inlet, affecting the rest of the flow upstream and downstream. The hydraulic patterns observed do not represent the flume observations (Figure 4).

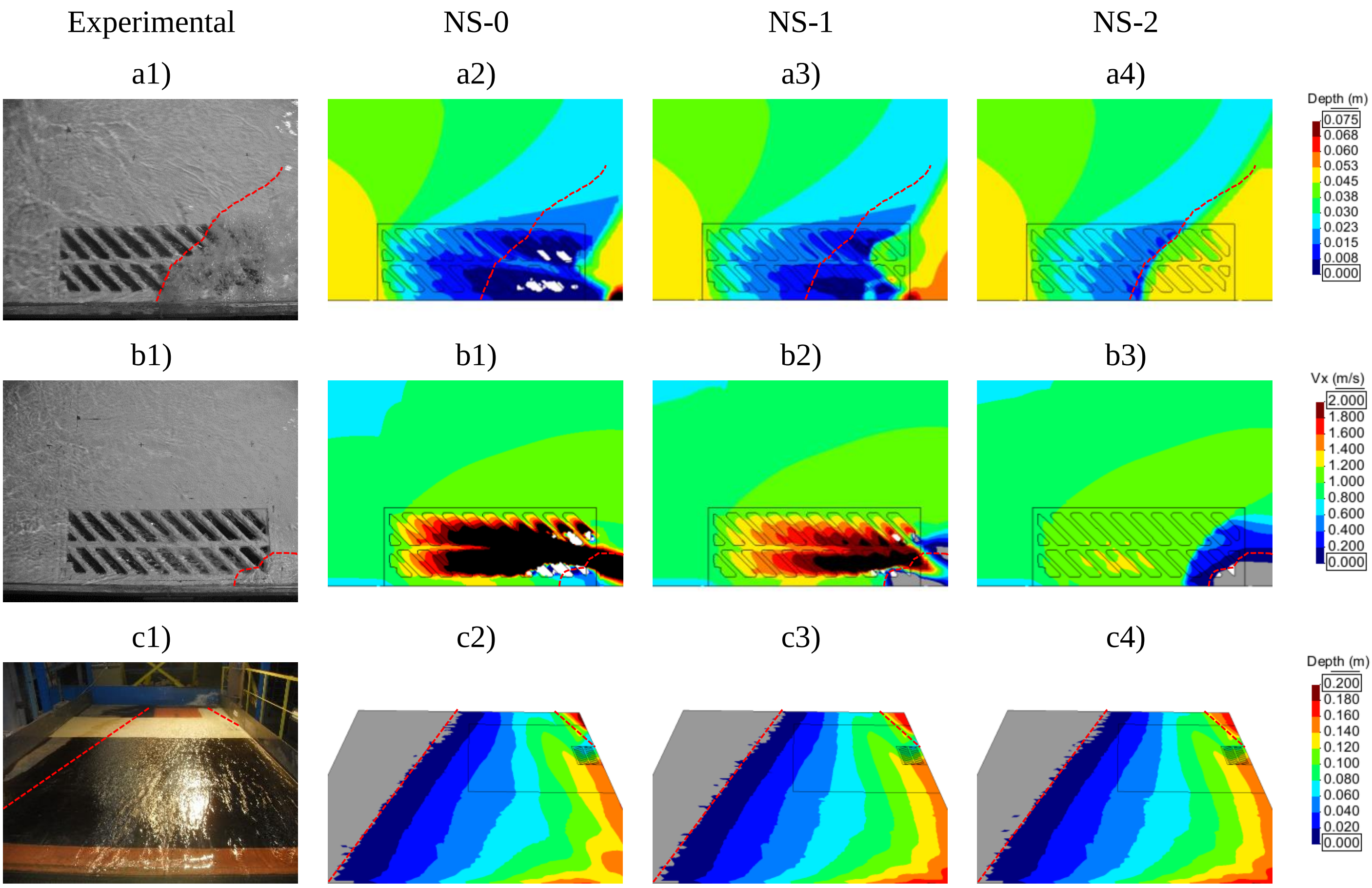

Figure 4: Flow patterns: hydraulic jump (a); recirculation area (b); cross waves (c). Flume configuration [discharge and longitudinal-transverse slopes]: 200 l/s and 4 %-0 % (a); 100 l/s and 2 %-0.5 % (b); 100 l/s and 4 %-10 % (c). Flow patterns are highlighted in red-dashed line.

NS-1 strategy was closer to the real behaviour but small flow acceleration was produced, thus the recirculation areas and the hydraulic jumps were in some cases unstable. Instead, NS-2 reached a suitable fit regarding all flow patterns types. The velocity reduction achieved in NS-1, and especially in NS-2, improve considerably the hydrodynamics behaviour of the whole system.

Experimental data did not show specific behaviour (patterns) in transversal inlets. In any case, the velocity reduction and depth increment from NS-0 to NS-2 adjusts reasonably well to the observations. Depending on the slope and the discharge, splash phenomenon occurs in both kind of inlet, but 2D-SWE models cannot reproduce this kind of pattern.

4.3 Computational time

The simulations were performed in an eight-threads Intel® Core™ i7 CPU with 3.5 GHz, launching 8 simulations at the same time (1 core per model). The simulation time oscillates between 0.75 and 7.5 hours. For gutter inlets NS-1 simulations 1.43-times faster than NS-0 and NS-2 was 2.11-times faster than NS-0 (average values). For transversal inlets NS-1 simulations 1.45-times faster than NS-0 and NS-2 was 1.85-times faster than NS-0 (average values). Comparing with 3D simulations performed by [9], the averaged speed-up achieved was higher than 15-times.

5. CONCLUSIONS

This work presents three different strategies to simulate sewer inlets using 2D-SWE Iber model, focusing in properly reproducing the observed hydraulic patterns in the laboratory (hydraulic jumps, recirculation areas, cross waves, etc.) and to reduce the computational time in comparison with 3D models.

The numerical strategies presented explores the capabilities of the hydrological approach based on 2D-SWE hydraulic models, using the similarity between the infiltration models and the discharge captured by the inlet: NS-0 (no-modification); NS-1 (modification of Momentum); NS-2 (simplified modification of Momentum).

The results NS-0 strategy did not adequately reproduce the hydrodynamics over the inlet, since an unreal acceleration of the flow is generated above the grid. The momentum reduction in the NS-1 model, although significant, was not enough to achieve a good fitting with the experimental data. Finally, the NS-2 strategy presented good results, both qualitative (hydraulic jumps, recirculation, wave fronts, etc.) and quantitative.

REFERENCES

[1] F. Spaliviero, R.W.P. May, M. Escarameia, Spacing of road gullies. Hydraulic Performance of BS En 124 gully gratings and kerb inlets, Report SR 533. HR Wallingford, UK, 2000.

[2] M. Gómez, B. Russo, Hydraulic Efficiency of Continuous Transverse Grates for Paved Areas, J. Irrig. Drain. Eng. 135 (2009) 225–230. doi:10.1061/(ASCE)0733-9437(2009)135:2(225).

[3] M. Gómez, B. Russo, M. Gómez, B. Russo, Methodology to estimate hydraulic efficiency of drain inlets, Proc. Inst. Civ. Eng. - Water Manag. 164 (2011) 81–90. doi:10.1680/wama.900070.

[4] B. Russo, M. Gómez, J. Tellez, Methodology to Estimate the Hydraulic Efficiency of Nontested Continuous Transverse Grates, J. Irrig. Drain. Eng. 139 (2013) 864–871. doi:10.1061/(ASCE)IR.1943-4774.0000625.

[5] B. Russo, M. Gómez-Valentín, Diseño de sistemas de sumideros en medio urbano, Primera ed, Barcelona : Mcharly, 2014 (España), 2014. www.mcharly.com.

[6] M. Gómez, G.H. Rabasseda, B. Russo, Experimental campaign to determine grated inlet clogging factors in an urban catchment of Barcelona, Urban Water J. 10 (2013) 50–61. doi:10.1080/1573062X.2012.690435.

[7] S. Djordjević, A.J. Saul, G.R. Tabor, J. Blanksby, I. Galambos, N. Sabtu, G. Sailor, Experimental and numerical investigation of interactions between above and below ground drainage systems, Water Sci. Technol. 67 (2013) 535–542. doi:10.2166/wst.2012.570.

[8] P. Lopes, J. Leandro, R.F. Carvalho, B. Russo, M. Gómez, Assessment of a VOF Model Ability to Reproduce the Efficiency of a Continuous Transverse Gully with Grate, J. Irrig. Drain. Eng. (2016) (in production). doi:10.1061/(ASCE)IR.1943-4774.0001058.

[9] M. Gómez, J. Recasens, B. Russo, E. Martínez-Gomariz, Assessment of inlet efficiency through a 3D simulation: numerical and experimental comparison, Water Sci. Technol. 74 (2016) 1926–1935. doi:10.2166/wst.2016.326.

[10] E. Bladé, L. Cea, G. Corestein, E. Escolano, J. Puertas, E. Vázquez-Cendón, J. Dolz, A. Coll, Iber: herramienta de simulación numérica del flujo en ríos, Rev. Int. Métodos Numéricos Para Cálculo y Diseño En Ing. 30 (2014) 1–10. doi:10.1016/j.rimni.2012.07.004.

[11] J. Tellez, M. Gómez, B. Russo, F. Zanon, Comparación de métodos de análisis de imágenes para determinar campo de velocidades aplicado a las rejas de alcantarillado, in: V Jornadas Ing. Del Agua, Coruña (Spain), 2017: pp. 1–12.

[12] L. Cea, M.E. Vázquez-Cendón, Unstructured finite volume discretization of two-dimensional depth-averaged shallow water equations with porosity, Int. J. Numer. Methods Fluids. 63 (2009) 903–930. doi:10.1002/fld.2107.

[13] R.L. LeVeque, Finite Volume Methods for Hyperbolic Problems, Cambridge Texts Appl. Math. 31 (2002).

[14] E.F. Toro, Riemann Solvers and Numerical Methods for Fluid Dynamics, Springer, Berlin (Heidelberg), 2009. doi:10.1007/b79761.

[15] L. Cea, E. Bladé, A simple and efficient unstructured finite volume scheme for solving the shallow water equations in overland flow applications, Water Resour. Res. 51 (2015) 5464–5486. doi:10.1002/2014WR016259.

[16] C.A. Caro, Modelación hidrológica distribuida basada en esquemas de volúmenes finitos, PhD Thesis. School of Civil Engineering, UPC. Barcelona, Spain, 2016. http://hdl.handle.net/10803/398144.

[17] C.A. Caro, E. Bladé, P. Acosta, Modelo de transferencia vertical dentro de un modelo hidrológico continuo en un esquema de volúmenes finitos, L'esprit Ingenieux. 3 (2012) 144–154.

[18] A.K. Rastogi, W. Rodi, Predictions of heat and mass transfer in open channels, J. Hydraul. Div. 104 (1978) 397–420.