Assessment of a VOF Model Ability to Reproduce the Efficiency of a Continuous Transverse Gully with Grate

Pedro LOPES, Ph.DCandidate (Corresponding author)

*Department of Civil Engineering & MARE - Marine and Environmental Sciences Centre,
University of Coimbra, Portugal.pmlopes@student.dec.uc.pt*

Jorge LEANDRO, Ph.D

*Department of Civil Engineering & MARE - Marine and Environmental Sciences Centre,
University of Coimbra, Portugal.leandro@dec.uc.pt*

Rita F. CARVALHO, Ph.D

*Department of Civil Engineering & MARE - Marine and Environmental Sciences Centre,
University of Coimbra, Portugal.ritalmfc@dec.uc.pt*

Beniamino RUSSO, Ph.D

*Hydraulic and Environmental Engineering, Technical School of La Almunia, University of
Zaragoza, Spain.brusso@unizar.es*

Manuel GÓMEZ, Ph.D

*Flumen Research Institute, ETSECCPB - Technical University of Catalonia (UPC),
Barcelona, Spain. manuel.gomez@upc.edu*

ABSTRACT

This paper deals with the numerical investigation of the drainage efficiency of a continuous transverse gully with grate's slots aligned in the flow direction and compared with experimental data sets. The gully efficiency attained with a 3D numerical model is compared and validated with experimentally data. The numerical simulations are performed using the CFD solver *interFoam* of OpenFOAM[®]. The numerical mesh is generated with the open-source and freeware software Salome-Platform[®]. Different slopes, from 0 to 10 % and a wide range of drainage flows, from 6.67 to 66.67 L/s/m are numerically simulated. The linear relation between Froude number and efficiency of the gully is in agreement to the one experimentally obtained.

KEYWORDS: Numerical validation, VOF, transverse gully, drainage efficiency.

INTRODUCTION

The complete and safe drainage in a flooding situation, caused by extreme rainfall events is one of the most challenging concerns for hydraulic engineers in urban areas, especially since a large part of cities are covered with impermeable areas. To drain such flows, efficient Urban Drainage System (UDS) needs to be designed giving special attention to gullies and manholes. These elements, also known as “linking elements”, contribute to the drainage of the flow from surface to the underground pipe systems. Theoretical and numerical studies on linking elements efficiency are of imperative importance to create constitutive relations between the underground and surface systems, and with them, calibrate urban drainage models (Djordjević, *et al.*, 2005; Leandro, *et al.*, 2009, 2016).

In areas where it is possible to define one point where the major part of the flow drains to (e.g. in curves along streets, sidewalks or gardens), the gullies are typically efficient structures, capturing part of the flow from the surface to the sewer systems. Gullies’ inlet is normally rectangular and placed at the same level as the pavement, covered, in the most of the cases, by a grate which can have different sizes and shapes. In some countries, gullies are replaced by drop sewer manholes covered with a resistant grate, and placed on streets following the same rules as for the gullies.

The current design guidelines for United States of America gullies are based upon a report published by FHWA (Federal Highway Administration) with topics about efficiency and size requirements (FHWA, 1984, 2001). In Portugal, urban drainage elements design follows the by-law RGSPDADAR (1995) which contains general rules to implement gullies but nothing related to their efficiency. Occasionally, the hydraulic performance and efficiency of the gullies can be found in some catalogues of grates manufacturers (e.g. NFCO, 1998). Given this worldwide diversity of grate inlets and implementation, detailed studies are needed. Efficiency studies of several grate types and curb inlets have been conducted by WEF & ASCE

(1992), Spaliviero and May (1998), Comport and Thornton (2009), Russo and Gómez (2010) or Comport *et al.* (2012). The efficiency of Portuguese gullies was numerically studied by Carvalho *et al.* (2011) using a 2D VOF/FAVOR (Volume of Fluid / Fractional Area Volume Obstacle Representation) model. The latter study was further extended by Martins *et al.* (2014) with 3D numerical simulations using the OpenFOAM[®] and the *interFoam* solver. The Authors compared water depths through the gully and the discharge coefficients for a large set of flow rates. In case of drop manholes, the studies are mainly focused on their efficiency, performance, shape and energy dissipation (Carvalho, and Leandro, 2012; Rubinato, *et al.*, 2013; Granata, *et al.*, 2014).

When underground pipes get pressurized, the flow can emerge to the surface through the gullies and manholes forming a jet. Romagnoli *et al.* (2013) used an ADV (Acoustic Doppler Velocimeter) to measure the flow behaviour and the turbulent structure of the flow under reverse conditions in Portuguese gullies. Lopes, *et al.* (2015) and Leandro *et al.* (2014) furthered the study of Portuguese gullies and found numerical relations between the flow from the ground pipe system and the height of the jet, and characterized the 2D vertical velocity profile inside a gully using the *interFoam* solver within OpenFOAM[®]. Vertical frequency and preferential direction of the jet at the surface were also studied. An experimental and numerical study in a typical United Kingdom gully under surcharge was made by Djordjević *et al.* (2013) to replicate the interactions between the surface flow and the ground drainage systems.

In large paved areas, such as squares, airports or high sloping roads, the common gullies inlets are ineffective to capture all the amount of rainfall water. In such cases, the continuous longitudinal transverse gullies represent a widely accepted solution, since the main design concern is on the positioning of the grate in the direction perpendicular to the flow. Due to the lack of tests in transverse gullies and their efficiency in different conditions, Gómez and

Russo(2009)studiedexperimentally four types of grates, found typically in Spain and differing in the alignment and distribution of the slots,under different longitudinal slopes and five approaching flows. They formulatedfour linear relations, one for each grate type, which link thehydraulic efficiency to some particular flow conditions (Froude number and water depth) and the grate length. The study was further extended by Russo *et al.* (2013) with the formulation of empirical expressions to relate grate hydraulic performance to flow parameters and grate geometry.

The advances of the last decades in Computational Fluid Dynamics (CFD) allow the prediction of the flow in some components of the UDS and the evaluation of the different design factors for an efficient project and operation(Jarman, *et al.*, 2008; Tabor, 2010; Bennett, *et al.*, 2011; Djordjević, *et al.*, 2013). The freeware and open-source OpenFOAM® is one of these CFD packages.Special attention is given to the solver *interFoam* due to its capability to deal with free-surface flow with arbitrary configurations.

In this paper we aim to investigate the ability of the *interFoam*VOF model in reproducingdrainage efficiency of acontinuous transverse gully with a grate,under different flow rates and slopes. In particular we aim to assessthe numerical model's ability to predict the hydraulic efficiency of urban drainage elements.Section "Experimental Model" describes the experimental installation and procedure. The numerical mesh and thenumerical model sectionare presented in section "Numerical Model", along with the simulations performed, the different meshes tested,the boundary conditions of the numerical model and the numerical simplifications.Section "Results" compares and validates the grate efficiency obtained numerically with the experimental datasets, further discussed in Section "Discussion".Finally,Section "Conclusions" summarizes andconcludes the work.

EXPERIMENTAL MODEL

The experiments were performedin the laboratory of the UPC Hydraulic Department

using the same installation described in Gómez and Russo (2009). It consists of a 1:1 scale model of a 1.5 m wide and 5 m long rectangular surface and a transversal grate placed downstream (**Fig. 1a**). The platform is able to simulate lanes with transversal slope up to 4 % and longitudinal slope up to 14 % and a wide set of flow rates from 20 to 200 L/s. A motorized slide valve regulates the flow discharged to the model and an electromagnetic flow meter measures the flow rates with an accuracy of 1 L/s. Upstream of the platform, the flow passes through a tank to dissipate the energy, providing horizontal flow conditions to surface water. The flow intercepted by the gully is conveyed to a V-notch triangular weir and the flow measurement is carried out through a limnimeter with an accuracy of 0.1 mm. Flow depths on the platform are visually obtained by reading a thin graduated scale.

The transverse grate used in this study is a composition of three single grates type 2 (**Fig. 1b**). This work comprises forty combinations of eight different longitudinal slopes $i_x = 0, 0.5, 1, 2, 4, 6, 8, 10$ % and five unit inflows $q = 6.67, 16.67, 33.33, 50.00, 66.67$ L/s/m. The transversal slope is $i_y = 0$ %.

NUMERICAL MODEL

Mathematical formulation

The numerical simulations are performed using the solver *interFoam* within the freeware and open-source CFD OpenFOAM[®] v.2.1.0. The mass and momentum equations are solved for isothermal, incompressible and immiscible two phase flows, written in their conservative form:

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{g} + \mathbf{f} \quad (2)$$

where ρ and μ are the fluid density and viscosity, $\mathbf{g}$ the gravitational acceleration, $\mathbf{u}$ the fluid velocity vector, $\mathbf{f}$ the volumetric surface tension force, $\boldsymbol{\tau}$ the viscous stress tensor and

127 $\nabla \cdot \tau$ the viscous stress term, defined as follows:

$$\nabla \cdot \tau = \nabla \cdot (\mu [\nabla \mathbf{u} + (\nabla \mathbf{u})^T]) = \nabla \cdot (\mu \nabla \mathbf{u}) + (\nabla \mathbf{u}) \cdot \nabla \mu \quad (3)$$

128 The modified pressure p^* is adopted in the model by removing the hydrostatic
129 pressure from the total pressure p . This is advantageous for the specification of pressure at the
130 boundaries (Rusche, 2002).

$$\nabla p = \nabla p^* + \rho \mathbf{g} + \mathbf{g} \cdot \mathbf{x} \nabla \rho \quad (4)$$

131 The *interFoam* uses the VOF technique, first developed by Hirt and Nichols (1981), to
132 follow and capture the interface between two different fluids by using a transport/advection
133 equation. The transport/advection equation is given by **Eq. (5)**; the last term is the compressive
134 term included by Berberović *et al.* (2009) in order to mitigate the effects of numerical diffusion
135 and to keep the gas-liquid interface sharp rather than using interface reconstitution schemes.
136 This term uses the compressive velocity ($\mathbf{u}_c$).

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) + \nabla \cdot [\mathbf{u}_c \alpha (1 - \alpha)] = 0 \quad (5)$$

137 Each cell of the domain is attributed an α value, ranging from 0 to 1 depending on which
138 portion of cell is occupied by fluid 1. Giving to fluid 1 the characteristics of water and to fluid
139 2, the physical characteristics of air, it means that cells with α equal to 1 only have water;
140 cells with α equal 0 only have air and therefore, cells with intermediate values are interface
141 cells (or free-surface cells) (Ubbink, 1997). The physical properties of the mixture are defined
142 as a weighted average of the physical properties of the two fluids denoted by subscripts 1 and
143 2:

$$\rho = \alpha \rho_1 + (1 - \alpha) \rho_2 \quad (6)$$

$$\mu = \alpha \mu_1 + (1 - \alpha) \mu_2 \quad (7)$$

144 The volumetric surface force function is explicitly estimated by the Continuum
145 Surface Force (CSF) model developed by Brackbill *et al.* (1991) which is also function of the

146 volume fraction:

$$f = \alpha \mathbf{u} \cdot \mathbf{u} \quad (8)$$

147 After discretization in space and time domains, the Pressure Implicit with Splitting of
148 Operators (PISO) algorithm (Issa, 1985) is used for the pressure-velocity coupling.

149 Model description

150 **Fig. 2** shows the full domain experimentally studied by Gómez and Russo (2009) for
151 the continuous grate with slots parallel to the longitudinal flow direction. The different
152 inflows are included in the simulations by an input of uniform velocity and flow depths,
153 whereas the different slopes are carried out numerically changing the direction of the gravity
154 vector.

155 Earlier work on modelling gullies (Carvalho, *et al.*, 2012; Lopes, *et al.*, 2015) proved
156 that setting the *interFoam* to “laminar” characteristics can provide accurate results with fast
157 convergence. The “laminar” model solves the mass and momentum equations together with
158 the advection equation for α without any turbulence approach. To achieve a quick
159 convergence to the steady state flow, the domain is initially filled by water up to a height
160 corresponding to the expected flow depth. The steady state is achieved after 5 seconds of
161 simulation, time from which the volume of water in the domain becomes constant. The results
162 presented are a further average of 5 s to 6 s of simulation. The simulation time step starts at
163 5×10^{-5} s and it is progressively increased to values which do not allow the Courant number to
164 exceed the pre-fixed value of 0.5. Each simulation is carried out in parallel over 3 sub-
165 domains using two Quad-Core processors of the Centaurus Cluster, installed on the
166 Laboratory for Advanced Computing of the University of Coimbra. The three sub-domains are
167 vertically divided in the longitudinal direction of the mesh.

Numerical mesh

A 3D section of the grate is built using the open-source Salome v.6.4.0 software (Salome, 2011). This software offers several pre-and post-processing modules of which especially relevant are *Geometry* and *Mesh*. The *Geometry* module provides a rich set of functions to create, edit and import CAD models while *Mesh* module divides the solid designed in *Geometry* in finite volume elements using selectable algorithms (e.g. NETGEN, GHS3D). In this study two different meshes are used: Mesh 1) is built with the NETGEN tetrahedral algorithm with edges ranging from 2 to 3 mm, making a total of 42772 cells (**Fig. 3a**); and Mesh 2) refined both closer to the channel bottom and on the gully inlet (**Fig. 3b**). This refinement limits the edges to lengths lower than 1 mm, resulting in a total of 94 215 cells. It should be noted, that the simulation of the full domain (i.e. the whole grate) is challenging because of the mesh generation and the added computational cost due to the large number of cells required. Our earlier work on this study, showed that in order to assure a good representation of the flow field, the mesh near to the grate slots needs to be as small as 1 mm. This value is in-line with other researchers work on gullies (Djordjević, *et al.*, 2013) that also used such fine meshes to model urban drainage devices. Extrapolating this edge size to the full domain, the total number of cells can quickly rise to approximately 45 million of cells. In order to reduce the number of cells in the domain, this study simplifies the grate geometry to one grate slot with 27.72 mm width and assures its representativeness by recognizing the symmetry planes between slots and applying it in its boundary conditions (see also Boundary Conditions section).

Boundary conditions

Five types of boundary conditions (BC) are included in the simulations (**Fig. 4**). The “Inflow” BC allows to specify the flow entering to the channel. Due to the wide quantity of

simulations performed with different inflow conditions (flow depth and velocity), the inflow is defined according to flow field conditions using a dynamic BC – extensions `swak4Foam` (OpenFOAM Wiki, 2013). Two “Outflows” are defined, one on the channel platform and other on the bottom. The “Atmosphere” boundary condition allows the air to leave and to enter in the domain and is defined on the top of the domain. The pressure on both “Outflow” and “Atmosphere” boundaries is set to zero. The “Wall” condition, which represents the bottom of the channel and the grate walls, is set to slip boundary conditions. The lateral walls are “Symmetry Planes” to assume the continuity of the domain.

Numerical approaches

The complete grate, i.e. the 3 groups of grate type 2, exhibit non homogeneous characteristics along the y direction (**Fig. 2**). Some grate slots are completely blocked with concrete (marked in the **Fig. 2** with black colour) whilst others feature different lengths (marked in **Fig. 2** with grey colour). In a homogeneous grate, i.e. if all the grate slots had the same dimensions, the single grate slot could numerically replicate the efficiency of the complete grate. However, in a non-homogeneous grate, using the 150 mm length slot as representative of the real grate could lead to an overestimation of the grate efficiency, due to an increased inlet area and therefore intercepted flow. To overcome this issue, the total intercepted flow is calculated as a function of the uncovered area used for drainage by the use of correction coefficients implemented on the Numerical Intercepted Flow (NIF), based on area proportionality, resulting in a Corrected Intercepted Flow (CIF), **Eq. (9)**.

$$CIF = \beta \times NIF \quad (9)$$

where, β is a coefficient that takes into account the obstructed area, which can take either the value of coefficient C_1 , or C_2 or a multiplication of C_1 and C_2 , as shown in **Fig. 5**. The reasoning behind the C coefficients will now be explained. Q_i , Q_{if} and Q_{out} are the inflow, intercepted flow and outflow for just one slot, respectively and Q_I and Q_{IF} are the inflow and

intercepted flow for the group of five slots, respectively. Coefficient C_1 takes into account the blocked slots. When one slot is blocked, Q_{IF} is calculated as five times the C_1 , times the Q_{if} , where C_1 is given by the number of uncovered slots, divided by the total number of slots. In this case, the Q_{IF} (or CIF) becomes equal to four times the Q_{if} . Coefficient C_2 takes into account the lower drainage efficiency of the shorter slots (i.e. with 120 mm length). If two slots are 0.5 shorter in length, Q_{IF} (or CIF) is calculated as $5 \times C_2 \times Q_{if}$, where C_2 is equal to three slots with length L_s , plus two slots with $0.5 \times L_s$, divided by the simulated five slots with length L_s .

In this study, coefficients C_1 and C_2 are:

$$C_1 = \frac{\text{N. of unblocked slots in the complete grate}}{\text{N total of slots in 1.5m}} = \frac{50}{55.5} = 0.9009 \quad (10)$$

$$C_2 = \frac{\text{Length of slots w/120mm} + \text{Length of slots w/150mm}}{\text{Total length simulated numerically}} = \frac{6 \times 120 + 11 \times 150}{17 \times 150} = 0.9294 \quad (11)$$

C_1 is always applied to the numerical intercepted flow unless the experimental efficiency of the grate is 100 %. C_2 is applied if during the numerical simulations $L_w/L_s > 4/5$ (i.e. 120 mm /150 mm), where L_w is the slot length occupied with water (**Fig. 6**). Ideally both coefficients should be verified numerically by running the complete grate, however the computational time required to do so, and the existence of real data upon the coefficients could be validated deemed the test-and-try process as preferable solution. **Table 1** summarizes the distribution of the coefficients in simulations and **Fig. 6** presents the 3D average free-surface position for simulations with $i_x = 4\%$ and the corresponding coefficients to be employed in each case.

RESULTS

Influence of mesh refinement on inflows

The inflow values acquired with Mesh 1 and Mesh 2 are assessed against the inflows imposed on the model, i.e. experimental inflows. **Table 2** presents the relative deviations between the numerical and experimental inflows for the entire set of drainage flows and the highest slopes ($> 2\%$). The relative deviations on the inflows (RDI) are calculated through **Eq. (12)**.

$$RDI(\%) = \frac{\text{Experimental Inflow} - \text{Numerical Inflow}}{\text{Experimental Inflow}} \quad (12)$$

Grate efficiency (numerical vs. experimental)

Fig. 7 shows the contour graph of the relative deviations between the numerical and experimental efficiencies of the grate for the complete set of simulations using Mesh 2. The relative deviations on the efficiencies (RDE) are calculated in a similar way as RDI– **Eq. (12)**, but instead Experimental and Numerical Inflows, the variables are Experimental and Numerical Efficiency.

Fig. 8 presents the numerical efficiency plotted against the experimental values. Limits of $\pm 10\%$, -28% (maximum negative) and $+19\%$ (maximum positive) errors are added to the graph.

Gómez and Russo (2009) related the efficiency (E) to hydraulic and dimensional parameters such as the Froude number (F), the width of the grate in the flow direction (B_g) (in our case $B_g = 195$ mm) and the flow depth upstream of the grate (h). The procedure is shown on **Fig. 9**, where the efficiency is related to those parameters. The R^2 correlation coefficient of the more adequate linear trend line, adapted to the experimental results ($\text{Exp}R^2$) is presented as well on **Fig. 9**. The $\text{Num}R^2$ coefficient is the correlation between the experimental trend line

and the numerical data. The simulations with 6.67 L/s are discarded from the graphic due to the constant efficiency equal to 1. This disposal was also adopted in the study of Gómez and Russo (2009).

Table 3 presents four quantitative statistical coefficients used to verify the accuracy of the numerical model. The numerical efficiency is tested against the experimental for the flows from 16.67 to 66.67 L/s/m. The Index of agreement (d), developed by Willmott (1981), measures the degree of model prediction error and varies between 0 and 1. Value 1 indicates the perfect agreement between the observed and predicted values. The Nash-Sutcliffe Efficiency (NSE) coefficient (Nash, and Sutcliffe, 1970) ranges between $-\infty$ and 1 and reaches the optimal value when $NSE = 1$. Values between 0 and 1 are viewed as good levels of performance whereas values lower than 0 indicate unacceptable performance. The RMSD (Root-Mean-Square Deviation) presents good agreement when the residuals are closer to zero. The PBIAS (Percent Bias) indicates, in percentage, the average tendency of the simulated values to be larger or smaller than observed ones (Gupta, *et al.*, 1999). Positive values indicate overestimation, whereas negative values indicate underestimation. The optimal value is 0.

DISCUSSION

Influence of mesh refinement on inflows

By the analysis of **Table 2** it is possible to note that for Mesh 1, in the lowest flows ($q = 6.67$ L/s/m), RDI assumes values higher than 15 %. This high discrepancy can be attributed to the lack of cells in the bottom of the channel, and consequently the vertical velocity profiles either the turbulent boundary layer are not accurately modelled. For the highest flow ($q = 50.00$ and $q = 66.67$ L/s/m), the relative deviations are not too high, assuming values which do not exceed 7 %. Using a second mesh generation, refined near to the channel bottom with cells not larger than 1 mm (Mesh 2), the RDI are far better than in Mesh 1. The values are now lower than 2 % both for the highest and lower values of flow rates. This result shows

that the refinement near to the bottom of the channel influences highly the accuracy of the data included on the model and consequently the results obtained.

Grate efficiency (numerical vs. experimental)

Fig. 7 shows that the relative deviation errors (RDE) are much more sensible to the inflow than to the slope conditions, although the global RDE increases slightly with the increment of both variables. The maximum positive RDE values occur for the intermediate slope, $i_x = 4\%$ and for a relative small inflow, $q = 16.67\text{ L/s/m}$ (RDE = +19 %) whereas the maximum negative RDE occurs for the maximum slope, $i_x = 10\%$ and the maximum inflow, $q = 66.67\text{ L/s/m}$ (RDE = -28 %). It can be concluded that the *interFoam* model is more efficient in the medium range inflows (about $q = 33.33\text{ L/s/m}$) and medium-small slopes (which we define as $\leq 2\%$) than in the range of high and low flow rates and high slopes where the errors magnitude can be greater than $\pm 10\%$. Similar conclusion was obtained by (Martins, *et al.*, 2014) for the simulation of a gully box under drainage flow situations. For small flow rates, the *interFoam* model misrepresented the flow dropping from the surface into the gully, by producing a falling jet attached to the side wall, instead of producing a free-fall jet profile. It is likely that the same should occur in other drop structures as manholes, weirs or orifices.

Eq. (12) can help clarify the reason for RDE larger than 10%. According to **Eq. (12)**, a negative RDE value means that the numerical efficiency is higher than the experimental (efficiency is overpredicted) whereas positive RDE values represent the opposite (efficiency is underpredicted). For lower discharges the outflow jet remains attached to the upstream wall of the gully box, misrepresenting the jet profile and the existing void fraction which should be observed between the upstream wall and the outflow jet. This phenomenon decreases the velocity of the jet and the amount of flow intercepted by the gully, thus resulting in the underprediction of the gully efficiency and the positive values of RDE. For highest discharges the jet is detached from the wall and the numerical simulation improves; nonetheless, in this

case a 3D vortex appears between the wall and the outflow jet which increases both jet velocity and the amount of flow intercepted and overpredicts the efficiency of the gully (negative RDE). Nonetheless it should be emphasized that these discrepancies are mostly found in the regions of extreme values (both inflows and slopes). For the intermediate inflow ranges and medium-small slopes ($\leq 2\%$), which represent most of the drainage inlet in real urban systems, show a good numerical accuracy and the *interFoam* gully model can be employed for assessing its efficiency.

Fig. 8 displays experimental efficiencies against the numerical efficiencies. The perfect agreement between those values is represented by the continuous line and relative deviations are represented with dashed lines. Although the limits of good accuracy for the numerical model are not generally defined, because it depends on the detail we want to resolve, in CFD simulations of UDS and for most engineering purposes it is usual to consider a good numerical performance for simulations where the relative deviations from the real data fall below $\pm 10\%$ (e.g. (Begum, *et al.*, 2011)). As such **Fig. 8** shows the $\pm 10\%$ limits and the maximum and minimum error lines. 60 % of the simulations fall within the limits $\pm 10\%$. 30% are found in the zone limited by the -10 % and -28 % lines, and 10% are found in the zone limited with the lines RDE +10 % and +19 %. Therefore it is suggested that the *interFoam* model must be carefully applied for continuous grate whenever the efficiency falls below 0.68 (limit marked with dot-slash vertical line).

Fig. 9 shows the relations between the Froude number, depth and grate length with the efficiency both for experimental and numerical data. The numerical correlation coefficient ($\text{Num}R^2$) is very similar to the one obtained with the experimental data, showing that the expression established by Gómez and Russo (2009) for this type of continuous grate with bars parallel to the flow direction is similar to the one obtained applying the *interFoam* model. The statistical coefficients shown on **Table 3** support the previous conclusion. The numerical

efficiency (predicted) obtained is similar to the experimental one (observed). For all the tests simulated, the obtained coefficients are similar to the optimal value of the test (Moriassi, *et al.*, 2007).

CONCLUSIONS

This article presents the assessment of a VOF model ability to reproduce the efficiency of a continuous transverse gully with grate with bars parallel to the flow direction. The validation of the VOF model was done by comparing with experimental real-scale data. The study focuses on its hydraulic efficiency which is the basis for the comparison and validation. In total, forty combinations of flow rates and slopes were tested. Two different meshes were generated with the Salome-Platform software to represent the geometry of the grate inlet with different refinements. The refinement of the mesh near to the channel bottom was shown to be preponderant to achieve the good accuracy of the numerical model, especially for shallow waters.

The relative deviations between the numerical and experimental data were calculated in relation to the flow rates and the various slopes. It was concluded that the numerical model is much more efficient in medium-high efficiencies range, which are mostly found in urban drainage systems. A linear relation was found between the flow Froude number and the efficiency of the grate. The R^2 values found were similar to the experimental relations achieved in Gómez and Russo (2009).

This study showed that the *interFoam* VOF solver can provide results similar to the ones obtained using experimental facilities, rendering the use of the numerical model a useful alternative to laboratory testing in the efficiency prediction of this and other types of gullies with grate.

356 **ACKNOWLEDGMENTS**

357 The authors would like to acknowledge the financing through FCT (“Fundação para a
358 Ciência e Tecnologia, IP” - Portuguese Foundation for Science and Technology) through
359 projects PTDC/AAC-AMB/101197/2008, PTDC/ECM/105446/2008 and
360 UID/MAR/04292/2013. Pedro Lopes was financed by FCT, through the PhD scholarship
361 Grant SFRH/BD/85783/2012, sub-financed by MEC (Portuguese Ministry of Education and
362 Science) and FSE (European Social Fund), under the programs POPH/QREN (Human
363 Potential Operational Programme from National Strategic Reference Framework) and POCH
364 (Human Capital Operational Programme) from Portugal 2020. The computational time spent
365 on the Centaurus Cluster of the Laboratory for Advanced Computing at University of
366 Coimbra is also acknowledged.

367 **NOTATION**

368 *The following symbols are used in this paper:*

369 B = Width of the channel (mm);
370 B_g = Width of the grate in the flow direction (mm);
371 d = Index of agreement (-);
372 E = Efficiency (%);
373 F = Froude number (-);
374 $\mathbf{f}$ = Volumetric surface tension force ($\text{kg/m}^2/\text{s}^2$);
375 g = Gravitational acceleration (m/s^2);
376 h = Flow depth (mm);
377 i_x = Longitudinal slope of the channel (%);
378 L = Length of the channel (mm);
379 L_g = Length of the grate (mm);
380 L_s = Length of the grate slot (mm);
381 L_w = Length of the grate occupied with water (mm);
382 Q_i, Q_{if}, Q_{out} = Inflow, Intercepted Flow and Outflow for one slot (l/s);
383 Q_I, Q_{IF}, Q_{Out} = Inflow, Intercepted Flow and Outflow for a group of slots (l/s);
384 q = Unit discharges (l/s/m);
385 $\mathbf{u}$ = Vector of mean velocity (m/s);
386 u_c = Compressive velocity at air-water interface (m/s).
387 α = Ratio between water and air in each cell of computational domain (-);
388 β, C_1, C_2 = Correction coefficients (-);

389 *The following abbreviations are used in this paper:*

390 2D, 3D = Two-, Three-Dimensional;
391 ADV = Acoustic Doppler Velocimeter;
392 BC = Boundary Conditions;
393 CFD = Computational Fluid Dynamics;
394 FAVOR = Fractional Area Volume Obstacle Representation;
395 FHWA = Federal Highway Administration;
396 G1,G2,G3 = Real grate geometry, intermediate geometry, rectangular geometry;
397 NIF = Numerical Intercepted Flow;
398 NSE = Nash-Sutcliffe Efficiency;
399 PBIAS = Percent Bias;
400 RDE = Relative Deviations on the Efficiencies;
401 RDI = Relative Deviations on the Inflows;
402 RMSD = Root Mean Square Deviation;
403 UDS = Urban Drainage System;
404 VOF = Volume of Fluid.

405

406 **BIBLIOGRAPHY**

- 407 Begum, S., Rasul, M.G., Brown, R.J., Subaschandar, N., and Thomas, P., 2011. "An
408 Experimental and Computational Investigation of Performance of Green Gully for
409 Reusing Stormwater". *Journal of Water Resuse and Desalination*, 1 (2), pp. 99–112.
- 410 Bennett, P.R., Stovin, V.R., and Guymer, I., 2011. "Improved CFD Simulation Approaches
411 for Manhole Mixing Investigations". In: *12th International Conference on Urban
412 Drainage (ICUD12)*. September 11-16, 2011, Porto Alegre/RS, Brazil. Published by
413 IWA Publishing, London, UK, pp. 1–8.
- 414 Berberović, E., van Hinsberg, N.P., Jakirlić, S., Roisman, I., and Tropea, C., 2009. "Drop
415 impact onto a liquid layer of finite thickness: Dynamics of the cavity evolution".
416 *Physical Review E*, 79 (3) DOI: 10.1103/PhysRevE.79.036306, pp. 1–15.
- 417 Brackbill, J.U., Kothe, D.B., and Zemach, C., 1991. "A continuum method for modeling
418 surface tension". *Journal of Computational Physics*, 100, pp. 335–354.
- 419 Carvalho, R.F. and Leandro, J., 2012. "Hydraulic Characteristics of a Drop Square Manhole
420 with a Downstream Control Gate". *Journal of Irrigation and Drainage Engineering*, 138
421 (6) DOI: 10.1061/(ASCE)IR.1943-4774.0000437, pp. 569–576.
- 422 Carvalho, R.F., Leandro, J., David, L.M., Martins, R., and Melo, N., 2011. "Numerical
423 Research of the Inflow Into Different Gullies Outlets". In: *Computing and Control for
424 the Water Industry*. Exeter, UK.
- 425 Carvalho, R.F., Leandro, J., Martins, R., and Lopes, P., 2012. "Numerical study of the flow
426 behaviour in a gully". In: *4th IAHR International Symposium on Hydraulic Structures*.
427 February 9-11, 2012, Porto, Portugal. Published by APRH Publishing, Lisbon, Portugal.
- 428 Comport, B.C., Cox, A.L., and Thornton, C.I., 2012. "Performance Assessment of Grate
429 Inlets For Highway Median Drainage". Denver[Report].
- 430 Comport, B.C. and Thornton, C.I., 2009. "Hydraulic Efficiency of Grate and Curb Inlets for

Urban Storm Drainage".

Djordjević, S., Prodanović, D., Maksimović, C., Ivetić, M., and Savić, D.A., 2005. "SIPSON-simulation of interaction between pipe flow and surface overland flow in networks.". *Water Science and Technology*, 52 (5), pp. 275–283.

Djordjević, S., Saul, A.J., Tabor, G.R., Blanksby, J., Galambos, I., Sabtu, N., and Sailor, G., 2013. "Experimental and numerical investigation of interactions between above and below ground drainage systems.". *Water Science and Technology*, 67 (3) DOI: 10.2166/wst.2012.570, pp. 535–42.

FHWA, 1984. "Drainage of Highway Pavements - HEC12".

FHWA, 2001. "Urban Drainage Design Manual - HEC22".

Gómez, M. and Russo, B., 2009. "Hydraulic Efficiency of Continuous Transverse Grates for Paved Areas". *Journal of Irrigation and Drainage Engineering*, 135 (2) DOI: 10.1061/(ASCE)0733-9437(2009)135:2(225), pp. 225–230.

Granata, F., de Marinis, G., and Gargano, R., 2014. "Flow-improving elements in circular drop manholes". *Journal of Hydraulic Research*, (June 2014) DOI: 10.1080/00221686.2013.879745, pp. 1–9.

Gupta, H., Sorooshian, S., and Yapo, P., 1999. "Status of Automatic Calibration for Hydrologic Models: Comparison with Multilevel Expert Calibration". *Journal of Hydrologic Engineering*, 4 (2) DOI: 10.1061/(ASCE)1084-0699(1999)4:2(135), pp. 135–143.

Hirt, C.W. and Nichols, B.D., 1981. "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries". *Journal of Computational Physics*, 39 DOI: 10.1016/0021-9991(81)90145-5, pp. 201–225.

Issa, R.I., 1985. "Solution of the implicitly discretised fluid flow equations by operator-splitting". *Journal of Computational Physics*, 62 (1) DOI: 10.1016/0021-9991(86)90099-9, pp. 40–65.

Jarman, D.S., Faram, M.G., Butler, D., Tabor, G.R., Stovin, V.R., Burt, D., and Throp, E., 2008. "Computational fluid dynamics as a tool for urban drainage system analysis : A review of applications and best practice". In: *11th International Conference on Urban Drainage*. Edinburgh, Scotland, UK, pp. 1–10.

Leandro, J., Chen, A., Djordjević, S., and Savić, D.A., 2009. "Comparison of 1D/1D and 1D/2D Coupled (Sewer/Surface) Hydraulic Models for Urban Flood Simulation". *Journal of Hydraulic Engineering*, 135 (6) DOI: 10.1061/(ASCE)HY.1943-7900.0000037, pp. 495–504.

Leandro, J., Lopes, P., Carvalho, R., Páscoa, P., Martins, R., and Romagnoli, M., 2014. "Numerical and experimental characterization of the 2D vertical average-velocity plane at the centre-profile and qualitative air entrainment inside a gully for drainage and reverse flow". *Computers & Fluids*, 102 DOI: 10.1016/j.compfluid.2014.05.032, pp. 52–61.

Leandro, J., Schumann, A., and Pfister, A., 2016. "A step towards considering the spatial heterogeneity of urban key features in urban hydrology flood modelling". *Journal of Hydrology*, DOI: 10.1016/j.jhydrol.2016.01.060.

Lopes, P., Leandro, J., Carvalho, R.F., Páscoa, P., and Martins, R., 2015. "Numerical and experimental investigation of a gully under surcharge conditions". *Urban Water Journal*, 12 (6) DOI: 10.1080/1573062X.2013.831916, pp. 468–476.

- Martins, R., Leandro, J., and Carvalho, R.F. de, 2014. "Characterization of the hydraulic performance of a gully under drainage conditions". *Water Science & Technology*, 69 (12) DOI: 10.2166/wst.2014.168, pp. 2423–2430.
- Moriasi, D.N., Arnold, J.G., Liew, M.W. Van, Bingner, R.L., Harmel, R.D., and Veith, T.L., 2007. "Model Evaluation Guidelines for Systematic Quantification of Accuracy in Watershed Simulations". *American Society of Agricultural and Biological Engineers*, 50 (3), pp. 885–900.
- Nash, J.E. and Sutcliffe, J. V., 1970. "River flow forecasting through conceptual models part I — A discussion of principles". *Journal of Hydrology*, 10 (3), pp. 282–290.
- NFCO, 1998. "Catalog". *Neenah Foundary Company*.
- OpenFOAMWiki, 2013. Contrib/swak4Foam - OpenFOAMWiki [online]. Available from: <http://openfoamwiki.net/index.php/Contrib/swak4Foam> [Accessed 16 Sep 2013].
- RGSPDADAR, 1995. "Regulamento Geral dos Sistemas Públicos e Prediais de Distribuição de Água e Drenagem de Águas Residuais". *Decreto Regulamentar 23/95 de 23 de Agosto - Diário da República - I Série-B*, pp. 5284–5319.
- Romagnoli, M., Carvalho, R.F., and Leandro, J., 2013. "Turbulence characterization in a gully with reverse flow". *Journal of Hydraulic Engineering-ASCE*, 139 (7) DOI: 10.1061/(ASCE)HY.1943-7900.0000737, pp. 736–744.
- Rubinato, M., Shucksmith, J., Saul, A.J., and Shepherd, W., 2013. "Comparison between InfoWorks hydraulic results and a physical model of an urban drainage system". *Water Science & Technology*, 68 (2) DOI: 10.2166/wst.2013.254, pp. 372–379.
- Rusche, H., 2002. "Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions". *PhD Thesis*. University of London.
- Russo, B. and Gómez, M., 2011. "Methodology to estimate hydraulic efficiency of drain inlets". *Proceedings of the ICE - Water Management*, 164 (WM2) DOI: 10.1680/wama.900070, pp. 81–90.
- Russo, B., Gómez, M., and Tellez, J., 2013. "Methodology to Estimate the Hydraulic Efficiency of Nontested Continuous Transverse Grates". *Journal of Irrigation and Drainage Engineering*, 139 (10) DOI: 10.1061/(ASCE)IR.1943-4774.0000625, pp. 864–871.
- Salome, 2011. SALOME 6 Platform Download [online]. Available from: <http://www.salome-platform.org/downloads/salome-v6.4.0>.
- Spaliviero, F. and May, R.W.P., 1998. "Spacing of road gullies. Hydraulic performance of BS EN 124 gully gratings".
- Tabor, G., 2010. "OpenFOAM: An Exeter Perspective". In: *V European Conference on Computational Fluid Dynamics - ECCOMAS CFD*. Lisbon, Portugal.
- Ubbink, O., 1997. "Numerical prediction of two fluid systems with sharp interfaces". *PhD Thesis*. Imperial College of Science, UK.
- WEF & ASCE, 1992. "Design and Construction of Urban Stormwater Management Systems".
- Willmott, C.J., 1981. "On the validation of models". *Phys. Geog.*, 2, pp. 184–194.

518

519

(a)

(b)

520 **Fig. 1.** Photographs of the experimental installation built at UPC Hydraulic Department: (a)
521 rectangular platform with transverse grate downstream; (b) single grate type 2.

522

Fig. 2. Sketch of the experimental installation: (a) top view, (b) detail of grate and (c) cut B-B'.

(a)

(b)

Fig. 3. Detail of the meshes used in this study: (a) Mesh 1 - homogeneous mesh with ranging spaces between 2 and 3 mm (b) Mesh2 – mesh refined at the channel bottom with cells 1 mm spaced.

534
535
536
537

Fig. 4. Sketch of the simplified geometry of the grate, flow comes from the inflow boundary to the outflow. All the measures are in [mm]. The red point on the beginning of the grate hole identifies the axis origin.

539

540 **Fig. 5.** Sketch of the process to calculate the coefficients C_1 and C_2 , used to calculate the
 541 corrected intercepted flow (CIF) using the numerical intercepted flow (NIF). Q_i , Q_{if} and Q_{out}
 542 are respectively the Inflow, Intercepted Flow and Outflow for one slot; Q_I , Q_{IF} and Q_{OUT} are
 543 respectively the Inflow, Intercepted Flow and Outflow for a group of slots.

544

Fig. 6. 3D image of free-surface average position over the slot and the correspondent choice of coefficients. For all the simulations, $i_x = 4 \%$. L_w is the slot length occupied with water, L_s is the length of the grate slot.

552

553 **Fig. 7.** Contour graph of RDE. q are the unit discharges and i_x the longitudinal slope of the
554 channel.

555

556

557

558 **Fig. 8.**Relative deviations between the numerical and the experimental efficiency.

559

560

561

562 **Fig. 9.** Graph relating the efficiency with Froude number (F), flow depth (h) and width of the
563 grate in the flow direction (B_g).

564

565

566
567 **Table 1.**Distribution of the applied correction coefficients (C_1) and (C_2), respectively
568 described by (10) and (11). q are the unit discharges in L/s/m and i_x the longitudinal slope of
569 the channel in %.

RelativeDeviation(%)		q (l/s/m)			
i_x (%)		6.67	16.67	33.33	50.00 66.67
10.00	*	C_1	C_1	$C_1 \times C_2$	$C_1 \times C_2$
8.00	*	C_1	C_1	$C_1 \times C_2$	$C_1 \times C_2$
6.00	*	C_1	C_1	$C_1 \times C_2$	$C_1 \times C_2$
4.00	*	C_1	C_1	$C_1 \times C_2$	$C_1 \times C_2$
2.00	*	C_1	C_1	$C_1 \times C_2$	$C_1 \times C_2$
1.00	*	C_1	C_1	$C_1 \times C_2$	$C_1 \times C_2$
0.50	*	*	C_1	C_1	$C_1 \times C_2$
0.00	*	*	C_1	C_1	$C_1 \times C_2$

* Simulations where the experimental efficiency is 1.

Table 2. Relative deviations on the inflows (RDI) for the two meshed proposed – Mesh 1 and Mesh 2. q are the unit discharges and i_x the longitudinal slope of the channel.

RDI (%)	MESH1 - q (l/s/m)					MESH2 - q (l/s/m)				
i_x (%)	6.67	16.67	33.33	50.00	66.67	6.67	16.67	33.33	50.00	66.67
10.00	22.22	37.78	16.67	3.70	1.67	0.00	0.00	1.11	0.00	0.56
8.00	38.89	13.33	7.78	6.67	0.00	0.00	0.00	0.00	0.74	0.00
6.00	16.67	11.11	7.78	0.74	2.78	0.00	0.00	1.11	0.00	0.56
4.00	22.22	6.67	8.89	2.96	0.56	0.00	2.22	0.00	0.00	1.67
2.00	22.22	13.33	5.56	1.48	0.00	0.00	0.00	0.00	0.74	2.22

Note: The background of the values are filled in grey scale, where the darkest square mark the deviations higher than 20%, for deviations between 10 and 20%, between 5 and 10% and deviations lower than 5%.

581
 582 **Table 3.**Quantitative statistical coefficients to investigate the model accuracy. The
 583 coefficients relate the gully efficiency (experimental vs. numerical) for the range of flows
 584 from 16.67 to 66.67 L/s/m. d – Index of agreement; NSE – Nash-Sutcliffe Efficiency; RMSD
 585 – Root Mean Square Deviation; PBIAS – Percent Bias.

Coeff.	Obtained value	Optimal value
d	0.887	1
NSE	0.712	1
RMSD	0.087	0
PBIAS	-0.909	0

586
 587
 588