Hydraulic behaviour of a gully under surcharge conditions

Pedro Lopes[1] , Jorge Leandro[2], Rita Fernandes de Carvalho[3] and Ricardo Martins[4]

[1] IMAR Researcher, Civil Engineering Department, University of Coimbra, Portugal, pmlopes@student.dec.uc.pt
[2] Assistant Professor, Civil Engineering Department, University of Coimbra, Portugal, leandro@dec.uc.pt
[3] Assistant Professor, Civil Engineering Department, University of Coimbra, Portugal, ritalmfc@dec.uc.pt
[4] PhD Student, Civil Engineering Department, University of Coimbra, Portugal, rdd45466@student.dec.uc.pt

ABSTRACT

This paper presents a study of the hydraulic behaviour of a gully under surcharge conditions using both numerical and experimental models. These results can be useful for the validation of the linking elements in Dual Drainage (DD) models, recently created. The final numerical results for the gully were obtained using the grid generator SALOME Platform and the Computational Fluid Dynamics (CFD) model OpenFOAM™. Experiments were carried out in a 8 m long and 0.5 m wide channel, fitted with a 600 x 300 x 300 [mm] gully and a gully outlet with a 80 mm diameter pipe that works as inlet in this study. The selected solver, mesh size and contraction at the bottom inlet allowed for an adequate modelling of the gully under surcharge conditions. The experimental and numerical results are in good agreement.

KEYWORDS

CFD, experimental data, gully, surcharge flow, urban drainage.

1 INTRODUCTION

Urban flooding is presently a priority for the EU (e.g. Directive 2007/60/CE). According to the IPCC (Intergovernmental Panel for Climate Changes), it is expected an increase of extreme events as a result of climate change, risking both populations and infrastructures.

The study of the hydraulic behaviour of some components of urban drainage systems is important in case of flooding, for predicting affected areas. The urban drainage systems do not operate under the conditions that they were originally conceived for. During a flood event gullies could work under drainage conditions while the sewer system does not reach its full capacity, however when the sewer systems reaches its full capacity it becomes pressurized, and the flow through the gullies may surcharge. These phenomena, in extreme cases may cause “urban geysers”, and quite possibly the violent projection of the grate. An example of this extreme events occurred in Calgary in Alberta, Canada on 7 March 1999 (YoutubeVideo, 2007).

Due to recent computer power developments, numerical models can now be used to reproduce the complexity of these flows (Carvalho et al., 2012). Although surcharge flow through gullies has three-dimensional behaviour, it can be translated into 1D or 2D Dual Drainage models (DD) through linking models (Djordjević et al., 2011) able to predict floods in large areas. However a careful validation is required. Due to its complexity, the validation is difficult and real data is inexistent and/or of poor quality. Therefore, a research group from IMAR-in the aim of the project "Multiple Linking Element" funded by FCT (English acronym: Foundation for Science and Technology) (PTDC/AAC-AMB/101197/2008) is focus to improve and validate such DD numerical models and the linking elements in a real scale experimental installation built for this purpose.

The CFD (Computational Fluid Dynamics) model OpenFOAM^{TM}, a free Open source Field Operation and Manipulation C++ libraries, developed by the "OpenFOAM Team" at "SGI Corp" and distributed by "OpenFOAM Foundation", is used to perform highly complex numerical simulations, such as surcharge flow in gullies. The aim of this paper is to study a specific gully under surcharge conditions and to compare both quantitatively and qualitatively the experimental results with CFD simulations. The study allows full characterization of the flow behaviour in a specific gully under surcharge conditions without the grade on top of the gully; this may happen during a severe flooding event, or simply as an operational procedure when the municipality fears an approaching heavy storm.

2 METHODOLOGY

2.1 Mesh Generation

Two kind of meshes was used: (1) the mesh regular and non-uniform with spaces ranging of 1 cm to 4 cm adapted from study under drainage conditions of flow into the gully (Martins, 2011) generated using *blockMesh* utility in OpenFOAM^{TM} – Mesh 1; (2) The mesh created using an open source integration platform for numerical simulations SALOME v.6.4.0 (Salome, 2011) – Mesh 2. The SALOME environment contains different and separate working spaces for a progressive creation of mesh, with a geometry creator (Geometry) and a mesh generator (Mesh). Using the Mesh working space is possible to create sub-meshes to improve the discretization, only in some parts of the structure.

The SALOME is able to create tetrahedral, hexahedral and prismatic cells. Several meshes were tested and finally an automatic tetrahedral mesh created using *Netgen* routine was choosen (Kortelainen, 2009). The cell maximum size was set to 0.02m for the channel and 0.015cm for the gully, making a total of 250 000 points. In this process, four types of boundary conditions were defined: inlet (pipe), outlet, atmosphere and walls. After, the mesh was exported from SALOME in I-deas UNV format was converted to a format readable by OpenFOAM^{TM} using the *ideasUnvToFoam* utility (Kulakov, 2010).

2.2 OpenFOAM^{TM} Simulations

Solutions are obtained using OpenFOAM^{TM} v1.7.1 (OpenFOAM, 2010) with the solver *interFoam* (Ubbink, 1997) and algorithm PISO (Pressure Implicit Splitting of Operators) (Issa, 1986) used for interactive computation of unsteady incompressible flows. The solver *interFoam* is based in VOF (Volume-Of-Fluid) method (Hirt & Nichols, 1981) where a transport equation is able to determine the relative volume fraction of the two phases (alpha1) in each computational cell (OpenFOAM, 2011a). The PISO show robust convergence behaviour and required less computational effort than SIMPLER and SIMPLEC (Versteeg & Malalasekera, 1995).

Three options of modelling are tested: *laminar* (OpenFOAM™ nomenclature), RAS *k-ε model* and LES *Smagorinsky model*. The *laminar* uses no turbulence models (OpenFOAM, 2011b). The RAS *k-ε model* (Reynolds-Average Simulation) (Launder & Spalding, 1974) uses two closure equations for k and ε. In this model the unsteadiness in flow is averaged out and regarded as part of the turbulence (Furbo et al., 2009). It is the most widely used and validated turbulence model due to the excellent performance and simplicity of boundary parameters used (Versteeg & Malalasekera, 1995) (detailed information can be consulted in Fluent manual (Fluent Inc., 2003)). The LES *Smagorinsky model* is a type of LES (Large-Eddy Simulation) model proposed by Smagorinsky (1963). This resolves large scales of the flow field solution with better fidelity than RAS but, on the other hand, leads to higher computational cost for most hydraulic engineering problems.

The boundary conditions are used with different parameters depending on their functional characteristics: The *inlet* only allows flow in at a fixed velocity; the *outlet* is a boundary where the fluids exits the domain, where the relative pressure is fixed to 0; in the *atmosphere* the air can make exchanges with the outside and the relative pressure is set to 0 and the *wall* have the condition of no slip ant therefore the velocity is set to 0. Where one parameter is stipulated the other boundary parameters are calculated by OpenFOAM™.

2.3 Validation with Experimental Setup

The experimental installation was constructed in the Department of Civil Engineering at the University of Coimbra with the objective to simulate accurately the flow thought the gully using standard components, see Figure 1a. The gully, with dimensions 600 x 300 x 300 [mm] has an inlet a pipe with 80 mm of internal diameter, stocked by a reservoir with hole in the base, providing a static pressure and a steady state check. The channel has 500 mm width and 1% slope (Figure 1b). This structure allows a good range of surcharge flow, in range 2 to 8 litre/second.

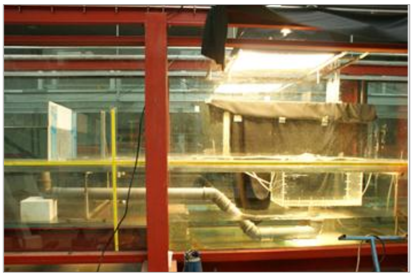

Figure 1a. Experimental setup photography.

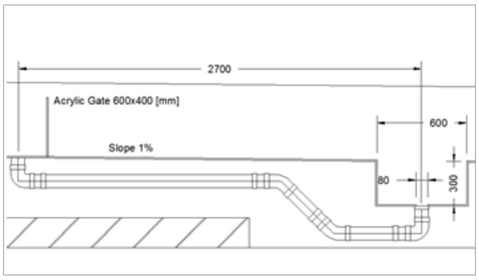

Figure 1b. Experimental setup dimensions (mm).

To capture free-surface a computational vision model in Simulink® was used (Roque, 2011). This model is able to calculate the free-surface directly from a video.

3 RESULTS AND DISCUSSION

3.1 Gully with Straight Inlet (GSI)

Figure 2 illustrate the Mesh 1 (see chapter 2.1), used for representing simple straight inlet.

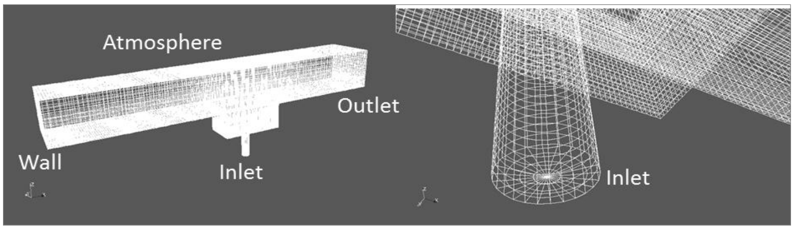

Figure 2. GSI mesh.

Different turbulence models were tested using the same flow conditions in the simulation with Q=6 l/s and inlet velocity U=1.2 m/s. For RAS *k-ε model*, the parameters k=0.0030 m^2/s^2 and ε=0.0048 m^2/s^3 were considered (Versteeg & Malalasekera, 1995; Fluent Inc., 2003). For LES *Smagorinsky model*, the Smagorinsky coefficients are Cε=0.93 and Ck=0.094, using Moeng & Wyngaard (1988) formulas. Identical values are proposed by Lilly (1966) and Deardorff (1970).

Figure 3 shows the average contour for 15 to 20 seconds (time interval which ensures a steady flow) in a middle transversal section of gully for laminar (Q6laminar), RAS (Q6RAS) and LES (Q6LES) simulations (OpenFOAM, 2011a). These numerical results are shown with a random photograph in the background.

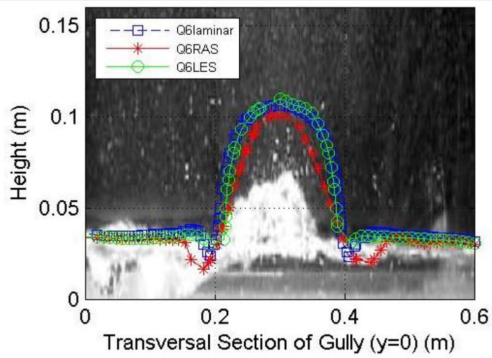

Figure 3. Contour average in a middle section of gully for GSI mesh.

A good agreement was observed between simulations Q6laminar and Q6LES in comparison to Q6RAS, nonetheless all are qualitatively similar. The LES model is considered the most compressive turbulence model, nonetheless the simpler *laminar* model with a quicker convergence seemed to represent well the main features of the flow found in surcharge flow. However, all the computational simulations are distant of the experimental results in predicting free surface position above the gully bottom inlet.

3.2 Gully with Inlet Curve and Energy Losses (GICEL)

To improve the numerical results, especially in jet definition, a new geometry was created using the Salome-platform - Mesh 2 (see chapter 2.1), where the energy losses in the experimental circuit and the influence of the curve in stream lines are taken into account. The energies losses are achieved by the inclusion of a sudden enlargement in the pipe (Figure 4 at right). This configuration is based on the coefficients of charge loss given by (Lencastre, 1987) and implemented on Salome-platform. The mesh is calculated using a "Tetrahedralization (Netgen)" algorithm with the following parameters:

maximum 1D element size for free-surface platform in range 0.01 to 0.015m, 1D element size in range 0.01m to 0.012m for the gully box and pipe (Figure 4).

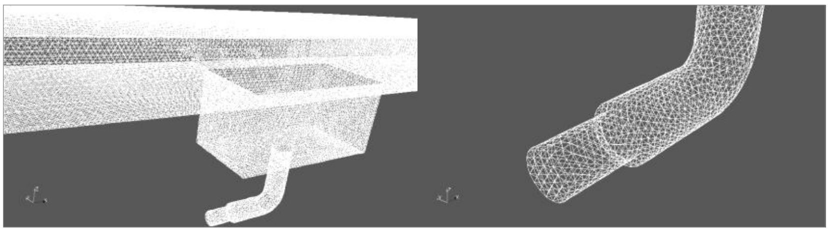

Figure 4. Details of computational mesh for GICEL: at right, the sudden enlargement proposed.

Three simulations were performed, relatively to inlet flow with 2, 4 and 6 litres per second and the results were obtained with the average of last 5 seconds of simulation, between 15 and 20 seconds, in a middle transversal section of gully (y=0). To validate these results, we compared the surface obtained numerically with the experimental results using computational vision in Simulink® model (Roque, 2011) and photographs in background (Figure 5). A good agreement was found in the results comparatively with results obtained with GSI mesh, fundamentally in jet definition since the free surface has also good results. Some improvement in Q6 simulation must be considered in future works.

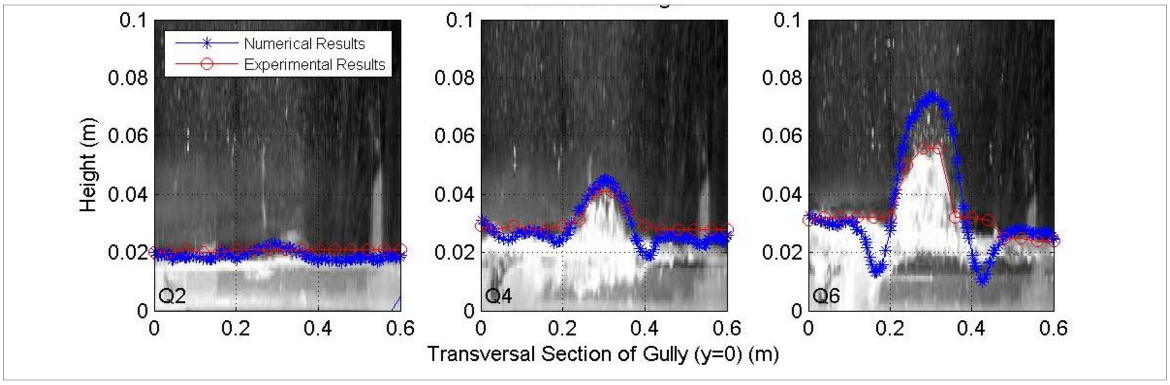

Figure 5. Contour average for Q2, Q4 and Q6 simulation.

Figure 6 show the adjustment of results to a normal distribution. The lines presented are the limits of 95% confidence interval for the average (middle line on graphics and marked with circles). In all flows, the average is calculated with great confidence.

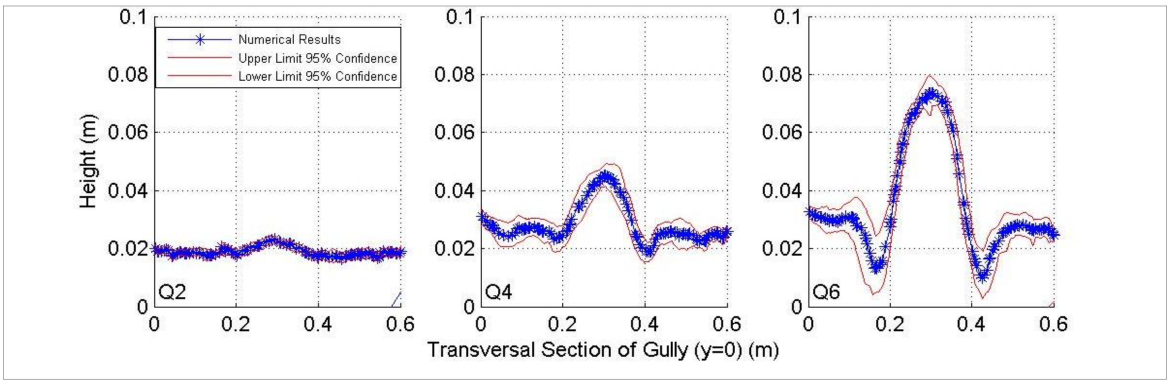

Figure 6. Limits of 95% confidence interval for the average.

Figure 7 shows in background the velocity fields and the pressure at left and right wall of the gully for the three flows simulated. The pressure graphic is similar to a hydrostatic pressure profile in both sides. This result shows that the pressure exerted by the vortices is almost null.

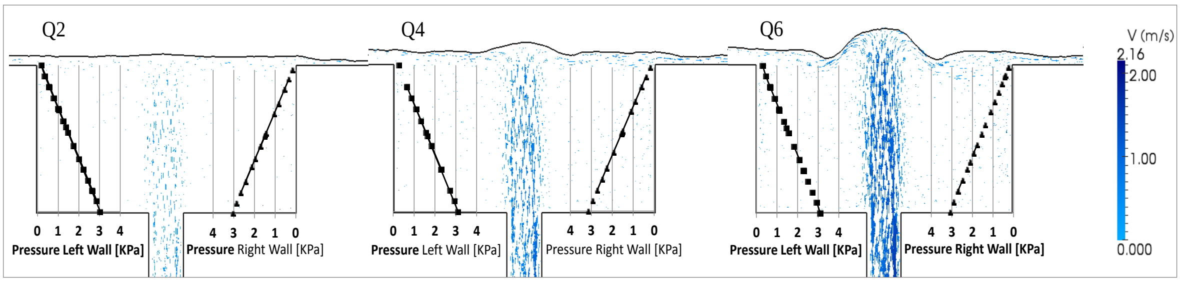

Figure 7. Pressure at left and right wall for Q2, Q4 and Q6 simulations. At background can be seen the velocity vectors in a blue scale.

Figure 8 shows the velocity and the pressure at the gully bottom. It can be seen that the velocity profile in the inlet is asymmetric, tending to the right. This may be caused by the curve at inlet. The pressure in the bottom is almost constant and equal to 3KPa, value that could be compared with hydrostatic value.

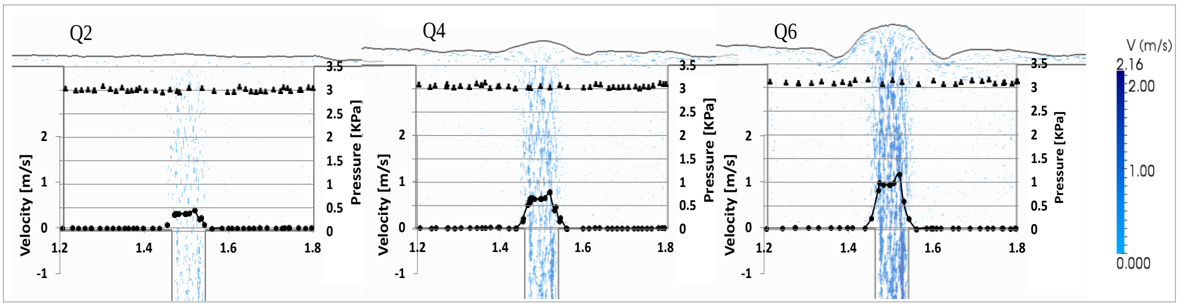

Figure 8. Velocity and pressure at gully bottom. The left scale is for the velocity [m/s] and the right scale for pressure [kPa]. At background can be seen the velocity vectors in a blue scale.

Figure 9 presents the stream lines in a middle transversal section. In lower flows (Q2) it can be seen two vortices on the left side of the gully, while only one large vortex and several small exist on the right side . This asymmetry is mainly due the fact that on the left side of the gully, the flow is almost static and thus the vortex is trapped into the gully. A higher degree of symmetry was found for the largest flow (Q6).

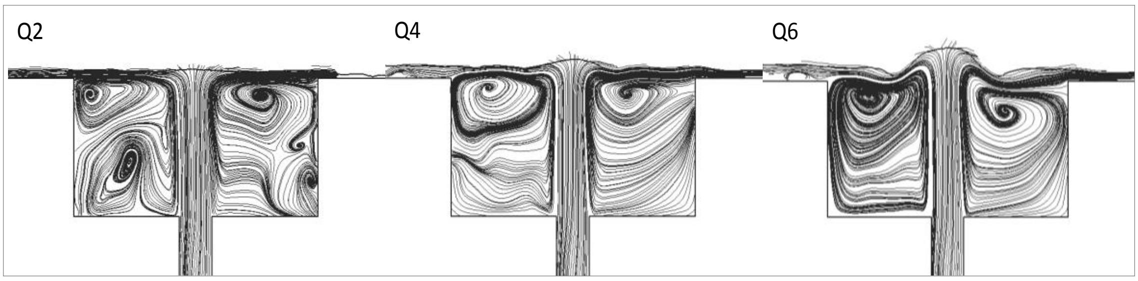

Figure 9. Stream lines for Q2, Q4 and Q6 simulations.

One point centred in the top of the gully, illustrated in Figure 10, was chosen to study 3D behaviour of the free surface and the jet. Figure 11 shows the velocity on the three directions x, y and z for the chosen point.

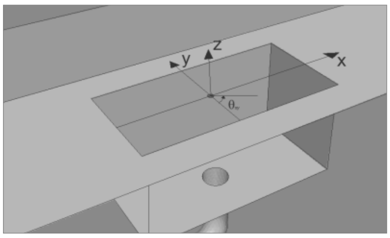

Figure 10. Centre point on top of gully and global directions.

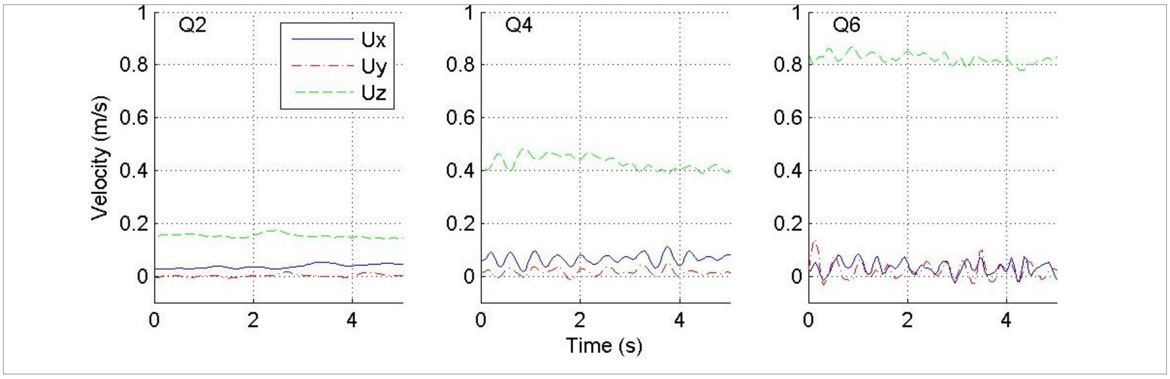

Figure 11. Velocities in directions x, y and z in centre point on top of gully.

The speed direction was calculated as a function of an angle theta (showed in Figure 10) and was found that the oscillation of this angle increases with increasing of the flow (Figure 12). For Q2, the oscillation occurs around 1.5 radians, for Q4, around 1.7 radians and for Q6 around 2 radians. It can also be seen that in Q2 and Q4, velocity direction only occurs in first and second quadrant, only in positive direction of xx. For Q6, all the directions were founded which shows the more unsteady character of Q6.

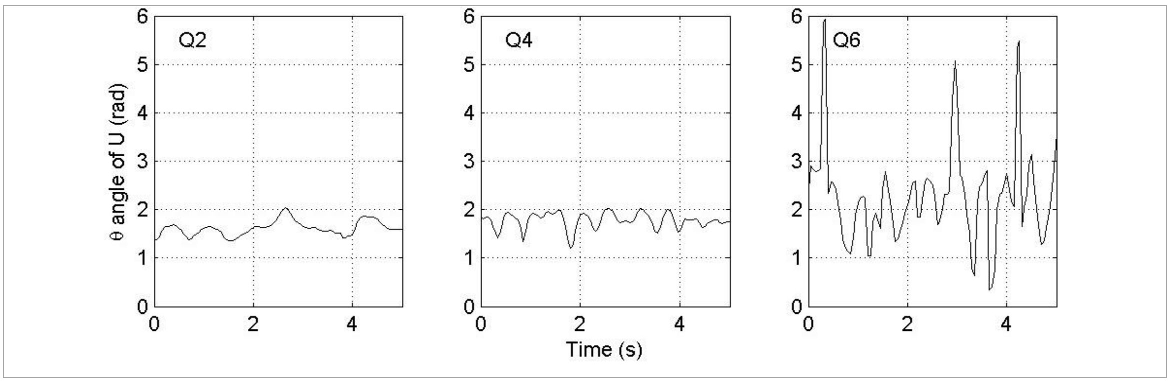

Figure 12. Angular variation of velocity in centre point on top of gully.

4 CONCLUSIONS

In this paper, the numerical CFD model OpenFOAM^{TM} was used to simulate the complexity of flows observed in a gully under surcharge conditions produced in the laboratory. The experimental and numerical results presented prove that computational simulations can be a useful tool to fully characterize the behaviour of a gully under surcharge conditions. The validation with experimental results and the comparisons of water depths in the channel including over the gully highlight the importance of the mesh generation and the boundary conditions. The analysis of the numerical results demonstrates the potential of simulations namely in regarding the oscillation of the jet and the direction of the flow.

5 ACKNOWLEDGEMENTS

The authors would like to acknowledge founding through the FCT ("Fundação para a Ciência e Tecnologia") and the COMPETE ("Programa Operacional Temático Factores de Competitividade"), supported by FEDER ("Fundo Europeu de Desenvolvimento Regional") through project PTDC/AAC-AMB/101197/2008. The help provided by Mr. Joaquim Cordeiro da Silva.

The first author's work was conducted under a Scientific Researcher IMAR Grant of the Research Project PTDC/AAC-AMB/101197/2008.

6 REFERENCES

Carvalho, R. F., Leandro, J., Martins, R., & Lopes, P. (2012). Numerical study of the flow behaviour in a gully. *4th IAHR International Symposium on Hydraulic Structures* (pp. 1-8). Porto, Portugal.

Deardorff, J. W. (1970). A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers. *Journal of Fluid Mechanics, 41*, 453-480. doi:10.1017/S0022112070000691

Directive 2007/60/CE. (2007). Directive 2007/60/EC of the European Parliament and of the Council. *Official Journal of the European Union, 299*, 27-34. Retrieved from http://eur-lex.europa.eu/LexUriServ/LexUriServ.do?uri=OJ:L:2007:288:0027:0034:EN:PDF

Djordjević, S., Saul, A. J., Tabor, G. R., Blanksby, J., Galambos, I., Sabtu, I., & Sailor, G. (2011). Experimental and numerical investigation of interactions between above and below ground drainage systems. *12th International Conference on Urban Drainage* (Vol. 1999, pp. 10-15). Porto Alegre/Brazil.

Fluent Inc. (2003). Determining Turbulence Parameters in FLUENT. Retrieved January 10, 2012, from http://jullio.pe.kr/fluent6.1/help/html/ug/node178.htm

Furbo, E., Harju, J., & Nilsson, H. (2009). *Project 9 - Evaluation of Turbulence Models for Prediction of FLow Separation at a Smooth Surface*.

Hirt, C. W., & Nichols, B. D. (1981). Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. *Journal of Computational Physics, 39*, 201-225.

Issa, R. I. (1986). Solution of the implicitly discretised fluid flow equations by operator-splitting. *Journal of Computational Physics*, *62*(1), 40-65. doi:10.1016/0021-9991(86)90099-9

Kortelainen, J. (2009). *Meshing Tools for Open Source CFD - A Practical Point of View* (pp. 1-25). Espoo, Finland.

Kulakov, Y. (2010). Salome to OpenFOAM mesh conversion tutorial. Retrieved from http://staff.um.edu.mt/__data/assets/pdf_file/0016/106144/Salome_to_OpenFOAM.pdf

Launder, B. E., & Spalding, D. B. (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, *3*(2), 269-289. Elsevier. doi:10.1016/0045-7825(74)90029-2

Lencastre, A. (1987). *Handbook of Hydraulic Engineering* (English tr., p. 540). John Wiley & Sons Inc.

Lilly, D. K. (1966). The Representation of Small-Scale Turbulence in Numerical Simulation Experiments. *NCAR MANUSCRIPT*, *281*, 1-24. Retrieved from http://nldr.library.ucar.edu/repository/assets/manuscripts/MANUSCRIPT-000-000-000-806.pdf

Martins, R. (2011). *Estudo do comportamento hidráulico de sumidouros em modelo numérico OpenFOAM*. Master Thesis - Universidade de Coimbra.

Moeng, C.-H., & Wyngaard, J. C. (1988). Spectral Analysis of Large-Eddy Simulations of Convective Boundary Layer. *Journal of the Atmospheric Sciences*, *45*(23), 3573-3587.

OpenFOAM. (2010). OpenFOAM v1.7.1 Download. Retrieved from http://www.openfoam.org/archive/1.7.1/download/index.php

OpenFOAM. (2011a). *OpenFOAM - The Open Source CFD Toolbox - User Guide*. OpenCFD.

OpenFOAM. (2011b). OpenFOAM User Manual - 7.2 Turbulence models. Retrieved December 20, 2011, b from http://www.openfoam.org/docs/user/turbulence.php

Roque, J. M. (2011). *Medição de alturas de água usando visão computacional num modelo Simulink* ®. Universidade de Coimbra.

Salome. (2011). SALOME 6 Platform Download. Retrieved from http://www.salome-platform.org/downloads/salome-v6.4.0

Smagorinsky, J. (1963). General Circulation Experiments with the Primitive Equations. *Weather Review*, *91*(3), 99-164. doi:10.1126/science.27.693.594

Ubbink, O. (1997). *Numerical prediction of two fluid systems with sharp interfaces*. Imperial College of Science, Technology & Medicine.

Versteeg, H. K., & Malalasekera, W. (1995). *An Introduction to Computational Fluid Dynamics - The Finite Volume Method*. *Longman Scientific & Technical* (1st ed., pp. 75, 78 and 154).

YoutubeVideo. (2007). Flooded Sewer Explosion. Retrieved February 6, 2012, from http://www.youtube.com/watch?v=9f4wS3hxHYg