Experimental and numerical investigation of interactions between above and below ground drainage systems

Slobodan Djordjević, Adrian J. Saul, Gavin R. Tabor, John Blanksby, Istvan Galambos, Nuridah Sabtu and Gavin Sailor

ABSTRACT

This paper presents the results of the experimental and numerical investigation of interactions between surface flood flow in urban areas and the flow in below ground drainage systems (sewer pipes and manholes). An experimental rig has been set up at the Water Engineering Laboratory at the University of Sheffield. It consists of a full scale gully structure with inlet grating, which connects the 8 m^2 surface area with the pipe underneath that can function as an outfall and is also further connected to a tank so that it can come under surcharging conditions and cause outflow from the gully. A three-dimensional CFD (Computational Fluid Dynamics) model has been set up to investigate the hydraulic performance of this type of gully inlet during the interactions between surface flood flow and surcharged pipe flow. Preliminary results show that the numerical model can replicate various complex 3D flow features observed in laboratory conditions. This agreement is overall better in the case of water entering the gully than for the outflow conditions. The influence of the surface transverse slope on flow characteristics has been demonstrated. It is shown that re-circulation zones can form downstream from the gully. The number and size of these zones is influenced by the transverse terrain slope.

Key words | CFD, flood modelling, gullies, urban pluvial flooding

Slobodan Djordjević (corresponding author)
Gavin R. Tabor
Istvan Galambos
University of Exeter,
Centre for Water Systems,
North Park Road,
Exeter EX4 4QF,
UK
E-mail: *s.djordjevic@exeter.ac.uk*

Adrian J. Saul
John Blanksby
Nuridah Sabtu
Gavin Sailor
Department of Civil and Structural Engineering,
University of Sheffield,
Mappin Street,
Sheffield S1 3JD,
UK

INTRODUCTION

The 'normal' working mode of gullies (inlets, catch pits) is that they collect surface runoff and convey it to the below ground drainage network manholes and pipes. During heavy rainfall and flood events, flow through gullies and manholes may alternate direction from surface to below ground and vice versa. Flow in both directions may be under free overflow (or free outflow) conditions or it may be partially or fully submerged, with transition zones between all mentioned regimes. In addition, whilst changes in surface flow rates may usually be considered as gradual, pressure changes in sewer pipes may be fairly rapid following surcharging and accompanying transients and instabilities. Finally, the geometry of numerous types of gully structures and manholes is such that flow in them during flood events is strongly three-dimensional (3D).

Therefore, the interactions between surface flood flow in urban areas and the flow in the below ground drainage systems can be very complex indeed. These interactions often take place through gully structures and also through the holes on manhole covers or following their partial or complete removal due to high pressure in the manhole caused by surcharging flow in sewer pipes. Probably the most extreme event of this kind for which a good quality video (CCTV) recording exists is the one that occurred in Calgary in Alberta, Canada on 7 March 1999 (http://www.youtube.com/watch?v=9f4wS3hxHYg).

Clearly, an understanding of the complex interactions between the above and below ground drainage systems is essential for calibration and reliable application of the coupled 1D/1D and 1D/2D urban flooding models (Leandro *et al.* 2009; Saul *et al.* 2011). Such models are increasingly being used to control the distribution of runoff between surface and sub-surface systems (or major and minor systems as they are termed in the context of dual drainage). Coupled models are used to simulate laying out and sizing of both systems and for managing

doi: 10.2166/wst.2012.570

and rationalizing the connections between these two systems via the approach known as *designing for exceedance in urban drainage* (Balmforth *et al.* 2006), for flood impacts assessment (Dawson *et al.* 2008) as well as for other types of analysis required for an effective urban flood risk management.

General references in this subject often disregard the details of the interaction between these two systems. The most common way in which this has been modelled is by using either a weir or on orifice equation, or a combination of the two. However, the parameters in these equations are highly uncertain and they are not fully representative of the real flow conditions. Consequently, this element is a notable contributor to the uncertainty of urban flood simulation.

Experimental data exist for the assessment of the inlet grating capacity at their 'normal' functioning mode and design methods for spacing of gullies in road profiles exist (e.g. Guo 2000), however very little experimental data and computational modelling experience exist about sub-surface/surface interactions during flood events.

In order to study this problem, an extensive research programme has been conducted within the UK Flood Risk Management Research Consortium (FRMRC) phase 2 Work Package 3.7. This research has two closely linked strands:

- Two laboratory systems have been built at the Water Engineering laboratory at the University of Sheffield, one of which is described in this paper. It consists of a full scale gully structure with inlet grating, which connects the 8 m^2 surface area with the pipe underneath.
- A 3D CFD (Computational Fluid Dynamics) model has been set up to investigate the hydraulic performance of this type of gully inlet during the interactions between surface flood flow and surcharged pipe flow.

The first step in this research was to compare CFD modelling results with experiments, both qualitatively and quantitatively. The ultimate aim is to improve understanding of such interactions and enable determination of the head-discharge relationships (required by coupled models) that would be much more reliable than those currently in use. This will be achieved by extending the acquired knowledge to flow conditions beyond those that will have been studied experimentally and for other gully geometries. This paper describes the preliminary results of this study.

EXPERIMENTAL SET-UP

The experimental system was established to replicate a full scale gully system (see Figure 1), using full scale industry manufactured components. It consists of a full scale gully structure with inlet grating, which connects the surface area (4,270 mm by 1,830 mm) with the pipe underneath that can function as an outfall but is also further connected to a tank so that it can come under surcharging conditions. The flow for this system is provided by an overhead tank and is circulated through the entire system before being transferred into a sump to be pumped back to the overhead tank again. The gully itself is a trapped gully with spigot outlet, which is one of the most commonly used gully types designed with an outlet that forms a water seal and a rodding eye which helps to retain floating pollution within the gully pot. The gully has a 375 mm diameter and 750 mm nominal depth.

Figure 1 | Experimental system.

For an extensive range of flow rates to the gully, in the range 0–60 L/s, the depth of flow in the vicinity of the gully was measured using six pressure transducers positioned in the bed of the platform (Figure 1, right) and one transducer located at the base of the gully pot.

All experimental data is recorded in real time and the results were analysed to establish the conventional coefficient of discharge using a standard weir equation. Typical results are shown in Figure 2 for the 'terminal system' (where water leaves the experimental installation only through the gully) and the 'intermediate system', which permits a portion of the approaching flow to pass the gully and leave the system elsewhere. However the flow pattern on entry to the gully changes significantly with increase in flow rate and changes from a weir condition to a partial orifice and weir condition and finally to a full orifice condition, which is beyond the scope of this paper.

CFD MODEL

The Open source Field Operation and Manipulation (OpenFOAM) C++ libraries (Weller *et al.* 1998) were used to numerically analyse the dynamics of flow. The numerical method is based on the Reynolds Averaged Navier–Stokes equations, which describe the flow of a viscous incompressible fluid. The free surface is captured by the Volume of Fluid (VOF) method on arbitrary unstructured polyhedral meshes, in which a phase fraction function is used to distinguish between the two fluids (air and water) and to capture the interface between them. The turbulence closure is based on the k–ω turbulence model, which solves one equation for turbulent kinetic energy and a second equation for the specific turbulent dissipation rate (or turbulent frequency).

The solution is calculated using the Finite Volume approach and the PISO (Pressure Implicit with Splitting of Operators) algorithm is used to handle pressure–velocity coupling (Tabor 2010). It involves a momentum predictor and a correction loop in which a pressure equation based on the volumetric continuity equation is solved and the momentum is corrected based on the pressure change (Jasak 1996; Rusche 2002). Transport equations for the phase fraction as well as the turbulent kinetic energy and its dissipation rate are solved only once per iteration at the end of the sequence. Several computational meshes have been constructed, with up to 1.5 million elements. Figure 3 shows details of two mesh setups.

RESULTS AND DISCUSSION

Different setups (surface slopes and grating types) are considered in this study to investigate the interaction between surface flow and pipe flow. Figure 4 shows surface streamlines in the plan view as simulated at different transverse slopes (S_T). Relatively small transverse slope (1/200, Figure 4(a)) results in a single recirculation zone downstream from the gully and consequently water entering the

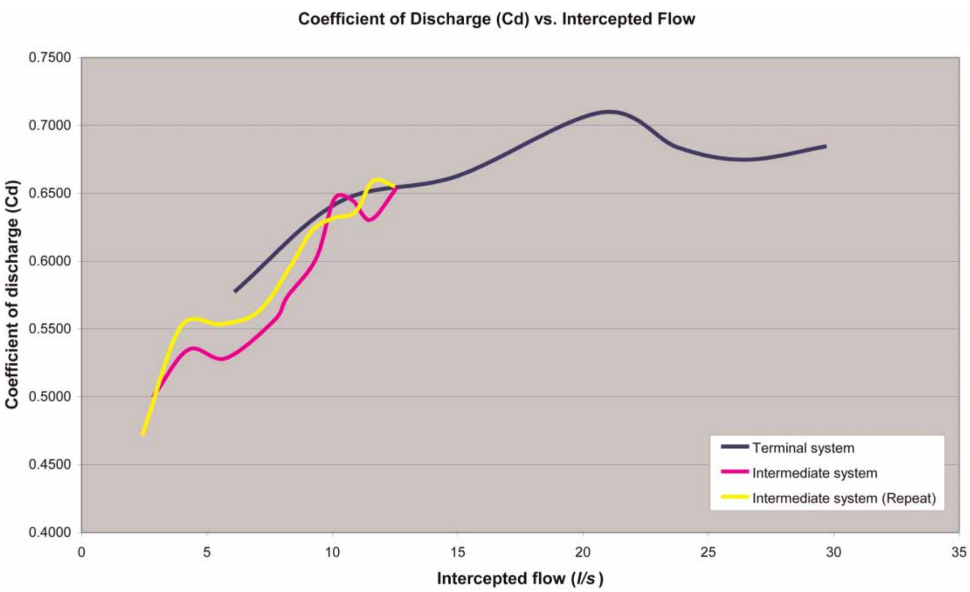

Figure 2 | Discharge coefficient as a function of intercepted flow.

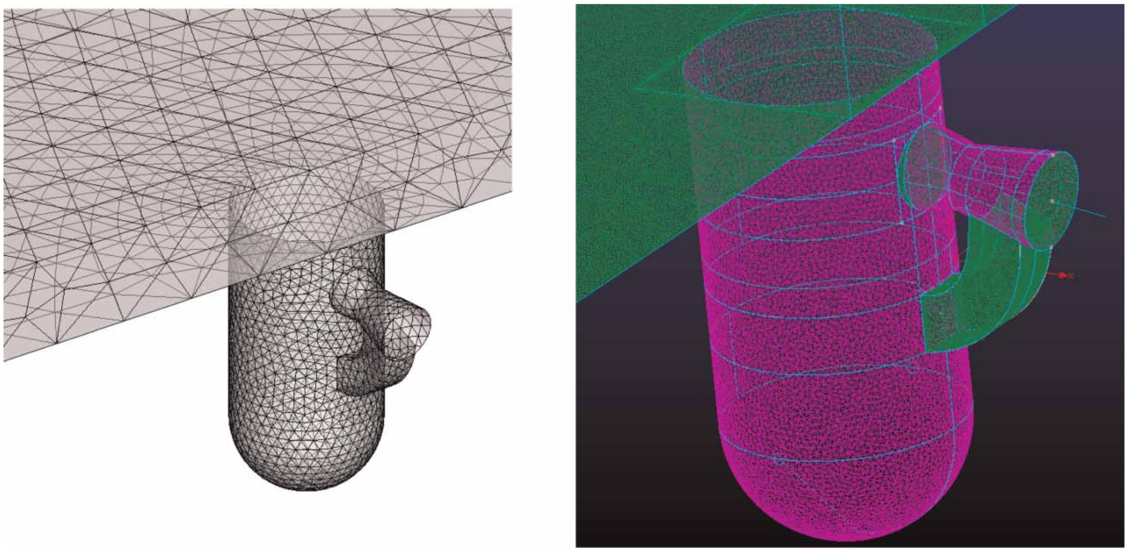

Figure 3 | Details of two computational meshes.

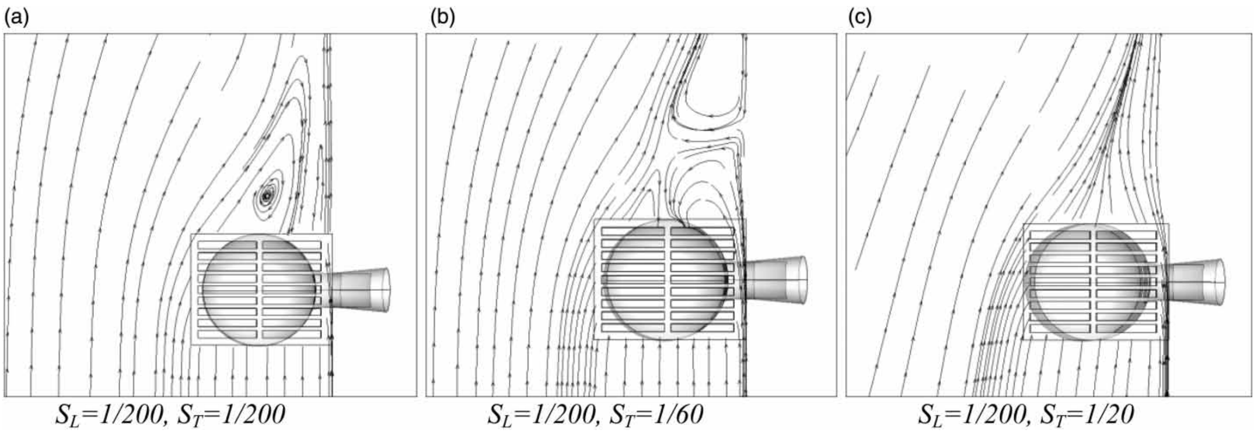

Figure 4 | The effect of transverse slope (S_T) variation on flow pattern formation (plan view), (a) $S_L = 1/200$, $S_T = 1/200$. (b) $S_L = 1/200$, $S_T = 1/60$. (c) $S_L = 1/200$, $S_T = 1/20$.

gully from all sides. At a larger slope (1/60, Figure 4(b)) two recirculation zones can be identified immediately next to the gully and another one further downstream. A fairly steep surface (1/20) resulted in no circulation zone and no inflow to the gully from its downstream side. Similar flow features have been observed in the experiments. This analysis is not so much relevant for the design of inlets but mostly to assess the ability of the turbulence model (implemented in the simulation model) to replicate complex flow features.

Figure 5 shows streamlines on the surface and in the gully for the flow situation shown in Figure 4(a) ($S_T = 1/200$). The colour code is based on the γ-parameter of the VOF method, whose values are 0 and 1 – blue (dark grey) and red (light grey) if a point is inside air or water and varies between 0 and 1 – blue (dark grey) to red (light grey) for the points inside the transitional area.

Comparison between measured and simulated flow depths at six measuring locations around the gully, for steady inflow of 36.7 L/s is shown in Figure 6. Red (grey) circles indicate depths simulated by the CFD model, whilst measured values are observed over a period of time, therefore they are shown as statistical measures of corresponding data series. The locations of measurement points (D1–D6) are indicated on the right hand side of Figure 6.

The agreement between observed and simulated depths is very good. Comparison of measurements with simulations for outflow from the gully (not shown here) was typically

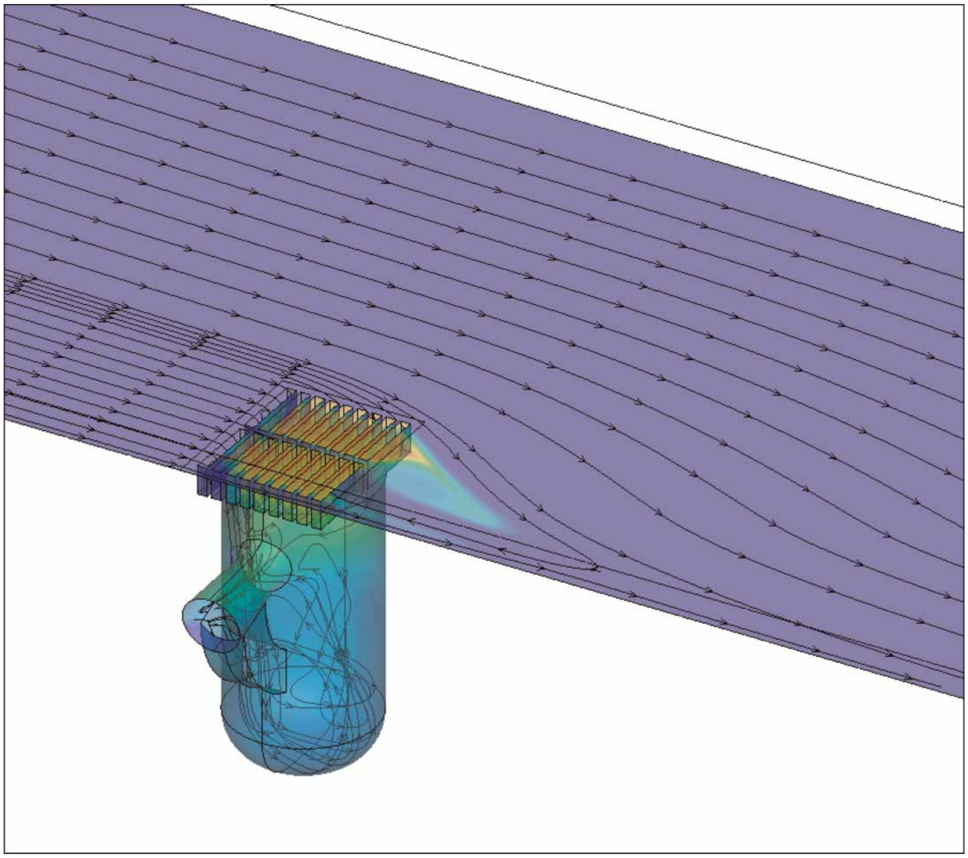

Figure 5 | Streamlines on the surface and in the gully. (The full colour version of this figure is available in the online version of this paper at http://www.iwaponline.com/wst/toc.htm.)

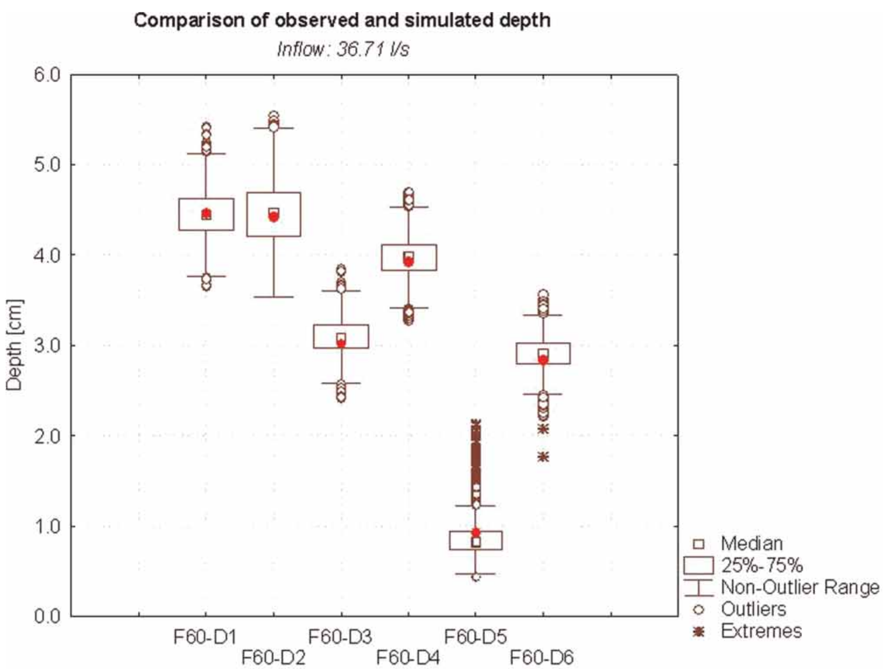

Figure 6 | Left: comparison between measured and simulated flow depths; right: measurement points. (The full colour version of this figure is available in the online version of this paper at http://www.iwaponline.com/wst/toc.htm.)

equally good for all four check points upstream and downstream from the gully but not as good for the two points aside from the gully (D3 and D4).

Figure 7 shows 16 snapshots at one second intervals of the unsteady flow simulation following sudden short-lived surcharging of the gully (70 L/s inflow to the initially filled up gully for 1 s). The cube shown in these images is only a cropped portion of the computational flow domain, which extends further through the top and the three sides of the cube selected here for the presentation of results.

It can be seen that initially (Figure 7, top row, first image) the gully is empty and the surface is dry. Once the

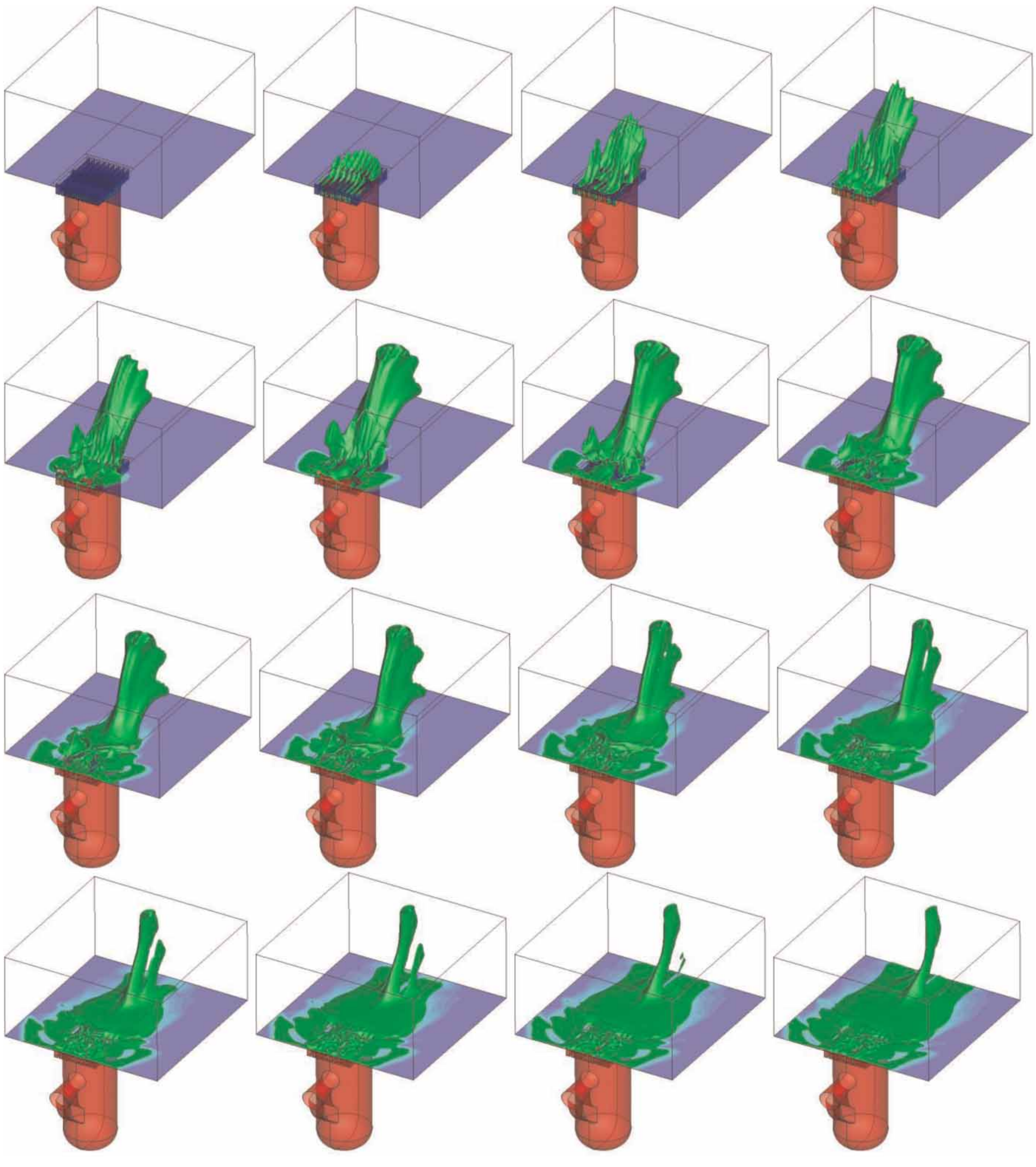

Figure 7 | Snapshots of the simulation of sudden surcharging of the gully and the consequent outflow (note: the top and three sides of the cube shown are not the boundaries of the computation domain).

water starts entering the gully from the pipe connection underneath (top row, second image), the water is leaving the gully through the inlet grating in a fashion that reflects water pressure distribution within the gully. Whilst the movement of the fountain leaving the grating is rapid, the spreading of water on the surface is relatively slow. The water volume on the surface remains more or less symmetrical throughout the part of the simulation that is presented. Drying and wetting of the surface progresses as the water is moving on the surface.

For a number of reasons, it would be too optimistic to expect that sudden flow changes such as those shown in Figure 7 could be simulated very accurately. First of all, on the experimental rig, a boundary condition that would lead to such a fast surcharging would be a sudden opening of the valve. The corresponding fast change in flow rate, velocity or pressure would not be possible to precisely replicate as a boundary condition in the CFD simulation and therefore for this situation (sudden fast outflow) any comparison between observations and simulation could only be qualitative. Second, experience with CFD modelling of free jets – for example, for the ski-jump flow (Bennett *et al.* 2011) – has shown that the simulated length reached by the jet may differ significantly from experimental observation.

In that respect, the images shown in Figure 7 are not intended to demonstrate ability to accurately predict very rapid flow changes during the outflow from the gully, but they clearly indicate that the model can simulate even the most complicated flow situations and produce qualitatively realistic results. It is therefore expected that the CFD simulation of steady flows and gradually varying unsteady flows in a range of regimes (inflow/outflow, free/submerged) will enable calibration and validation of the model for the setup studied experimentally. This knowledge will then enable a more reliable definition of head/discharge curves for any other geometry.

CONCLUDING REMARKS

Experimental and computational modelling results presented in this paper demonstrate that there is room for improvement of our understanding of interactions between above and below ground drainage systems. Complex flow features observed in laboratory conditions have been replicated by the CFD model. The agreement between observed and calculated steady flow depths on the surface is very good. This agreement is overall better in the case of water entering the gully than for the outflow conditions. The simulation model's ability to simulate unsteady outflow from the fast surcharging gully has been demonstrated, though it was not yet possible to test this experimentally. This is ongoing research and some preliminary results have been presented in this paper.

The influence of the surface transverse slope on flow characteristics has been demonstrated. It is shown that recirculation zones may form downstream from the gully. The number and size of these zones is influenced by the transverse terrain slope.

It is worth noting that – to the authors' best knowledge – this is the first time that this type of flow is studied experimentally on a full scale (1:1 scale) model and numerically simulated using a 3D model, therefore this is a pioneering work. It is expected that final results of the project presented herein will contribute to a better understanding of interactions between sub-surface and surface drainage systems. This will ultimately lead to the reduction of uncertainties in the application of coupled 1D/1D and 1D/2D urban flood models, which is the next step in this research.

ACKNOWLEDGEMENT

The authors would like to acknowledge funding through the Flood Risk Management Research Consortium 2 (FRMRC2) as well as Engineering and Physical Sciences Research Council (EPSRC) grant EP/I012222/1. The FRMRC2 is supported by Grant EP/F020511/1 from the EPSRC, in partnership with the Department of Environment, Food and Rural Affairs/Environment Agency (Defra/EA) Joint Research Programme on Flood and Coastal Defence, United Kingdom Water Industry Research (UKWIR), the Office of Public Works (OPW) Dublin and the Northern Ireland Rivers Agency (DARDNI).

REFERENCES

Balmforth, D., Digman, C., Kellagher, R. & Butler, D. 2006 Designing for exceedance in urban drainage – good practice. CIRIA Report C635, London.

Bennett, J., Dolecki, C., Fung, F., Heal, D., Holdcroft, L., Knights, R. & Sweetapple, C. 2011 Hydraulic design of ski-jump spillways. Group Project Report, College of Engineering, Mathematics and Physical Sciences, University of Exeter.

Dawson, R. J., Speight, L., Hall, J. W., Djordjević, S., Savić, D. & Leandro, J. 2008 Attribution of flood risk in urban areas. *Journal of Hydroinformatics* **10** (4), 275–288.

Guo, J. C. Y. 2000 Design of grate inlets with clogging factor. *Advances in Environmental Research* **4** (3), 181–186.

Jasak, H. 1996 Error analysis and estimation for the finite volume method with applications to fluid flows. PhD Thesis, Imperial College.

Leandro, J., Chen, A. S., Djordjević, S. & Savić, D. A. 2009 A comparison of 1D/1D and 1D/2D coupled (sewer/surface) hydraulic models for urban flood simulation. *Journal of Hydraulic Engineering* **135** (6), 495–504.

Rusche, H. 2002 Computational fluid dynamics of dispersed two-phase flows at high phase fractions. PhD Thesis, Imperial College.

Saul, A. J., Djordjević, S., Maksimović, Č. & Blanksby, J. 2011 Integrated urban flood modelling. In: *Flood Risk Science and Management* (G. Pender & H. Faulkner, eds). Wiley-Blackwell, Chichester, Chapter 13, pp. 258–288.

Tabor, G. R. 2010 OpenFOAM: an Exeter perspective, *5th ECCOMAS CFD Conference*, Lisbon, Portugal, 14–17 June 2010.

Weller, H. G., Tabor, G., Jasak, H. & Fureby, C. 1998 A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in Physics* **12** (6), 620–631.

First received 10 November 2011; accepted in revised form 28 August 2012