Computational fluid dynamics as a tool for urban drainage system analysis: A review of applications and best practice

D.S. Jarman[1, 2]*, M.G. Faram[1], D. Butler[2], G. Tabor[2], V.R. Stovin[3], D. Burt[4] and E. Throp[5]

[1] Hydro International, Shearwater House, Clevedon Hall Estate, Victoria Road, Clevedon, BS21 7RD, UK
[2] Centre for Water Systems, School of Engineering, Computing and Mathematics, University of Exeter, North Park Road, Exeter, EX4 4QF, UK
[3] Department of Civil and Structural Engineering, University of Sheffield, Western Bank, Sheffield, S10 2TN, UK
[4] MMI Engineering, Park House, 10 Park Street, Bristol, BS1 5HX, UK
[5] Ansys-Fluent, Sheffield Business Park, 6 Europa View, Sheffield, S9 1XH, UK

**Corresponding author, e-mail daniel.jarman@hydro-international.co.uk*

ABSTRACT
Computational Fluid Dynamics (CFD) can be applied to gain insights into most fluid processes and associated phenomena and so presents potential to add value in the analysis of urban drainage systems. This paper presents a review of CFD studies carried out in this field, with the objective of developing an appreciation of how and where it can be applied. Existing work has tended to focus around the analysis of four types of urban drainage structure, including Combined Sewer Overflows (CSOs), storage and attenuation systems, stormwater sediment interceptors and sewerage conveyance structures. Within the respective studies, the prediction of flowfields, particulate behaviour, water surface profiles and Residence Time Distributions (RTDs) are found to form the main focus, and as such, these are considered in most detail in the paper. It is concluded that CFD presents a number of opportunities in urban drainage system analysis, and that the scope of this opportunity will further develop as both computational hardware and software resources become more advanced.

KEYWORDS
CFD, CSO, modelling, residence time, sediment, sewerage infrastructure, simulation, storage

INTRODUCTION
Computational Fluid Dynamics (CFD) is a group of techniques aimed at solving the Navier-Stokes equations (or strictly, Reynolds-Averaged Navier-Stokes equations in most cases), thereby satisfying the conservation of mass, momentum and energy to predict the behaviour of fluidic systems. In its modern guise as a computer-aided engineering (CAE) software, CFD presents itself as a useful tool for investigating domain space for physical system design and performance variables, and for diagnosing or troubleshooting system behaviour. Typical scenarios where the application of CFD may complement or replace existing analytical techniques are when a high number of design variations are to be analysed or where physical testing may be prohibited due to restricting factors, such as scale, cost, accessibility, or the presence of physical or environmental hazards. CFD is especially prevalent in cases where modelling methodologies have been previously validated or operational data for validation is easily obtainable.

Key factors influencing the uptake of simulation techniques such as CFD have included business and legislative drivers demanding technological development and efficiency improvements. The advancement and increased accessibility of knowledge, techniques and computational resources from academia, software and hardware developers have also driven the use of such simulation techniques. The application of CFD to urban drainage systems is evident from the mid-1990's, with established modelling and evaluation techniques relating to specific technical scenarios becoming particularly evident during the early 2000's.

With established modelling techniques becoming more available, and with the objective of assisting users considering the use of CFD in selecting appropriate modelling methodologies, this paper reviews a range of studies relating to the application and development of CFD modelling techniques for urban drainage system analysis. The experience base of the Authors is drawn upon, representing an active cross section from the UK CFD and urban drainage community with backgrounds in academia, CFD code and software development, CFD/engineering consultancy and urban drainage equipment development and supply.

CFD APPLICATION IN THE FIELD OF URBAN DRAINAGE

CFD methods can be applied to gain insights into most fluid processes and associated phenomena, and so have the ability to add value in the study of urban drainage systems. Overviews of the opportunities for the use of CFD in urban drainage related applications are presented by Ta (1999), Faram and Harwood (2000) and Harwood (2006), verifying the compatibility and benefit of CFD for urban drainage system design and analysis.

A significant amount of the published information pertaining to the use of CFD for urban drainage applications indicates that it is most commonly applied in order to gain detailed design knowledge, or to evaluate structural or operational variables. Existing work has been focused around four typical urban drainage application areas. Table 1 identifies these, along with the associated studies relating to each area. It should be noted that these studies are concerned with the application of CFD for engineering objectives, rather than with the development of new CFD modelling methodologies.

Table 1: Urban drainage application areas utilising CFD and associated studies

Application area	Associated studies
Combined sewer overflows (CSOs)	Saul and Svejkovsky, 1994; Harwood and Saul, 1996; Faram and Andoh, 1999; Tyack and Fenner, 1999; Burt *et al.*, 2002; Lim *et al.*, 2002; Okamoto *et al.*, 2002; Pollert and Stransky, 2002; Burt and Balmforth, 2005; Kouyi *et al.*, 2005
Storage and attenuation structures	Ta and Brignal, 1998; Stovin and Saul, 2000; Schmitt *et al.*, 2002; Stovin *et al.*, 2002b; Faram *et al.*, 2004
Stormwater sediment capture devices	Faram and Harwood, 2002; Faram and Harwood, 2003; Gupta *et al.*, 2005; Stovin *et al.*, 2005
Sewerage conveyance structures	Bardiaux *et al.*, 2007; Lau *et al.*, 2007; Wertel *et al.*, 2007

Reviewing the studies outlined in Table 1, it is found that the focus of analysis can be divided into four main areas, listed below and discussed in greater detail later in the paper.

- Flowfield prediction
- Analysis of particulate behaviour
- Prediction of water surface profiles
- Analysis of system residence time distribution (RTD)

It should be noted that the prediction of water surface profiles, while seemingly fundamental, is actually absent from the majority of studies, in which free surfaces are often assumed to be flat and fixed, considerably reducing computational effort. This aspect of modelling is considered later. Table 2 quantifies the number of studies considering each of these types of output in the analysis, alongside the application areas identified previously in Table 1. Accurate prediction of flowfields within each application area is inherently important as a prerequisite to the correct prediction of other performance attributes and indeed, many early studies concentrated solely on consideration of this modelling output (Faram and Andoh, 1999). Prediction of water surface features may be considered an extension of flow pattern prediction for most systems, the solution of which has become more feasible as computational resources have developed.

Sediments are a fundamental consideration in all urban drainage systems. In reflection of this, after flowfield prediction, the simulation of sediment behaviour appears most frequently in the modelling studies. Prediction of water surface features and RTD characteristics follow closely in terms of analytical value and can still be related to all of the application areas covered in this paper. Although no engineering objective orientated studies were found that applied simulated RTD analysis to CSOs and sediment capture devices (Table 2), there were several that aimed at the development of these techniques, identifying these as emerging interest areas. Where appropriate, these are referred to later in the paper.

Table 2: Application areas and types of modelling output considered in the analysis

Application area	Modelling output			
	Flowfield prediction	Particulate behaviour	Water surface profile prediction	RTD analysis
CSOs	10/10	6/10	3/10	0/10
Storage and attenuation structures	5/5	4/5	2/5	2/5
Stormwater sediment capture devices	4/4	4/4	1/4	0/4
Sewerage conveyance structures	3/3	1/3	1/3	1/3

The majority of urban drainage studies reviewed utilised commercially-developed, general-purpose CFD codes. The Authors only discovered a single study obviously employing a user-developed code (Wertel *et al.*, 2007) applied to sewerage conveyance structures to improve instrumentation configurations. There may be more examples of user-developed or academic codes, but some studies did not explicitly state the code origin.

THE CFD MODELLING PROCESS

Best practice for the correct prediction of various flow phenomena is one of the most debated CFD topics. The CFD model set up process requires several stages specifying geometric and solution parameters prior to numerical analysis. Refinement of these stages may only be possible in a reactive manner, rather than proactive, after benefiting from the simulation results (example modelling procedure adapted from ANSYS (2007) presented in Figure 1). That said, the application of validated modelling practices in relation to specific application areas and modelling objectives often provides the most direct route to success.

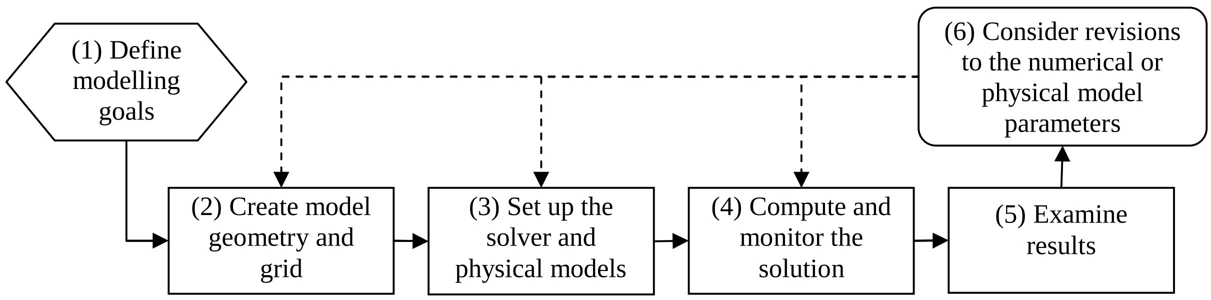

Figure 1: Iterative modelling procedure, adapted from ANSYS (2007)

CONSIDERATION OF SPECIFIC MODELLING OBJECTIVES

Flowfield prediction

The accurate simulation of flowfields using CFD is generally dependent on a robust approach to geometry and mesh generation and appropriate selection of the solver and physical model parameters (corresponding to stages 2 and 3 in Figure 1). Two important initial considerations are whether the flow is steady or unsteady and whether it is turbulent or laminar. For most urban drainage problems of practical interest the flows can generally be considered to be turbulent (one exception may be for low depth, low velocity surface run-off flow). Often, they can also be regarded as steady over a chosen timescale, which can significantly simplify the modelling process. However, this assumption is not always appropriate, for example, where the operation of a system is fundamentally dependent on the variable (often wet-weather related) nature of sewer flows. Schmitt *et al.* (2002) and Stovin *et al.* (2002b) have carried out dynamic simulations of storage chambers, this approach being necessary to take into account the characteristically time-dependent nature of their operation.

Prediction of the flow domain is sensitive to the size of the mesh cells. Although there are no definitive rules governing ideal mesh resolution, the general approach is to analyse several meshes of increasing resolution to establish at what point the computational results become independent of the mesh resolution. Appropriate mesh resolution is also linked to the hydraulic conditions and the flow features the user is aiming to resolve. It is also affected by the numerical solution parameters, such as the discretisation schemes.

The differences in the selection of appropriate mesh resolutions are highlighted by comparing the work of Ta and Brignal (1998) and Lau *et al*. (2007), which both utilised hexahedral cells and k-ε type turbulence models. The work by Ta and Brignal (1998) modelled a full scale storage reservoir in three dimensions using approximately 7,000 cells, and it was stated that "[a] finer mesh had not provided additional information for this application". In contrast, in the analysis by Lau *et al*. (2007), focusing on a manhole with geometry several orders of magnitude smaller than the reservoir, the mesh sensitivity limit was found to be between 55,000 and 130,000 cells. This emphasises that mesh resolution is largely dependent on the phenomena the modeller wishes to resolve. Additionally, older studies may have been more constrained by the computational resources available at the time. A general trend towards increasing mesh resolution can be observed in relation to advancements made in this regard.

Of the twenty-two studies identified in Table 1, six used a Reynolds Stress Model (RSM) and sixteen used a k-ε or re-normalised group (RNG) k-ε variant as the turbulence scheme. Stovin *et al*. (2002a) performed comparisons of all of the above mentioned turbulence models, as well as evaluating mesh sensitivity and discretisation effects for three-dimensional open channel flow. The findings indicate that the RSM with quadratic pressure strain (QPS) gave the closest match to laboratory findings with many secondary flow features being resolved.

A significant finding by Buxton *et al*. (2002) was the inability of a two dimensional model to resolve the secondary currents normally observed in open channel flow. This is similar to the effect the isotropy of k-ε turbulence models can cause. As a result of this, if the flow is swirling or dominated by large shear zones a three dimensional RSM model is preferable. However, where shear zones are small compared to the flow region, a k-ε turbulence model will offer significant computational advantages over the RSM (ANSYS, 2007).

Analysis of particulate behaviour

Three separate methods have been applied in analysing sediment removal or build-up in relation to urban drainage applications:

- Bed shear stress analysis
- Lagrangian particle tracking
- Eulerian phase tracking

The bed shear stress technique was applied by Stovin and Saul (1996; 2000). This modelling approach relates flow parameters, in this case bed shear stress, to the fraction of total suspended solids (TSS) deposited and is similar to that presented in the CIRIA guidelines for determining self-cleansing ability of sewerage pipelines (Ackers *et al.*, 1996). Stovin and Saul indicated that this modelling approach was promising, but that it was limited by certain conditions, namely it could only predict a sedimentation ratio of up to 60% of TSS and was reliant on experimental data to relate particle size and deposition ratio to bed shear stress. The technique was further developed by Adamsson *et al.* (2003) who used a critical bed shear stress boundary condition combined with Lagrangian particle tracking to simulate saltation, overcoming the sedimentation ratio restriction.

The Lagrangian particle tracking method calculates discrete particle trajectories in a Lagrangian reference frame using well established buoyancy and drag laws. Turbulent dispersion of particles is accounted for via a stochastic scheme. This modelling approach has been utilised in many studies, but early applications of this technique were performed by Saul and Svejkovsky (1994) and Stovin and Saul (1996) to investigate CSO and storage chamber behaviour respectively. These studies confirmed this method as promising, although it highlighted that the scheme is sensitive to its parameters and over predicted particle retention performance compared to experimental data. A subsequent study by Stovin and Saul (1998) concluded that an adequate number of Lagrangian time steps and selection of appropriate wall effects were critical to the integrity of the results, and also that small changes in the particle injection location caused significant variations in the results. Additionally, Burt *et al*. (2002) and Faram and Harwood (2003) established that, due to the stochastic nature of the model, a minimum sample size of 500 particles was required to produce particle quantity independent results. Further studies by Egarr *et al*. (2004) investigated several modelling parameters and highlighted that length scale, particle shape factor, and fluid temperature could also significantly affected Lagrangian predictions.

Faram and Harwood (2002; 2003) used Lagrangian particle tracking to assess the performance of stormwater sediment interceptors, devices in which captured sediments are stored in a base sump. These studies found that sediment removal is a dynamic interaction between sediment deposition and re-suspension. A conclusion of this was that, to enable comparison with experimental data or between different modelled configurations, particle tracking simulation times (i.e. from injection to termination) needed to be consistent between models and experimental practice.

It has been noted by the Authors that Lagrangian or Eulerian tracking approaches generally take into account only particle buoyancy and drag forces, neglecting lesser forces such as lift. Additionally, while several examples of validations for quiescent settling are available (Buxton *et al.*, 2002; Egarr *et al.*, 2004) there are no such examples for dynamic settling (i.e. settling under turbulent conditions). This is perhaps due to a lack of understanding regarding dynamic settling (Bagchi and Balachandar, 2003). For these reasons there may be some level of uncertainty when applying these techniques to turbulent systems without the application of appropriate drag and lift models, or extension of the understanding of these phenomena.

Burt *et al.* (2002) and Schmitt *et al.* (2002) apply and discuss a third sedimentation modelling procedure based on the Eulerian approach in the context of analysing CSOs and storage tanks respectively. This approach simulates particle behaviour via kinetic theory models and interphase drag laws, where coupling of the fluid and particulate phases is achieved through the conservation equations and phase fraction. In Eulerian models momentum is generally transferred in both directions between the fluid and particulate phases. As Eulerian-based models treat the particulate domain as a continuum, the particulates also inherently transfer momentum between oneanother. This allows deposition, accumulation and resuspension behaviour to be simulated, along with the resultant effect on the flow. Burt *et al.* (2002) concluded that the Eulerian method is superior to Lagrangian particle tracking for systems where sediment hold-up may be important to system behaviour.

Prediction of water surface profiles

In the majority of the studies identified in Table 1, and as is apparent in Table 2, water free surface profiles were not predicted as part of the solution process. Instead, in these studies, the free surface was approximated by a fixed, horizontal frictionless boundary, predefined as part of the model geometry. However, this type of approach is not always appropriate.

Whilst more computationally expensive, free surface profiles can be calculated as part of the model solution using a reduced form of the Eulerian multiphase model, known as the VOF (Volume of Fluid) model, which formulates the water-air interface. VOF modelling is useful in specific cases, for example where the water surface geometry is complex or transient, such as where hydraulic jumps are present. An example of a direct need for VOF modelling is presented by Kouyi *et al.* (2005), where water surface predictions were used to make recommendations to instrument a large CSO structure. However, VOF modelling may require a higher mesh resolution to fully resolve the water-air interface. Burt (2002) presents the findings of a study to compare the effect of using horizontal and profiled fixed frictionless boundary approximations, and of resolving the free surface using VOF methods, when predicting flowfields in a standard design of CSO chamber. The study found that a profiled frictionless boundary gave predictions that corresponded more closely to theoretical results than the horizontal boundary method, and that were similar to those from the VOF method.

Calculation of residence time distribution (RTD)

RTD provides a measure of the degree of flow mixing within a system. It is often used as an indicator of the likely performance of systems designed for removing solids from sewer flows (e.g. storm tanks), or for providing an assessment of chemical contact times (e.g. in CSO chambers incorporating disinfection). RTD can be measured experimentally by injecting a ‘tracer’ into the inlet of the system in question and then monitoring its discharge at the outlet. Two examples of tracer and detector combinations applied to urban drainage systems are; lithium chloride with an atomic absorption spectrophotometer (Tyack and Fenner, 1998); and a fluorescing tracer with a submersible fluorometer (SCUFA) (Guymer *et al.*, 2002).

Three separate methods have been applied to assessing the RTD characteristics of urban drainage systems using CFD:

- **Lagrangian particle tracking:** Dosing a known number of particles into the inlet and measuring their escape times from the system to produce an escape frequency profile.
- **Eulerian phase or scalar tracking:** Dosing a simulated phase or scalar with a known concentration into the inlet and monitoring this phase as it escapes from the system
- **User-defined scalar (UDS) local age method:** Calculation of the local mean age within the system using a transport scalar

These methods essentially mimic experimental methods. Both the Lagrangian and Eulerian methods provide scope to gain insights into the behaviour of solutes or sediments, through the appropriate definition of the particle properties. Analysis for solutes and sediments using Lagrangian methods have been performed by Lau *et al.* (2007) and Faram *et al*. (2004) respectively. The Eulerian method was applied by Ta (1998) to investigate the effect of baffle configurations on RTD in storage reservoirs.

The UDS method for the calculation of local mean age within the system using a transport scalar was developed by Roos (1999) for Heating, Ventilation and Air Conditioning (HVAC) applications. Egarr *et al.* (2005b) utilised this method for investigating the RTD of vortex separators designed for CSO treatment and compared this to other approaches. The methods were then validated against experimental data presented by Alkhaddar *et al.* (1999). Egarr *et al*. (2005a) determined that the UDS method gave the closest predictions to the experimental data, with the mean error between the data and the predictions being 11%.

VALIDATION

The validation of CFD simulations can present challenges. It is often the case that the reason CFD is being carried out in the first place is that it would be difficult or expensive to conduct experiments. Validation is, nevertheless, important.

At the base level, validation should be carried out, where possible, on system flowfields, as these will often determine the accuracy of secondary outputs such as particle trajectories and RTD. Often in practice, it is only possible to obtain experimental data at certain points in a flow, which can be collected using techniques such as:

- Acoustic Doppler Anemometry (ADV)
- Particle Image Velocimetry (PIV)
- Laser Doppler Anemometry (LDA)

Recent validations for the RSM turbulence model against ADV and PIV measurements have been performed for a small scale stormwater tank by Dufresne *et al.* (2007). Instantaneous velocities from ADV were used to calculate values of turbulent kinetic energy, which corresponded closely with the simulation outputs. General trends in mean velocity, recorded using PIV, were also found to match the simulation, though there were some discrepancies between the actual values. This was considered to be due to errors in the surface area of cross correlation introduced in the averaging process of PIV.

Often, it is only possible to conduct validations using secondary modelling outputs, for example, the particle removal efficiency of a treatment system, or the RTD of a chamber. With regard to studies concerning particulate behaviour, Faram and Harwood (2002; 2003) present a strong validation for the Lagrangian particle tracking approach for the assessment of sediment interceptors. Pathapati and Sansalone (2007) also produced positive validations for this method, in which the numerical predictions were found to lie within 10% of the experimental results over a range of operating flowrates. Burt *et al.* (2002) conducted validations on results produced for a CSO chamber using Eulerian and Lagrangian models, concluding that the Eulerian method was preferable in this case. Adamsson *et al.* (2003) performed validations for the bed shear stress method, supporting its ability to predict overall system efficiency and sediment distribution patterns in storage tanks.

Examples of the use of system efficiency, RTD, free surface profile and boundary pressure as validation references are found throughout the literature. For complex systems, several simple 'unit' validation cases are often performed relating to individual physical phenomena. These are then combined to validate the overall system. This concept is discussed further by Oberkampf and Trucano (2002). Examples of unit validations for particle settling under controlled conditions (corresponding to settling column analysis) are presented by Buxton *et al.* (2002) and Egarr *et al.* (2004), with the objective of assessing Lagrangian tracking methods for urban drainage system analysis.

CONCLUSIONS

CFD presents a number of opportunities in the analysis of urban drainage systems, as evidenced by the various published studies. The most extensive application of CFD to urban drainage system analysis has been in relation to CSO chambers, followed by storage and attenuation systems, stormwater interceptors and sewerage conveyance structures. For the various studies, the main focus of analytical interest has been in flowfield predictions, particulate behaviour, free surface effects and RTDs. In all of the studies reviewed, the flows were assumed to be turbulent, and either RSM or k-ε type models were used in the analysis. Flows were also most often assumed to be steady.

Three modelling techniques have been applied for the analysis of particulate behaviour, including bed shear stress analysis, Lagrangian particle tracking and Eulerian phase tracking. The Lagrangian and Eulerian methods tend to produce the most comprehensive outputs, with the latter method being most appropriate where particle-particle interactions are important. Fixed frictionless boundary approaches to representing free surfaces are most prevalent in the reviewed studies. VOF approaches can produce detailed predictions of surface profiles, but are more computationally expensive. RTD predictions can be useful and can be produced using particle tracking, second phase tracking and UDS local age methods. The latter method has been found to produce the most accurate predictions, though a higher level of user expertise is required in its setup compared to the other methods.

Though CFD can be considered a mature CAE tool, modelling procedures relating to specific application areas are continually developing. In the broader context, CFD is becoming increasingly utilised as part of larger simulation schemes for multi-physical systems; this type of simulation is yet to be seen in urban drainage system analysis. Additionally, as computational resources advance and become more accessible, and with the recent trend towards parallelisation of computer systems, the scope to run faster and multiple batches of simultaneous simulations is becoming more common. In many industries CFD modelling processes have been automated to allow the scope for optimisation of a number of design or solver parameters to improve model accuracy, operating conditions or system design.

ACKNOWLEDGMENTS

The first author would like to acknowledge support provided through a Knowledge Transfer Partnership between the University of Exeter and Hydro International, which has the objective of developing and implementing advanced numerical techniques into the product development process.

REFERENCES

Ackers, J. C., Butler, D. and May, R. W. (1996). Design of sewers to control sediment problems. London, Construction industry research and information association (CIRIA).

Adamsson, A., Stovin, V. and Bergdahl, L. (2003). Bed shear stress boundary condition for storage tank sedimentation. *Journal of Environmental Engineering* 129(7): 7.

Alkhaddar, R. M., Higgins, P. R., Phipps, D. and Andoh, R. Y. G. (1999). The residence time distribution of a hydrodynamic separator operating with a baseflow component. 8th Int. Conf. on Urban Storm Drainage. Sydney, Australia.

ANSYS (2007). Fluent User's Guide. Version 6.3. Fluent Inc.

Bagchi, P. and Balachandar, S. (2003). Effect of turbulence on the drag and lift of a particle. *Physics of Fluids* 15(11): 3496-3513.

Bardiaux, J. B., Vazquez, J., Mose, R., Bonakdari, H. and Larrate, F. (2007). Assessment of velocity fields through open channels: creation of empirical laws. Novatech: 6th Int. Conf. Lyon, France.

Burt, D. and Balmforth, D. (2005). Modelling of retrofits to CSO chambers incorporating screens. 10th Int. Conf. on Urban Drainage. Copenhagen, Denmark.

Burt, D., Corton, M., Hetherington, D. and Balmforth, D. (2002). Multiphase modeling and the prediction of retention efficiency in a side weir CSO. 9th Int. Conf. on Urban Storm Drainage. Portland, OR, United States. American Society of Civil Engineers.

Buxton, A. P., Tait, S., Stovin, V. and Saul, A. (2002). Developments in a methodology for the design of engineered invert traps in combined sewer systems. *Water Science and Technology* 45(7): 133-142.

Dufresne, M., Vazquez, J., Terfous, A., Ghenaim, A. and Poulet, J. B. (2007). Three-dimensional flow measurements and CFD modelling in a storm-water tank. Novatech: 6th Int. Conf. Lyon, France.

Egarr, D., Faram, M. G., O'Doherty, T., Phipps, D. and Syred, N. (2005a). Computational fluid dynamic prediction of the residence time of a vortex separator applied to disinfection. *Water Science and Technology* 52(3): 29-36.

Egarr, D., Faram, M. G., O'Doherty, T. and Syred, N. (2004). An investigation into the factors that determine the efficiency of a hydrodynamic vortex separator. Novatech: 5th Int. Conf. Lyon, France.

Egarr, D. A., Faram, M. G., O'Doherty, T., Phipps, D. A. and Syred, N. (2005b). Computational fluid dynamic prediction of the residence time distribution of a prototype hydrodynamic vortex separator operating with a base flow component. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering* 219(1): 53-66.

Faram, M. G. and Andoh, R. Y. G. (1999). Evaluation and optimisation of a novel self-cleansing combined sewer overflow screening system using computational fluid dynamics. 8th Int. Conf. on Urban Storm Drainage. Sydney, Australia.

Faram, M. G., Guymer, I. and Saul, A. J. (2004). Assessment of modular block stormwater storage systems. Novatech: 5th Int. Conf. Lyon, France.

Faram, M. G. and Harwood, R. (2000). CFD for the water industry; the role of CFD as a tool for the development of wastewater treatment systems. Fluent Users' Seminar. Sheffield, UK, Fluent.

Faram, M. G. and Harwood, R. (2002). Assessment of the effectiveness of stormwater treatment chambers using computational fluid dynamics. 9th Int. Conf. on Urban Drainage. Portland, OR, United States. American Society of Civil Engineers.

Faram, M. G. and Harwood, R. (2003). A method for the numerical assessment of sediment interceptors. *Water Science and Technology* 47(4): 167-174.

Gupta, K., Mehta, P., Thinglas, T., Tait, S. J. and Stovin, V. (2005). Optimization of sediment trap configuration using CFD modelling for Indian drainage systems. 10th Int. Conf. on Urban Drainage. Copenhagen, Denmark.

Guymer, I., Shepherd, W. J., Dearing, M., Dutton, R. and Saul, A. J. (2002). Solute retention in storage tanks. 9th Int. Conf. on Urban Drainage. Portland, OR, United States.

Harwood, R. (2006). Computational flow modeling applications expand into the water industry. *Water and Wastewater International* 21(6): 32-34.

Harwood, R. and Saul, A. J. (1996). CFD and novel technology in combined sewer overflow. 7th Int. Conf. on Urban Storm Drainage. Hannover, Germany.

Kouyi, G. L., Vazquez, J., Simonin, Y., Zug, M. and Poulet, J. B. (2005). Use of 3D modeling to instrument CSOs. 10th Int. Conf. on Urban Drainage. Copenhagen, Denmark.

Lau, S. D., Stovin, V. R. and Guymer, I. (2007). The prediction of solute transport in surcharged manholes using CFD. *Water Science and Technology* 55(4): 57-64.

Lim, M. T. N., Lam, S. W., Amal, R., Cathers, B. and Pinson, D. (2002). Computational and experimental studies of floc behaviour in a vortex separator. 9th Int. Conf. on Urban Drainage. Portland, OR, United States.

Oberkampf, W. L. and Trucano, T. G. (2002). Verification and validation in computational fluid dynamics. *Progress in Aerospace Sciences* 38(3): 209-272.

Okamoto, Y., Kunugi, M. and Tsuchiya, H. (2002). Numerical simulation of the performance of a hydrodynamic separator. 9th Int. Conf. on Urban Drainage. Portland, OR, United States.

Pathapati, S. and Sansalone, J. (2007). BMPs exposed: The use of computational fluid dynamics (CFD) to examine BMP performance. StormCon: 6th Annual North American Surface Water Quality Conf. and Exposition. Phoenix, AZ, United States.

Pollert, J. and Stransky, D. (2002). Combination of computational techniques — Evaluation of CSO efficiency for suspended solids separation. 9th Int. Conf. on Urban Drainage. Portland, OR, United States.

Roos, A. (1999). On the effectiveness of ventilation. Eindhoven, Eindhoven University of Technology, The Netherlands. Ph.D. Thesis.

Saul, A. J. and Svejkovsky, K. (1994). Computational modelling of a vortex CSO structure. *Water Science and Technology* 30(1): 9.

Schmitt, F., Milisic, V. and Chebbo, G. (2002). Storage settling basins design and management. Sewer Processes and Networks. France.

Stovin, V., Schellart, A. N. A., Tait, S. J., Ashley, R. M. and Burkhard, R. (2005). Sewer invert trap design using laboratory and CFD models and continuous simulation. 10th Int. Conf. on Urban Drainage. Copenhagen, Denmark.

Stovin, V. R., Grimm, J. P., Buxton, A. P. and Tait, S. J. (2002a). Parametric studies on CFD models of sewerage structures. 9th Int. Conf. on Urban Drainage. Portland, OR, United States. American Society of Civil Engineers.

Stovin, V. R., Grimm, J. P. and Saul, A. J. (2002b). Fine sediment retention in storage chambers: an assessment of time-dependent effects. *Water Science and Technology* 45(7): 123-131.

Stovin, V. R. and Saul, A. J. (1996). Efficiency prediction for storage chambers using computational fluid dynamics. *Water Science and Technology* 33(9): 163-170.

Stovin, V. R. and Saul, A. J. (1998). Computational fluid dynamics (CFD) particle tracking approach to efficiency prediction. *Water Science and Technology* 37(1): 285-293.

Stovin, V. R. and Saul, A. J. (2000). Computational fluid dynamics and the design of sewage storage chambers. *Journal of the Chartered Institution of Water and Environment Management* 14(2): 103-110.

Ta, C. T. (1999). Current CFD tool for water and waste water treatment processes. *American Society of Mechanical Engineers, Pressure Vessels and Piping Division* 396: 79-85.

Ta, C. T. and Brignal, W. J. (1998). Application of computational fluid dynamics technique to storage reservoir studies. *Water Science and Technology* 37(2): 219-226.

Tyack, J. N. and Fenner, R. A. (1998). Identification of flow regimes within a hydrodynamic separator. Novatech: 3rd Int. Conf. Lyon, France.

Tyack, J. N. and Fenner, R. A. (1999). Computational fluid dynamics modelling of velocity profiles within a hydrodynamic separator. *Water Science and Technology* 39(9): 169-176.

Wertel, J., Vazquez, J. and Mose, R. (2007). Model for the numerical simulation of 3D turbulent flow to improve sewer net instrumentation. Novatech: 6th Int. Conf. Lyon, France.