The use of CFD coupled with physical testing to develop a new range of vortex flow controls with attributes approaching the ideal flow control device

D. S. Jarman[1], G. Tabor[2], D. Butler[3] and R. Y. G. Andoh[4]

[1]Hydro International, Shearwater House, Victoria Road, Clevedon, BS21 7RD, UK; PH +44(0)1275 878371; email: daniel.jarman@hydro-international.co.uk
[2]Centre for Water Systems, College of Engineering, Mathematics and Physical Sciences, University of Exeter, North Park Road, Exeter, EX4 4QF, UK; email: g.r.tabor@exeter.ac.uk
[3]Centre for Water Systems, College of Engineering, Mathematics and Physical Sciences, University of Exeter, North Park Road, Exeter, EX4 4QF, UK; email: D.Butler@exeter.ac.uk
[4]Hydro International, 94 Hutchins Drive, Portland, Maine, 04102, USA; email: bandoh@hil-tech.com

ABSTRACT

Vortex flow controls (VFC) are devices which are well suited for use in drainage systems, as they exhibit non-constant, non-linear discharge coefficients that can be tailored to approach that of a constant flow-rate device. Also, they have no mechanical components or power requirements and have a reduced risk of blockage compared with traditional flow controls. However, due to their complex bi-stable discharge behaviour and the influences of turbulence, the design and scaling of these devices, is not a trivial process. In this paper a VFC design methodology is presented that enables the VFC geometry to be determined and optimized to approach the ideal hydraulic behaviour, for a given discharge limit. This is achieved through the calibration of simplified, axi-symmetric vortex solutions of the Navier-Stokes relationships, by means of Computational Fluid Dynamic (CFD) analysis and experimental hydraulic assessment.

INTRODUCTION

Historically, the objective of drainage system design has primarily focused on carrying water away from the local populace for immediate disposal or processing. This reduces the time of concentration of the catchment and hence time to peak flows. Impermeable surfaces caused by urbanisation and intensive land usage increase the overall runoff volume and peak flow for a given rainfall event. This is attributed to the resulting increased depth of flow and reduced surface roughness. The consequence of traditional design approaches coupled with increasing urbanisation exacerbate the detrimental effects of flooding, erosion and pollution on the local population and environment.

Modern design methodologies aim to improve over the traditional design approaches by emphasising the control and treatment of water near its source. Regardless of the terminology this objective is evident in most forward-looking, environmentally-considerate design philosophies; i.e. Best Management Practices (BMP), Low Impact Development (LID), Water Sensitive Urban Design (WSUD) and Sustainable Drainage Systems (SuDS). Although these philosophies are normally associated with pollutant control, several studies have shown that the application of distributed flow attenuation, in the form of detention or infiltration storage mechanisms, such as ponds or tanks with an associated flow control, increase network robustness and present cost benefits. This approach maximises infiltration across the catchment, reducing the total flow volume, and increases the time of concentration, mimicking the natural undeveloped catchment's response. This is demonstrated in Figure 1. Andoh and Declerck (1999) quantify this effect and demonstrate that source control and distributed storage arrangements deliver 25-80% cost savings compared to traditional end-of-pipe solutions, for similar hydraulic design criteria. Stovin and Swan (2003, 2007) also show that the retrofitting of SuDS elements into existing infrastructure, for an existing UK catchment, delivers potential cost savings of 12%, compared to traditional reactive measures. At a practical scale, there are now a number of examples, particularly in the USA, where source control approaches have been applied extensively as a means of reducing CSO spills and flooding. Well documented examples include that of Skokie, near Chicago (Carr and Walesh, 2008) and Evanston, Illinois (Barber et al., 1994 and Figurelli et al., 2000).

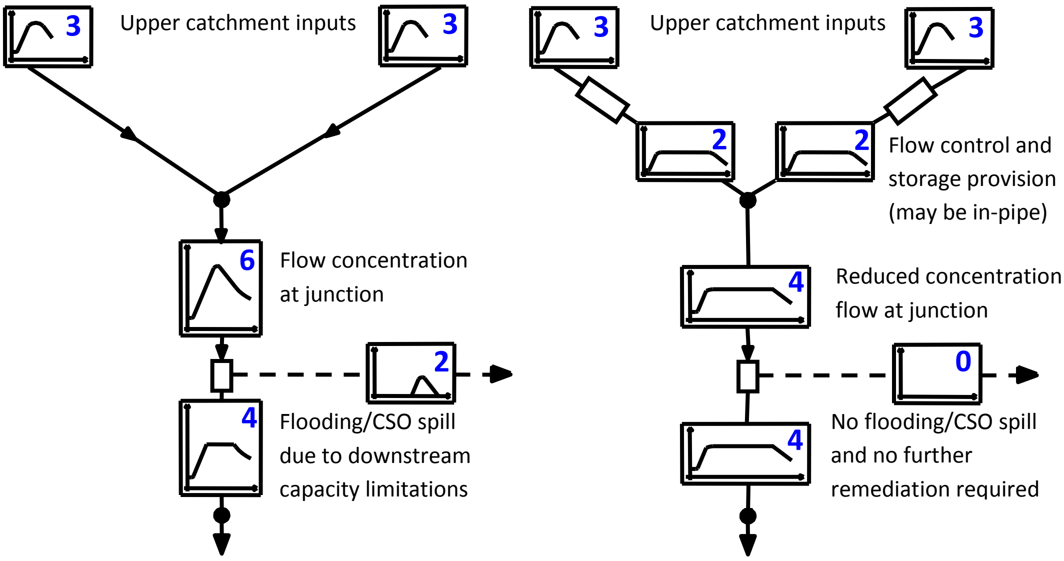


Figure 1: Illustration of hydrographs across a catchment with and without flow attenuation mechanisms (adapted from Andoh and Declerck (1999)).

General acknowledgment of these benefits has lead to the development of guidelines and legislation encouraging the use of these mechanisms. An example of one of the most recent implemented legislative drivers is the UK Floods and Water Management Act, 2010. This removes the automatic right to connect to the main sewer network and specifies drainage design philosophies based on risk management.

FLOW CONTROL REQUIREMENTS

Optimisation of the hydraulic behaviour of flow controls has the potential to reduce capital, installation and maintenance costs of detention systems, and also minimise the likelihood of flooding. This is especially true for modern design approaches, as a result of the multiplicity of the distributed storage and flow control elements in the network. In order to mimic the hydraulic conditions, prior to catchment development, the discharge criteria must be specified as the pre-development average annual runoff at a given point in the catchment. This ensures the risk of downstream flooding is limited to the pre-developed probability.

The ideal hydraulic condition can be considered as a constant discharge device, regardless of upstream head, as this maintains discharge rates at low heads minimizing premature filling (utilisation) of detention volumes during non-critical rainfall events. This either reduces detention volume requirements, or the frequency of flood or CSO spill events for a given detention volume. In comparison, orifice flow controls tend to over-restrict flows at low heads (as the flow-rate is proportional to the square-root of the diving head) meaning the hydraulic capacity of the downstream network is only fully utilised during the critical rainfall event. Orifice flow controls, therefore, prematurely fill the upstream attenuation storage and under utilise the in-pipe storage available throughout the drainage network. Although constant discharge behaviour is achievable with devices such as Real-Time Controlled (RTC) penstocks, this is not practical for distributed systems due to the power and maintenance requirements, and the small clearances imposed on the flow.

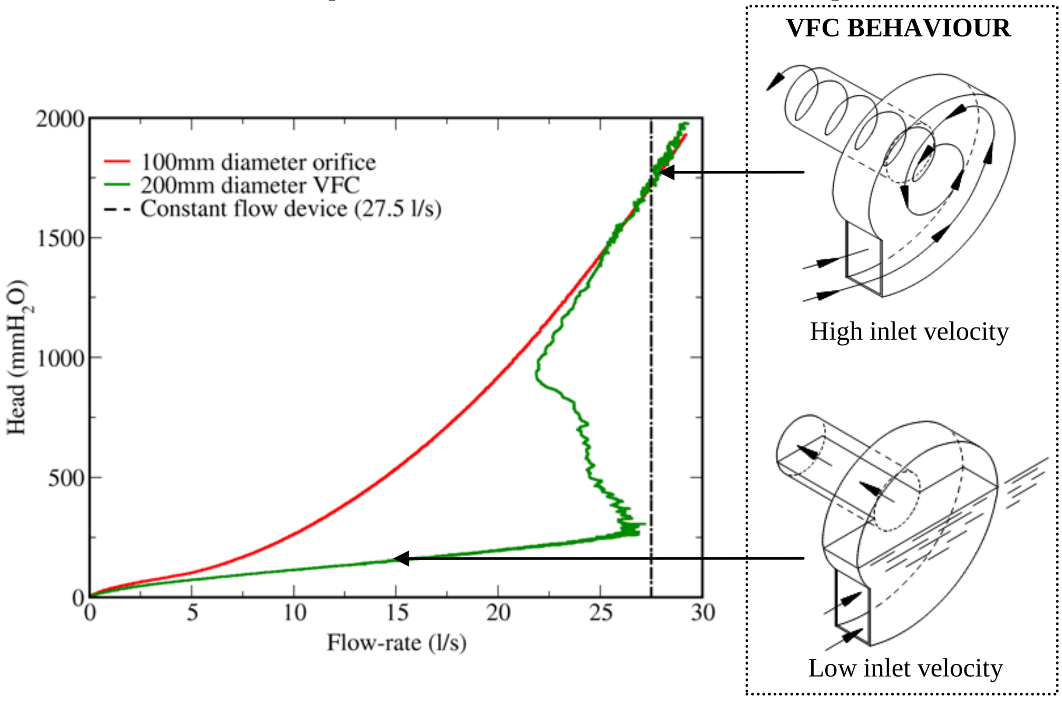


Figure 2: Comparison of hydraulic behaviour for a 100mm diameter orifice, a VFC with a 200mm diameter outlet approaching the constant discharge condition and a constant discharge device.

Vortex Flow Controls (VFCs) are bi-stable fluidic throttles that induce a swirling flow pattern through their chamber geometry, producing additional inertial and turbulent losses compared to devices that only impose a physical restriction. At low flow-rates the entrance velocity is not sufficient to sustain a swirling motion and normal (open-channel) orifice flow patterns prevail. With increasing flow velocity a swirling motion is developed. This allows these devices to be self-activating, with no mechanical components or power requirements; throttling the flow without imposing small clearances. A typical VFC geometry and its resulting hydraulic behaviour is illustrated in Figure 2. This plot shows the experimentally measured hydraulic discharge characteristic for a VFC with a 200mm minimum internal clearance plotted alongside a 100mm diameter orifice fabricated according to BSI (2003), which produces an equivalent discharge at 1750mm of head.

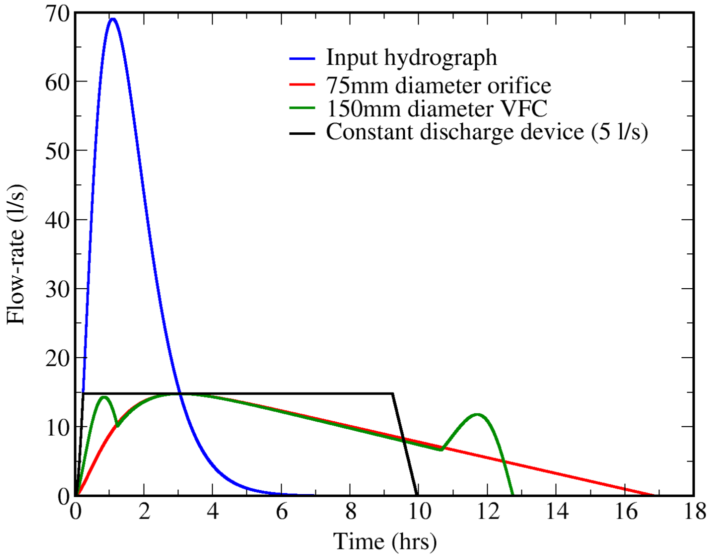


Figure 3: Comparison of hydrograph responses to a critical storm input for a 75mm diameter orifice, a VFC with a 150mm diameter outlet approaching the constant discharge condition and a constant discharge device (adapted from LeCornu et al. (2008)).

Although VFCs cannot fully satisfy constant discharge behaviour, their geometries can be manipulated to produce hydraulic characteristic approaching this condition, offering a suitable compromise between the draw-backs of powered, mechanical flow controls and benefits in hydraulic capacity and storage utilization. This essentially involves maximising the discharge behaviour throughout both of the bi-stable phases of VFC operation, without exceeding the design flow-rate limit. This concept is more clearly demonstrated by the hydrographs shown in Figure 3, which show a VFC hydraulic characteristic that approaches the constant discharge behaviour as studied by LeCornu et al. (2008). LeCornu et al. (2008) consider a series of input hydrographs and hydraulic design criteria for a 75mm orifice compared with a 150mm VFC and found that for the critical event the storage savings, compared to the orifice's requirements, for a constant discharge device and a VFC were 13% and 7% respectively. The detention volume recovery period was also found to be significantly reduced due to the increased drain rates, which increases network robustness in periods of continual rainfall. Simulations based on empirical assessments and physical installations have shown that storage savings of over 25% are attainable (Parsian and Butler, 1993; LeCornu et al., 2008; Faram et al., 2010). Figures 1 and 2 demonstrate that VFCs also typically present twice the minimum physical flow path restriction, compared to an orifice, making them resistant to blocking. For further details regarding the operation and benefits of VFCs the reader is referred to Faram et al. (2010) and Andoh et al. (2009).

DESIGN OF VORTEX FLOW CONTROLS

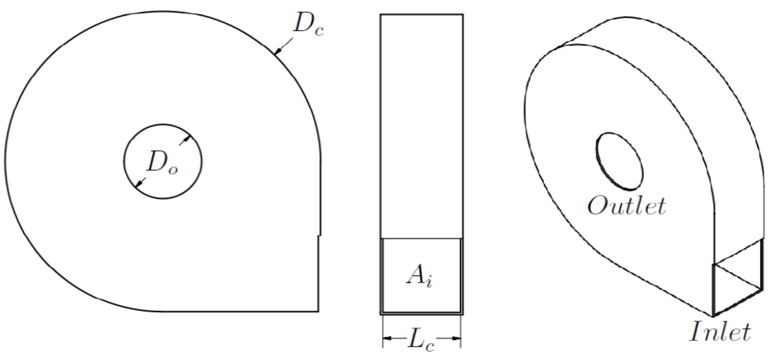
The application of VFCs is becoming increasingly common, with the combined number of installations by Hydro International, throughout the UK and US, being over 20,000 units; with outlet diameters reaching over 2000mm. The bi-stable behaviour of VFCs, although beneficial, makes it difficult to guarantee optimal behaviour for all hydraulic specifications and critical rainfall scenarios. This is because the loss at high inlet velocities, where a confined vortexing flow field dominates, mainly results from the inertial and turbulent losses, which are influenced by a number of geometric and flow parameters. Therefore, accurately predicting the losses for the vortexing phase of operation is the main objective when determining the optimal VFC geometry. At low inlet velocities the losses are similar to that of an equivalent sized orifice and so are easily determined through well established empirical theory based on Torricelli's law:

$$U \propto \sqrt{2gh}$$

where U is the velocity, g is the gravitational acceleration and h is the head loss.

Historically, VFC design has relied on semi-empirical relationships developed through the assessment of a large number of physical prototypes. This approach is both costly and restricted by the capacities of hydraulic assessment facilities. The flowing sections of this paper describe Hydro International's revised design approach to develop a new range of VFCs, which guarantee behaviour approaching the constant discharge condition by varying the unit geometry.

The VFC geometry studied is shown in Figure 4. Varying the VFC swirl parameter (N) presents as an effective means of controlling the losses associated with the vortex behaviour. This allows the VFC geometry to be tailored to give a hydraulic response approaching the constant flow condition. The geometry shown in Figure 4 is typical of VFC devices applied to stormwater drainage applications. As the inlet of the VFC is positioned below the outlet, this type of VFC requires a sump and therefore is generally avoided for wastewater scenarios.



$$Re = \frac{4Q}{\nu \pi D_o}$$

$$N = \frac{\pi D_c D_o}{4A_i}$$

Figure 4: Typical stormwater VFC geometry and flow parameters.

AXISYMMETRIC FLOW MODELS

In order to achieve a greater level of flexibility and confidence in the design and scaling of VFCs, a theoretical derivation based on an axisymmetric solution to the Navier-Stokes equations was applied to determine device behaviour based on the geometric and flow parameters. This is a more robust approach than the existing design models based on semi-empirical relationships. The design methodology was validated through a combination of experimental assessments and Computational Fluid Dynamic (CFD) simulations.

There are a number of steady or quasi-steady models for axisymmetric swirling flows. At high Reynolds and swirl numbers (*Re* & *N*) swirling flows exhibit a stagnation point or recirculation zone at the vortex axis, resulting in a bathplug-type or two-celled vortex. The inner core-cell, or recirculation zone is occupied by air, where as the outer cell is water. Where the inner and outer cell streamlines converge the radial velocity is zero and the water surface occurs. This is the type of vortex behaviour that is observed in the operation of VFCs. Sullivan (1959) derived a two-celled axisymmetry solution to the Navier-Stokes equations that describes this type of behaviour, which is shown in Table 1, where α_v is a suction coefficient, *r* is the radius, υ is the kinematic viscosity (which includes turbulent viscosity), Γ_∞ is the circulation in the far field, *p* is the pressure, and ρ is density.

Table 1: Sullivan's (1959) axisymmetric solution to the Navier-Stokes equations.

Component	Formula
Radial velocity	$U_r = -\alpha_v r + \frac{6\nu}{r}\left[1 - e^{\left(-\frac{\alpha_v r^2}{2\nu}\right)}\right]$
Azimuthal velocity	$U_\theta = \frac{\Gamma_\infty}{2\pi r}\left[\frac{H\left(\frac{\alpha_v r^2}{2\nu}\right)}{H(\infty)}\right]$ where $H(x) = \int_0^x e^{\left[-t+3\int_0^t \left(\frac{1-e^{-s}}{s}\right)ds\right]} dt$
Axial velocity	$U_z = 2\alpha_v z\left[1 - 3e^{\left(-\frac{\alpha_v r^2}{2\nu}\right)}\right]$
Pressure	$\Delta p = \rho\int_0^r \frac{U_\theta^2}{r} dr - \frac{\rho}{2}\left\{4\alpha_v^2 z^2 + \alpha_v^2 r^2 + \frac{36\nu^2}{r^2}\left[1 - e^{\left(-\frac{\alpha_v r^2}{2\nu}\right)}\right]^2\right\}$

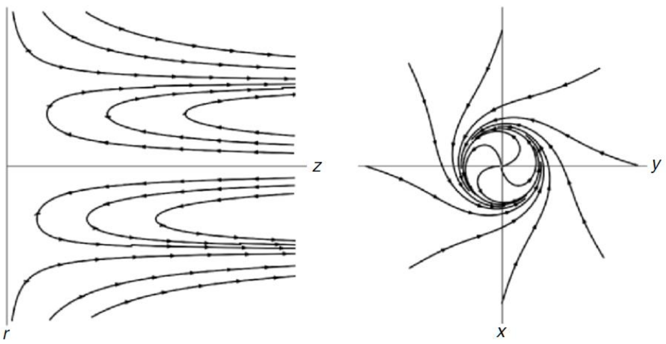


Figure 5: Characteristic velocity streamlines according Sullivan's (1959) axisymmetric solution to the Navier-Stokes equations.

Applying the correct boundary conditions to this model allows predictions for the head loss for a particular VFC geometry at a given flow-rate to be made. Factors considered to influence the boundary conditions were the inlet velocity profile and circumferential boundary layer thickness. An example of the characteristic velocity streamlines according to Sullivan's model are shown in Figure 5.

EXPERIMENTAL ASSESSMENT OF VORTEX FLOW CONTROLS

The hydraulic characteristics of a range of physical VFC prototypes were measured in order to validate the CFD modelling methodology. The units were assessed according to the quasi-steady assessment method described by LeCornu and Faram (2006) at Hydro International's UK hydraulic laboratory. An uncertainty survey of the equipment accuracy, assessment methodology and fabrication tolerances indicated the overall assessment 95% confidence interval to be ±1.36% of the measured flow-rate. The hydraulic facility was capable of assessing flow-rates up to approximately 50 l/s. This allowed the assessment of VFCs with outlet diameters of up to 200mm with swirl parameters of between 1.57 and 4.71, which corresponds to a maximum Reynolds number (*Re*) of 225,000. A total of 11 geometries were assessed experimentally, which covered the entire hydraulic range of the test facility.

CFD SIMUALTIONS OF VORTEX FLOW CONTROLS

Appropriate mathematical models must be selected in order to accurately simulate the flow behaviour. The recirculation zone at the axis of the vortex in VFCs results in multiphase behaviour as air is drawn in at the outlet and the air-core is formed. Therefore, a multiphase Volume-of-Fluid (VOF) formulation was applied, which considers the combined flow field of water and air behaving as a single mixture. The mixture was considered to be incompressible and behave in a Newtonian manner. Turbulent stresses where included as a Reynolds decomposed, ensemble average quantity. The governing physical relationships for this situation are given by the following equations, which represent the conservation of the fluid mass, momentum and phase constituents (air and water) respectively:

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{U}_m) = S_\rho$$

$$\rho_m \left(\frac{\partial \mathbf{U}_m}{\partial t} + \mathbf{U}_m \cdot \nabla \mathbf{U}_m\right) = -\nabla p + \nabla \cdot (\mu_m \nabla \mathbf{U}_m) + \rho_m \mathbf{g} \cdot \mathbf{x} + \sigma_{w,a} - \nabla \cdot (\rho \overline{\mathbf{u'}_m \mathbf{u'}_m})$$

$$\frac{\partial \alpha_w}{\partial t} + \nabla \cdot (\alpha_w \mathbf{U}_m) + \nabla \cdot [\alpha_w \alpha_a \mathbf{U}_r] = S_\alpha, \quad \text{where } \alpha_w + \alpha_a = 1$$

where S represents any sources or sinks, μ is the dynamic viscosity, $\mathbf{x}$ is a datum vector, $\sigma_{w,a}$ is the surface tension force according to Brackbill et al. (1992), and the turbulent stresses ($\rho\overline{\boldsymbol{u}'\boldsymbol{u}'}$) are formulated according to Launder, Reece and Rodi (1975). The formulation of the transport of the phase fraction quantity (α) was originally proposed by OpenCFD Ltd, where $\mathbf{U}_r$ represents the velocity between the phases and is manipulated to reconstruct the water surface to provide a sharp immiscible-like interface. For more information regarding this interface reconstruction method the reader is referred to Berberovic et al. (2009). The subscripts *m*, *w* and *a* indicate quantities related to the mixture, water or air respectively. The properties of the mixture are established according to the following relationships:

$$\rho_m = \alpha_w \rho_w + \alpha_a \rho_a$$

$$\mu_m = \alpha_w \mu_w + \alpha_a \mu_a$$

The governing equations were descretised and solved via a second-order accurate finite volume approach. This was performed with the open source continuum mechanics code OpenFOAM®. The modelled geometries mimicked the mounting arrangement of the experimental assessment facility, where possible, and were descretised in an unstructured hexahedral fashion, as shown in Figure 6.

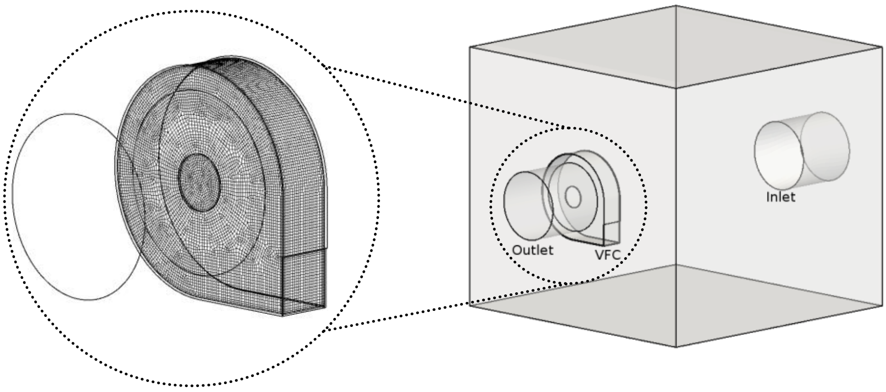


Figure 6: Simulated geometry and spatial discretisation example for a VFC with a 100mm diameter outlet and a swirl parameter of 3.14.

A total of 48 CFD simulations were performed, which included VFCs with outlet diameters up to 500mm with swirl parameters of 6.28. This corresponds to a maximum simulated Reynolds number of 750,000, and represents a flow-rate range of up to 250 l/s, which is more that 5 times the capacity of the hydraulic assessment facility.

RESULTS

The measured and simulated hydraulic behaviour of VFCs with outlet diameters of 50mm, 100mm, 150mm and 200mm and swirl parameters of 3.14 is shown in Figure 7. The flow-rates simulated only correspond to the quasi-stable swirling behaviour, which occurs at sufficiently high inlet velocities. This demonstrates that for a swirl parameter value of 3.14 the correlation of the experimental and simulated results are within the experimental uncertainty range of ±1.36% of the measured flow-rate. Correlations of a similar accuracy were obtained for the swirl parameter range 1.57 to 3.93. Above this range the CFD simulations over-predicted the flow-rate by up to 9%. This is most likely due to shortcomings in the turbulence modelling approach. However, VFCs with swirl parameters above 4.71 were not found to be beneficial as the swirling flow pattern at the chamber circumference was dissipated and linear flow patterns in the outer region of the chamber occurred, indicating the transition from confined to unconfined swirling flow behaviour.

Figures 7(a) and 7(b) show the simulated contours of velocity gradient and pressure plotted across the axial mid-section and vertical cross-section of a VFC with an outlet diameter of 100mm and a swirl parameter of 3.14 operating at 7l/s. These figures clearly demonstrate a stagnation zone at the vortex axis. They also confirm that the flow-field is relatively axisymmetric about the vortex axis, albeit subtlety displaced from the chamber axis, verifying that the application of axisymmetric vortex models is suitable for predicting the behaviour of VFCs.

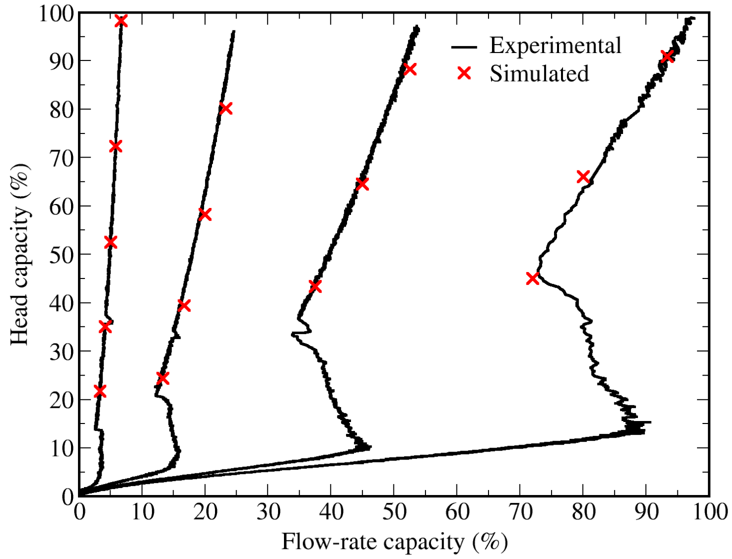


Figure 7: Comparison between measured and simulated hydraulic behaviour for VFCs with swirl parameters of 3.14 over the hydraulic capacity of the experimental facility.

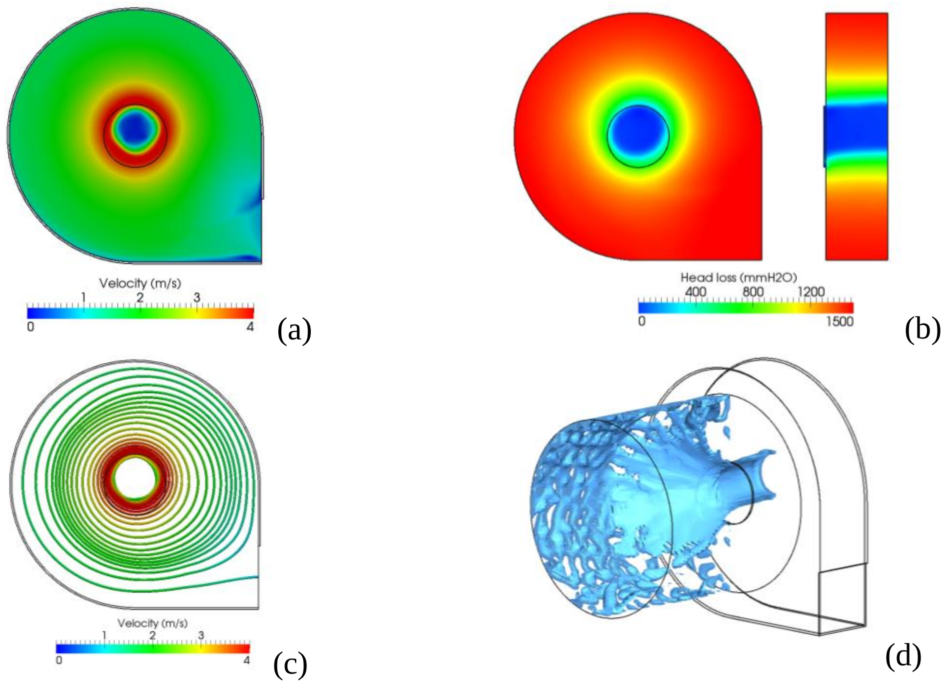


Figure 8: Velocity contours (a), pressure contours (b), velocity streamlines (c), and 0.5 phase isosurface (d) for a VFC with a 100mm diameter outlet and a swirl parameter of 3.14 operating at 7 l/s.

Figure 7(c) shows an infinitely long velocity streamline released from the centroid of the inlet face for a VFC with an outlet diameter of 100mm and a swirl parameter of 3.14 operating at 7l/s, coloured by velocity magnitude. Figure 7(d) shows the 0.5 phase fraction isosurface, which represents the location of the water surface for this configuration. These two figures show how the velocity streamline of the outer vortex-cell converges upon itself at the water surface, indicating that the flow pattern is consistent with those predicted by Sullivan (1959) and that this model is suitable for predicting the behaviour of VFCs.

Figures 8(a) and 8(b) show the normalised velocity profiles predicted via the VFC design equation based on Sullivan's model compared to the simulated values for VFCs with swirl parameters of 3.14. These non-dimension plots include the flow-rate range from 0-250l/s. This shows that although there is a small amount of asymmetry in the vortex behaviour the overall pressure loss across the vortex was found to correlate within ±5% of the simulated values for VFCs with swirl parameters below 3.93. Above this range Sullivan's model was found to increasingly over-predict the head loss. To overcome this shortcoming the effective viscosity of the fluid (see Table 1) was corrected empirically to achieve the correct velocity distribution. This was only required between swirl parameters of 3.93 and 4.71, as VFC geometries above this range do not present any further headloss or storage utilisation benefits.

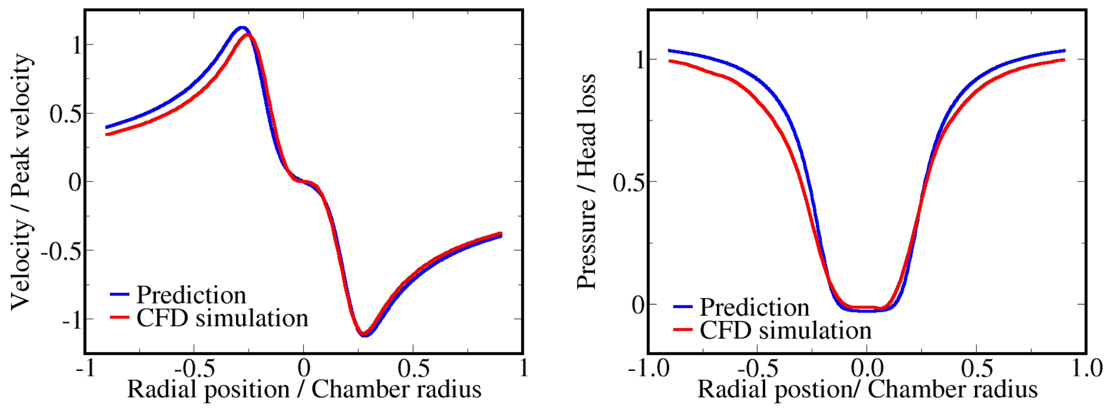


Figure 9: Normailsed Velocity (a) and pressure (b) distribution across VFCs with swirl parameters of 3.14 operating at up to 250 l/s.

CONCLUSIONS

The CFD modelling approach and VFC design model based on Sullivan's axisymmetric solution to the Navier-Stokes equations were found to give satisfactory predictions for VFC geometries with swirl parameters of less that 3.93. The correlation of the design equation in this range was found to be within ±5% of headloss for the experimental measurements and simulated values. Above this swirl parameter range the CFD models and design model predictions appeared to be sensitive to the turbulent viscosity as the flow transitioned from a confined swirling flow to an unconfined swirling flow. Between a swirl parameter range of 3.93 and 4.71 it was necessary to alter the effective viscosity empirically to improve the predictions of the VFC design model. This design approach is now being applied to ensure the VFCs supplied by Hydro International approach a constant discharge conditions and maximise in-system storage utilisation for all hydraulic specifications.

REFERENCES

Andoh, R. and Declerck, C. (1999). Source control and distributed storage - a cost effective approach to urban drainage for the new millenium? In 8th International Conference on Urban Drainage, pages 1997–2005, Sydney,Australia.

Andoh, R. Y. G., Faram, M. G. and Osei, K. (2010). Vortex Flow Controls in Integrated Stormwater

Management For Urban Environments. The 6th international water sensitive urban design conference and hydropolis 2010, Perth, Western Australia, 5–8 May.

Barber, D., Stoneback, D., Velon, J., and Persaud, R., (1994). "Evanston's (Ill.) Unique Solution to Large Combined Sewer Problems", Water Environment Federation 67th Annual Conference and Exposition.

Berberovic, E., van Hinsberg, N., Jakirlic, S., Roisman, I., and Tropea, C. (2009). Drop impact onto a liquid layer of finite thickness: Dynamics of the cavity evolution. Physical Review E, 79(3).

Brackbill, J., Kothe, D., and Zemnach, C. (1992). A continuum method for modelling surface tension. Journal of Computational Physics, 100:335–354.

BSI (2003). Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full - part 2: Orifice plates. Technical Report BS EN ISO 5167-2:2003, British Standards Institution, Milton Keynes.

Carr, R.W. and Walesh, S.G. (2008). *Micromanagement of stormwater in a combined sewer community for wet weather control – the Skokie experience*. 11th ICUD, Edinburgh, September, CD-ROM.

Faram, M. G., Osei, K., and Andoh, R. Y. G. (2010). Vortex Valves – Effective Flow Regulation for Economical Stormwater Management. World Environmental & Water Resources Congress 2010, Providence, Rhode Island, May 16-20.

Figurelli, R., Stoneback, D., Ramon, G., Velon, J. P., and Shadrake P. (2000). A approach to combined sewered areas: keeping basements dry at half the cost. WEF Specialty Conference 2000: Collection Systems Wet Weather Pollution Control.

Launder, B., Reece, G., and Rodi, W. (1975). Progress in the development of a Reynolds-stress turbulence closure. Journal of Fluid Mechanics, 68(3):537–566.

LeCornu, J. P. and Faram, M.G. (2006). Evolving Methods for the Calibration of Flow Controls for Stormwater and Wastewater Management. In: Emerging Environmental Issues and Future Challenges: 4th CIWEM Annual Conference, Newcastle upon Tyne, 12 – 14 September, 2006.

LeCornu, J.P., Faram, M.G., Jarman, D.S. and Andoh, R.Y.G. (2008). *Physical characterisation and hydrograph response modelling of vortex flow controls*. 11th ICUD, Edinburgh, UK, 31 August – 5 September.

Parsian, H. and Butler, D. (1993). Laboratory investigation into the performance of an in-sewer vortex flow regulator. Water and Environment Journal, 7(2):182–189.

Stovin, V. and Swan, A. (2003). Application of a retrofit SuDS decision-support framework to a UK catchment. In Proceedings of the 2nd National Conference on Sustainable Drainage, pages 171–180, Coventry, UK.

Stovin, V. and Swan, A. D. (2007). Retrofit suds - cost estimates and decision support tools. Proceedings of the Institution of Civil Engineers - Water Management, 160(WM4):207–214.

Sullivan, R. (1959). A two-cell vortex solution of the Navier-Stokes equations. Journal of Aerospace Science, 26:767–768.