CIWEM/AETT Millennium Conference; 'Wastewater Treatment: Standards and Technologies to meet the Challenges of the 21st Century'

Leeds, UK, 4th – 6th April 2000, pp 223-230

Application of Simulation and Predictive Techniques for the Evaluation of Hydrodynamic Separators

M. G. Faram and R. Y. G. Andoh
Hydro International plc, Shearwater House, Clevedon Hall Estate, Victoria Road, Clevedon, N. Somerset, BS21 7RD, UK
(e-mail: mike.faram@hrd.co.uk, bob.andoh@hrd.co.uk)

ABSTRACT

Computational fluid dynamics (CFD) simulation software has traditionally been most popular in hi-tech industries such as the aerospace and automobile industries, where the 'pay backs' can be very high. The uptake by the water industry has been somewhat slow however.

The paper provides an account of a study that was carried out in which CFD simulation was applied for the assessment of the efficiency of a hydrodynamic vortex separator intended for primary sedimentation applications. Qualitative comparisons of data with the predictions of a well validated semi-empirical model suggest that the software is effective for assessing the relative impacts of change.

As a focus for the study, the impact of a particular simplifying modification on the design of the system was assessed. Backed up by data obtained from small scale experimental testing, the modification proposed has been adopted, yielding direct savings in terms of fabrication costs for a particular project. The findings of the study have implications to the optimisation of other types of hydraulically based wastewater treatment systems, and indeed, hydraulic systems in general.

INTRODUCTION

The design of hydraulic processes or systems, more specifically, sedimentation systems, is generally based on knowledge that has been gained from either model or pilot scale experimental testing. The dependence of the performance of such systems on fluid dynamic phenomena dictates that they are inherently complex. Accurate prediction of the performance of scaled up systems is thus difficult, ideally requiring detailed characterisation data for units of different sizes, and the establishment of appropriate scaling laws, often in the form of semi-empirical models. In practice, this kind of activity can be both costly and time consuming, let alone difficult in many cases. Despite this, sizing procedures have developed for a number of systems. Some of these are very well developed, being based on extensive characterisation studies, whilst others are rooted simply on a combination of basic operational knowledge and assumptions.

A fundamental fallback of traditional hydraulic models and scaling methods is that they tend to be inflexible in terms of their ability to tolerate changes that fall beyond the boundaries of the data upon which they were originally based. Effectively, a full characterisation study of every design and condition of interest would be required in order to be able to develop a fully flexible and robust model. Such an undertaking would be impractical in the majority of cases.

Advances in the area of computer based fluid flow simulation have been considerable over the last decade. This has been largely due to the continual and rapid increase in computer system capabilities. Computational fluid dynamics (CFD) software has become more accessible, and has subsequently become widespread throughout various sectors of the engineering industry. Problems of increasing complexity can be simulated with increasing ease, with current CFD codes being able to model phenomena including chemical reactions, heat transfer and particle trajectories, as well as overall flow behaviour. Unlike traditional semi-empirical models, CFD software incorporates both flexibility and versatility, enabling both design changes and operating parameter changes to be taken into account during the course of a prediction. Although, currently, CFD codes are not sufficiently advanced to represent a complete replacement to traditional experimental techniques, they are very useful for providing information that could not otherwise be readily obtained. Where sufficiently validated, CFD is particularly useful for comparing the relative effects of change on a system, where qualitative predictions can be very good.

In the current work, the CFD software, Fluent, has been applied to the simulation of a 12 metre diameter hydrodynamic vortex separator (HDVS), intended for primary sedimentation wastewater treatment applications. The software has been used to predict the particle removal capabilities of the system, and also the impact of a particular design modification on operation. This data has subsequently been compared to that produced by a well validated semi-empirical model, also described in the paper.

HYDRODYNAMIC SEPARATORS FOR WASTEWATER TREATMENT

HDVS's are essentially high rate rotary sedimentation systems which, due to the nature of their operation, have a relatively small land area requirement. To give equivalent performance, a conventional rectangular primary sedimentation tank would typically need to occupy up to five times the area, having major implications to both installation and maintenance costs.

Figure 1 shows the main features of a well developed form of HDVS. By means of a tangentially located inlet, flow passing into the unit causes flow rotation within the

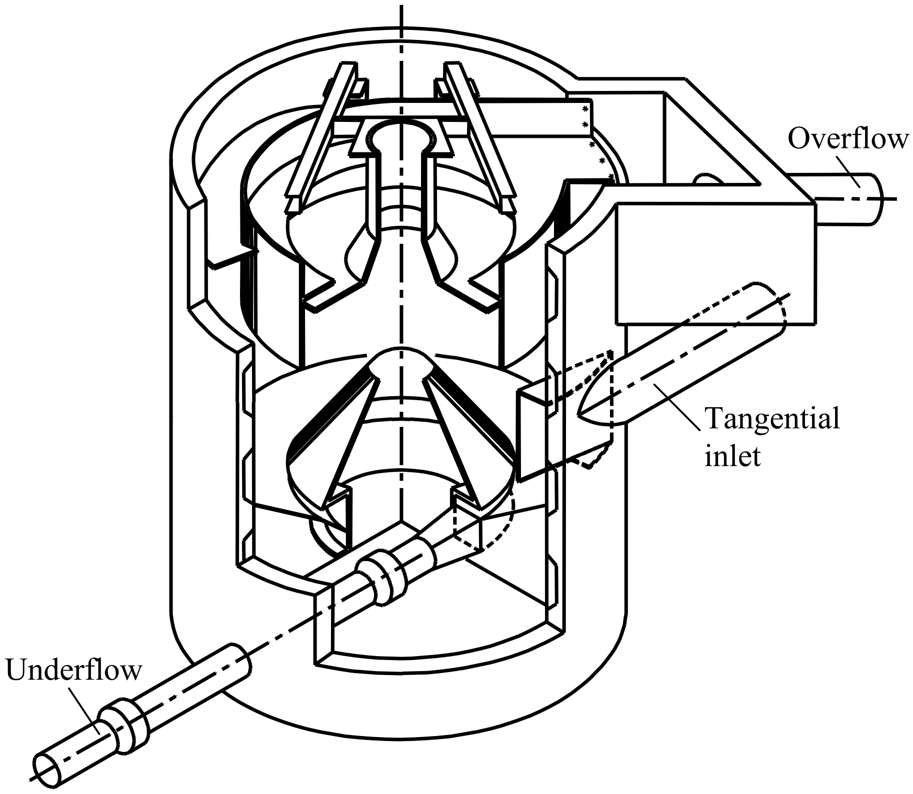

Figure 1 Sectional view through a technically superior HDVS system

vessel. Such systems have been shown to have a stable near-plug flow flow regime (1). This means that fluid elements pass through a very long, spiral flow path prior to exiting via the overflow. Since the path is long in terms of distance, compared to the physical dimensions of the system itself, it means that suspended particles have a relatively long period of time in which to settle. A secondary effect, caused by this type of flow structure, manifests as a sweeping action on the base, directing particles to a central location. A similar effect can be replicated by stirring a cylindrical vessel containing water and a small quantity of sand. The inclusion of baffles into the system, as shown, helps to both control and enhance the effects of these phenomena.

HDVS systems have been used for the removal of solids from liquid streams in a number of wastewater treatment related applications, including the removal of readily settlable material from combined sewer overflows (2), the removal of grits from sewage at treatment works (3), the removal of sediments and oils from stormwater run-off (4), and the removal of fine sediments from industrial and municipal effluent streams (5). When used in conjunction with chemical flocculants and coagulants, further enhancements in performance can be achieved due to the efficient contacting effect given by the flows. Whilst the principle of operation essentially remains the same, regardless of application, the difference between devices lies in how they are hydraulically loaded to remove a particular solids fraction. In the current work, a system with a relatively low loading rate has formed the focus. The system in question, with typical design surface loading rates of between 1 and 3 $l/s/m^2$ (based on flowrate/separator horizontal cross sectional area), has typically been applied for primary sedimentation applications.

SEMI-EMPIRICAL MODEL FOR PERFORMANCE PREDICTION

In the early 1990's, a semi-empirical mathematical model was developed, to enable predictions to be made for the performance of low loading rate HDVS's (6). Derived from first principles relating to sedimentation theory, this takes the form;

$$\frac{D^2 V_s}{Q} = K_a^{K_b} \sqrt{\frac{N}{R}\left(\frac{1-P}{P}\right)}$$

where

D = diameter of separator

V_s = settling velocity of particle in fluid

Q = flowrate

N = ratio between solids concentration in the underflow to that in the overflow

R = ratio of overflow to underflow

P = underflow proportion

K_a and K_b = empirical coefficients

The empirical coefficients, K_a and K_b were derived from the calibration of separator units large enough such that viscous and surface tension effects would not be significantly different in pilot scale units and full scale installations, and vary, depending on the configuration. This is in recognition of the impossibility of achieving complete similarity between model and prototype scale units, dictated by the conflicting operational requirements for Reynolds Number and Froude Number similarity.

Effectively, the model enables predictions to be made for separator removal efficiencies for particles with differing settling velocities. Figure 2 illustrates a

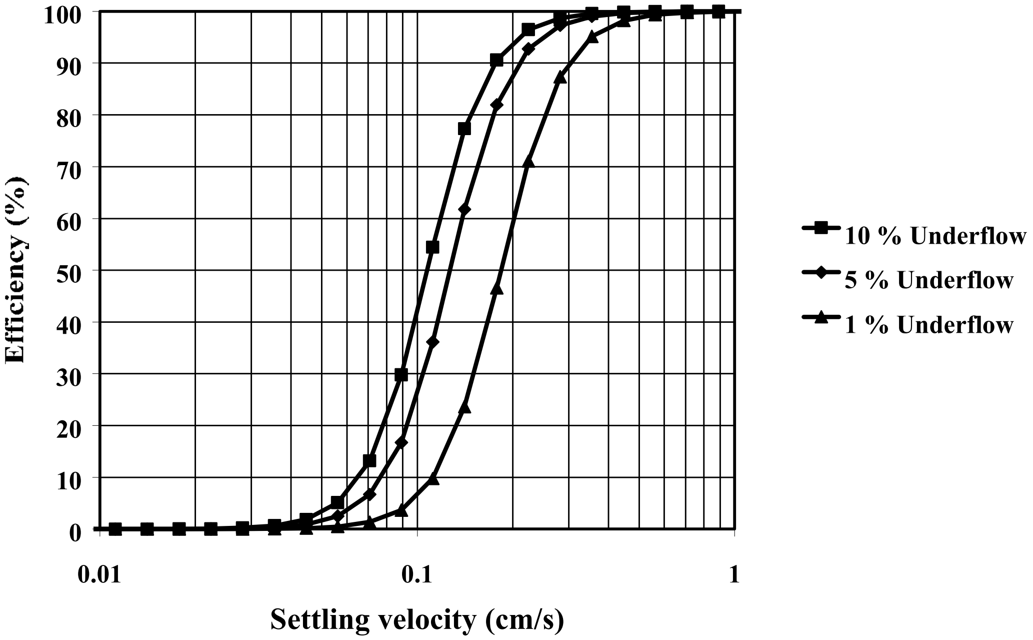

Figure 2 Semi-empirical model efficiency predictions for a surface loading rate of 2 l/s/m^2 and various underflow rates

number of typical efficiency curves produced by the model for a separator surface loading rate of 2 $l/s/m^2$ (defined previously), also illustrating how the outputs vary depending on the selection of the underflow proportion.

For practical application, where details of the effluent particle settling velocity distribution are known, it is possible, by means of reference to the efficiency curves, to compile overall performance figures. Such an operation is readily performed by means of a computer based spreadsheet. The model has been extensively validated, and has been shown to be an appropriate tool to aid in separator sizing.

Table 1 shows the results of a number of validation studies, where the model has been used to predict the performance of a number of actual installations (extracted from previous publication (6)). The figures were obtained by sampling and analysing flows from both the inlets and outlets of the installations in question. Utilising a technique developed by Aston University (7), the particle settling velocity distribution of the influent flow can be established, and this information, once fed into the semi-empirical model, enables the performance predictions to be produced. The ability of the model to provide reliable predictions for the overall efficiencies of the systems in question is clear.

Table 1 Observed and predicted efficiency comparisons

Site	Suspended solids removal (%)	
	Observed	Predicted
Backwell	25	28
Frome	51	51
Countess Weir	51	47
Totnes (crude only)	47	44
St Austell/Par	65	68
Motney Hill	52	50
Totnes (crude & recycle)	64	67

CFD SIMULATION OF A LARGE SCALE HYDRODYNAMIC SEPARATOR

As part of a project related study to investigate the impact of a particular cost saving design modification on performance, a 3-dimensional model was constructed of a 12 metre diameter HDVS. The commercial CFD code, Fluent (V4) was used, in conjunction with the Geomesh preprocessor software.

Fluent, as with most other CFD codes, uses a finite difference approach to solving the fundamental equations that govern fluid flows. Flow parameters are effectively propagated through the flow domain via a network of communicating control volumes, or computational cells. This is an iterative procedure, requiring that momentum and energy conservation is achieved to a substantial degree at all points before the solution can be regarded as converged. Further details of the underlying equations and procedures involved are well documented by Fluent (8).

In order to define the internal flow volumes of the separator with sufficient resolution, the system was divided into around 50,000 computational cells. A sectional view through the grid, also illustrating the geometry, is shown in Figure 3.

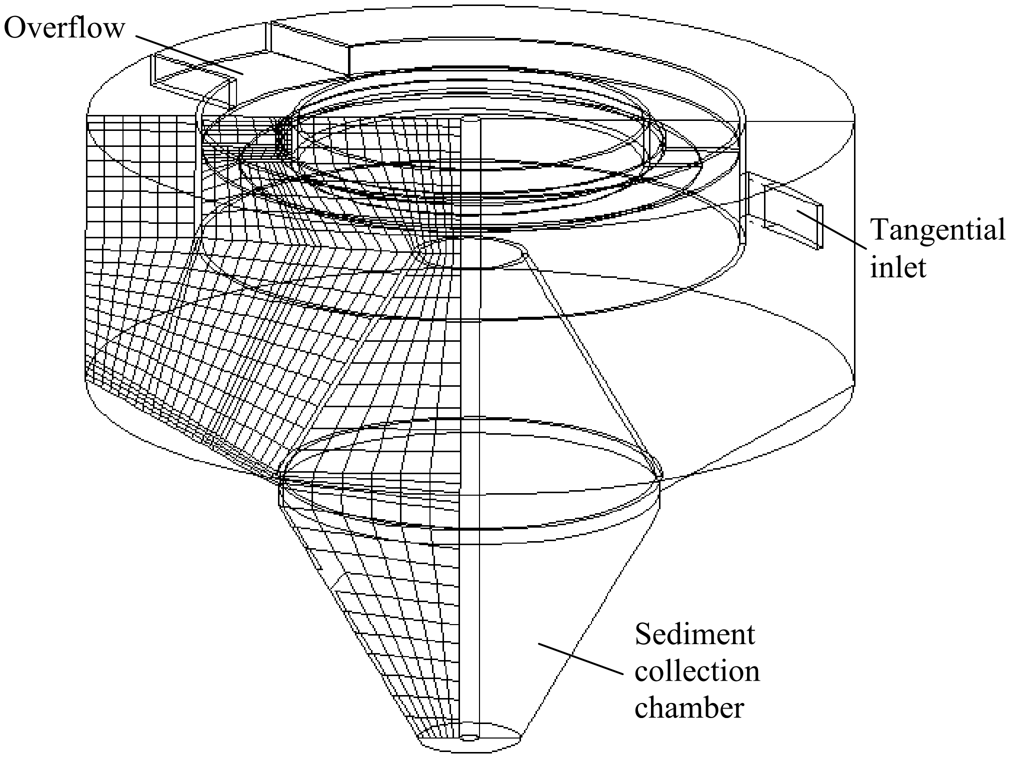

Figure 3 Sectional view through the CFD grid showing geometry features

Given the inherent complexity involved in the simulation of free surfaces, the system top water level was represented by a frictionless boundary, a simplification strategy that is well known to CFD modellers. Using the Reynolds stress model (RSM) of turbulence, the most appropriate for complex turbulent and swirling flow problems, the model was run until a converged solution was obtained at three different inlet flowrates. These were selected to give surface loading rates of 1, 2 and 3 $l/s/m^2$.

CFD SIMULATION VELOCITY PREDICTIONS

One of the main fundamental outputs of a CFD simulation are flowfield characteristics. These have varying relevance, depending on the objectives of the study. In some cases, particularly for academic studies, researchers are interested in comparing the fine details of the flows (e.g. velocity magnitude and turbulence quantities) with detailed measurements, such as those obtained via laser Doppler anemometry (LDA) measurements (9). This kind of work is fundamentally important in driving the further evolution of the codes. In other cases, general flow characteristics, for example, the location and direction of recirculation zones, and their inter-relation with geometry features, can be of interest for practical design/development purposes (10). Generally, if particular flow characteristics can be identified as being important to the quality of, for example, a process, then there is potential that these can be tuned, via geometry adjustment, to yield positive improvements.

Figure 4 shows a typical velocity vectors plot, predicted for the separator at a surface loading rate of 2 $l/s/m^2$. Axial-radial flow characteristics (excluding tangential/swirl components) across a vertical section through the system are shown. The presence of secondary flow recirculation in the system is clear. A relatively stagnant region under the lower cone is evident. This would be conducive to preventing the re-entrainment of sediments, once collected. Similarly, Figure 5 shows the tangential-radial flow characteristics predicted in a horizontal mid-sectional plane. The formation of a predominantly ‘forced’ type vortex within the chamber is evident. It is beyond the scope of the paper to discuss these predictions in detail, given that they were not the primary outputs of interest.

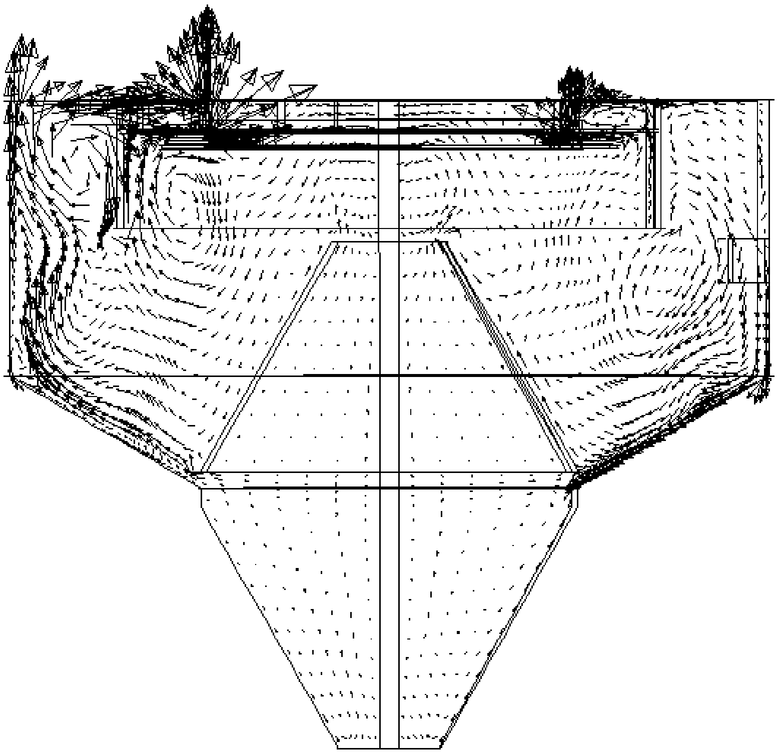

Figure 4 Axial-radial velocity vectors at a surface loading rate of 2 l/s/m^2 (peak vector magnitude 0.13 m/s)

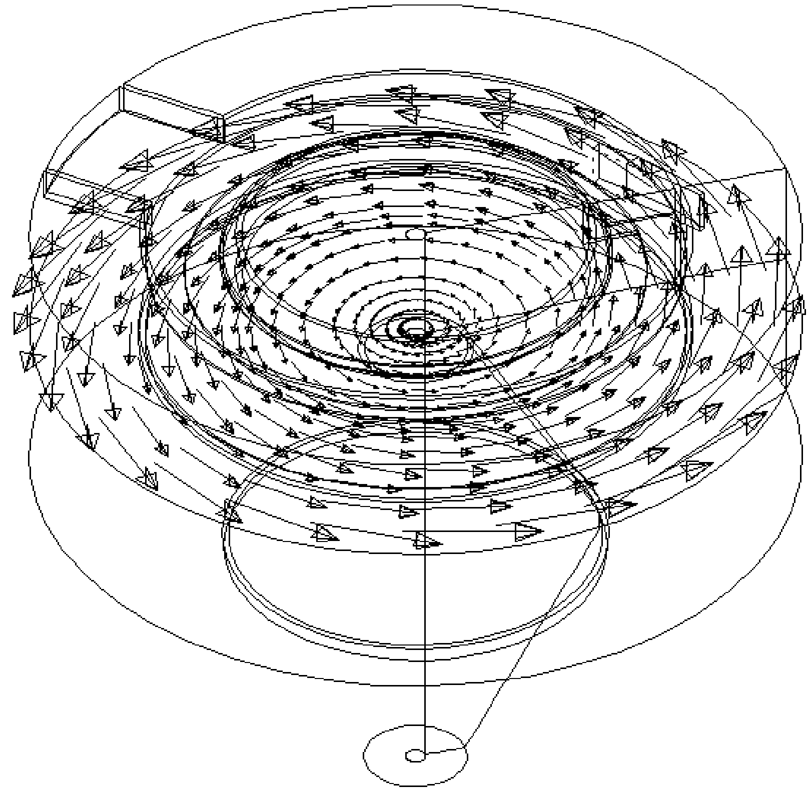

Figure 5 Tangential-radial velocity vectors at a surface loading rate of 2 l/s/m^2 (peak vector magnitude 0.57 m/s)

SEPARATOR EFFICIENCY PREDICTIONS AND COMPARISONS

In many CFD studies, secondary outputs, for example, chemical mixing effectiveness, or particle trajectories, have formed the focus of interest. Extensive studies performed by others have suggested that CFD models are capable of providing reliable predictions for the particle retention performance of various separator chambers, including hydrodynamic separators (11)(12)(although generally at higher loading rates than considered here). In the current work, although the primary output of interest was particle separation efficiency, the emphasis was mainly on relative change resulting from design modification, rather than the more specific qualitative outputs. Due to the confidential nature of the work, the details of the modification in question can not be revealed.

The Fluent software incorporates a Lagrangian particle tracking routine that enables the path of spherical particles of user-defined size and density to be calculated, based on the relative magnitudes of gravitational, centrifugal and drag forces. This is also able to take into account the chaotic effects of turbulence. In order to obtain acceptably reliable data, particles of various sizes and densities (sizes from 10 to 1000 μm, densities from 1040 to 2650 kg/m^3) , covering a range of settling velocities, were injected into the system at a number of locations in the inlet. For each, the destination of 500 injections was calculated. Any particles that came into contact with the inner walls of the sediment collection chamber were deemed to have been separated. Given the long duration particle residence time expected for the separator (potentially over 30 minutes at 2 l/s/m^2) and the constraints within the software relating to particle tracking duration, a number of particles, in each case, reached neither the sediment receptacle or the overflow, leading to termination of the calculation. Given the absence of potential recirculation traps, as might reside in the corners of a rectangular chamber, it was regarded that these particles were actually still in transit to a final destination, and were thus assumed to divide in the same proportion as those particles that did terminate in a coherent location.

The results of the particle tracking studies are shown in Figure 6. The predicted particle removal efficiency of the separator prior to modification is presented for the three flow conditions. Also shown on the plot is the efficiency prediction for the

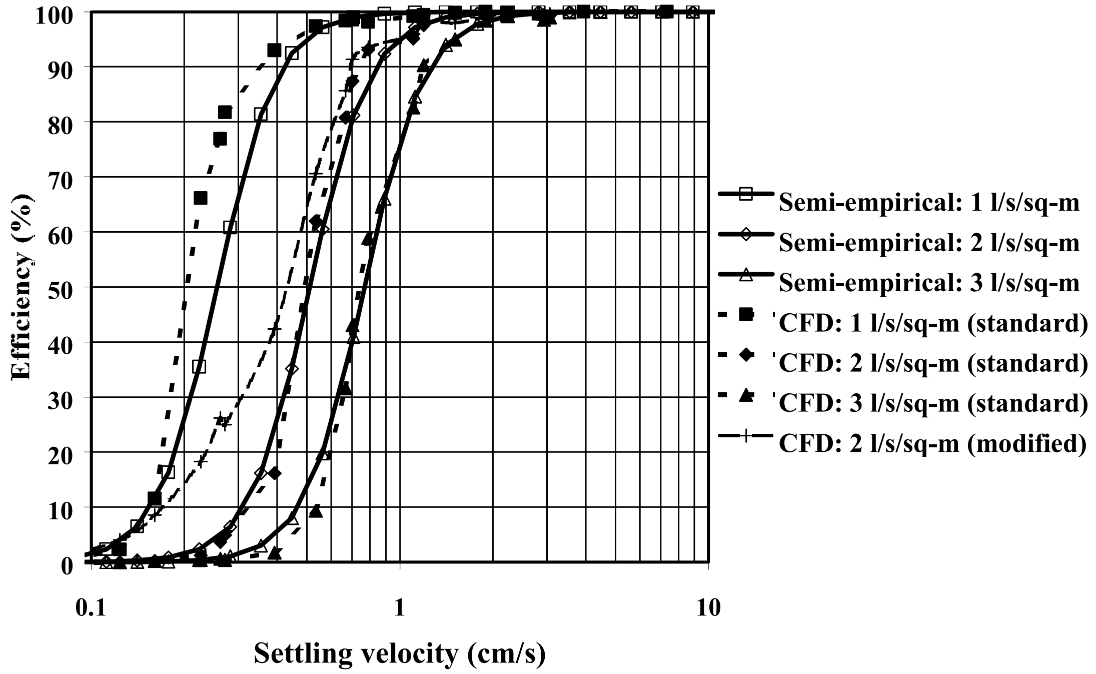

Figure 6 Comparison: CFD efficiency predictions for standard and modified designs and predictions from semi-empirical model

modified design at 2 l/s/m^2, along with curves produced from the semi-empirical model at the three flowrates, and for an underflow proportion of 0.01 %. The basis for this underflow selection is explained below.

The effect of increasing underflow rates in the semi-empirical model is that particle removal efficiencies increase. Conversely, as underflow rates tend to zero, efficiencies progressively reduce (the recovered fraction becomes coarser). In practice, separator systems tend to be designed to operate with a minimum underflow rate of 5 % (relating to potential solids fraction of the influent). Given that the CFD simulations considered a system with no underflow as such, it is impossible to make direct qualitative comparisons with the data produced by the semi-empirical model (indeed, the model will not accommodate an underflow value of zero). By one argument, a separator defined to have no underflow might be deemed to have zero efficiency, despite having particle retention capabilities. The selection of an underflow proportion of 0.01 % in the semi-empirical model was made purely as it gives good alignment of the curves, enabling some relative comparisons to be made.

Encouragingly, the shapes of the efficiency curves correspond particularly well, especially for the 2 $l/s/m^2$ and 3 $l/s/m^2$ cases. Also corresponding well is the relative shift in the curves that results from the change in the surface loading rate from 1 to 2 and 3 $l/s/m^2$. This gives confidence in the use of CFD as a tool for the assessment of 'relative' effects.

For the modified design of the system, the CFD model predicts a marginal increase in the efficiency of the system relative to that observed prior to modification.

PRACTICAL IMPLICATIONS

As a result of the above study, and following a number of small scale experimental investigations, it has been concluded that the simplifying modification proposed for the system is valid, having a positive, rather than detrimental impact on performance characteristics. The modification in question has subsequently been adopted, leading to a significant saving in terms of fabrication costs for the project in question.

CONCLUSIONS

A review of the design of many hydraulically based wastewater treatment processes or systems could potentially yield significant savings in terms of both installation costs, and also costs associated with improved operational characteristics (e.g. maintenance costs and plant size reduction associated costs).

The study has demonstrated how CFD simulation has been used to assess the relative impact of design change on a hydrodynamic separator, yielding direct savings in fabrication costs. Comparisons between efficiency curves produced by a well validated semi-empirical model and those produced by the CFD simulation suggest that CFD is an effective tool for assessing the relative impact of change on the outputs of a system. This clearly has implications to the optimisation of other wastewater treatment systems, and indeed, hydraulic systems in general.

REFERENCES

(1) ALKHADDAR, R. M., HIGGINS, P. R., PHIPPS, D. A. AND ANDOH, R. Y. G. The residence time distribution of prototype hydrodynamic vortex separator operating with a baseflow component. *8th Int. Conf. On Urban Storm Drainage*, Sydney, Australia. Vol 1. 1999.

(2) FARAM, M. G., ANDOH, R. Y. G. AND SMITH, B. P. Hydro-Jet Screen™: A non-powered self-cleansing screening system for storm overflow screening applications. *Wastewater Treatment: Standards and Technologies to meet the Challenges of the 21st Century*. CIWEM/AETT Millennium Conf., Leeds, UK. 2000 (submitted).

(3) GARDNER, P. AND DEAMER, A. P. An evaluation of methods for assessing the removal efficiency of a grit separation device. *Water Science Technology*. Vol 33, No 9. 1996.

(4) LECORNU, P., FARAM, M. G. AND ANDOH, R. Y. G. A novel device for the removal of grits and oils from stormwater run-off. *Wastewater Treatment: Standards and Technologies to meet the Challenges of the 21st Century*. CIWEM/AETT Millennium Conf., Leeds, UK. 2000 (submitted).

(5) COLE, T. J. AND WILLIAMS, C. A. A case study on the Eff-Pac™ process at Alcan Chemicals Europe Limited, Burntisland. *Int. Conf. on Industrial Effluent Technology: Meeting the Demands of the 21st Century*. BHR Group pub. 33. 1998.

(6) ANDOH, R. Y. G. AND SMISSON, R. P. M. High rate sedimentation in hydrodynamic separators. *2nd Int. Conf. on Hydraulic Modelling; Development and Application of Physical and Mathematical Models*, Stratford, UK. 1993.

(7) TYACK, J. N., HEDGES, P. D. AND SMISSON, R. P. M. A device for determining the settling velocity grading of storm sewage. *6th Int. Conf. on Urban Storm Drainage*. Niagara Falls, Ontario, Canada. Vol 2. 1993.

(8) FLUENT. User's Guide (Fluent Version 4.3). Fluent Incorporated. 1995.

(9) BATES, C. J., ZONUZI, F., FARAM, M. G. AND O'DOHERTY, T. Experimental and numerical comparison of the flow in a 90° tee junction. ASME Fluids Engineering Division (Publication) FEDSM. Vol 217. 1995.

(10) FARAM, M. G. AND ANDOH, R. Y. G. 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. Vol 3. 1999.

(11) HARWOOD, R. AND SAUL, A. J. CFD and novel technology in combined sewer overflow. *7th Int. Conf. on Urban Storm Drainage*. Hannover, Germany. Vol 2. 1996.

(12) HARWOOD, R. AND SAUL, A. J. The influence of CSO chamber size on particle retention efficiency performance. *8th Int. Conf. on Urban Storm Drainage*. Sydney, Australia. Vol 1 . 1999.