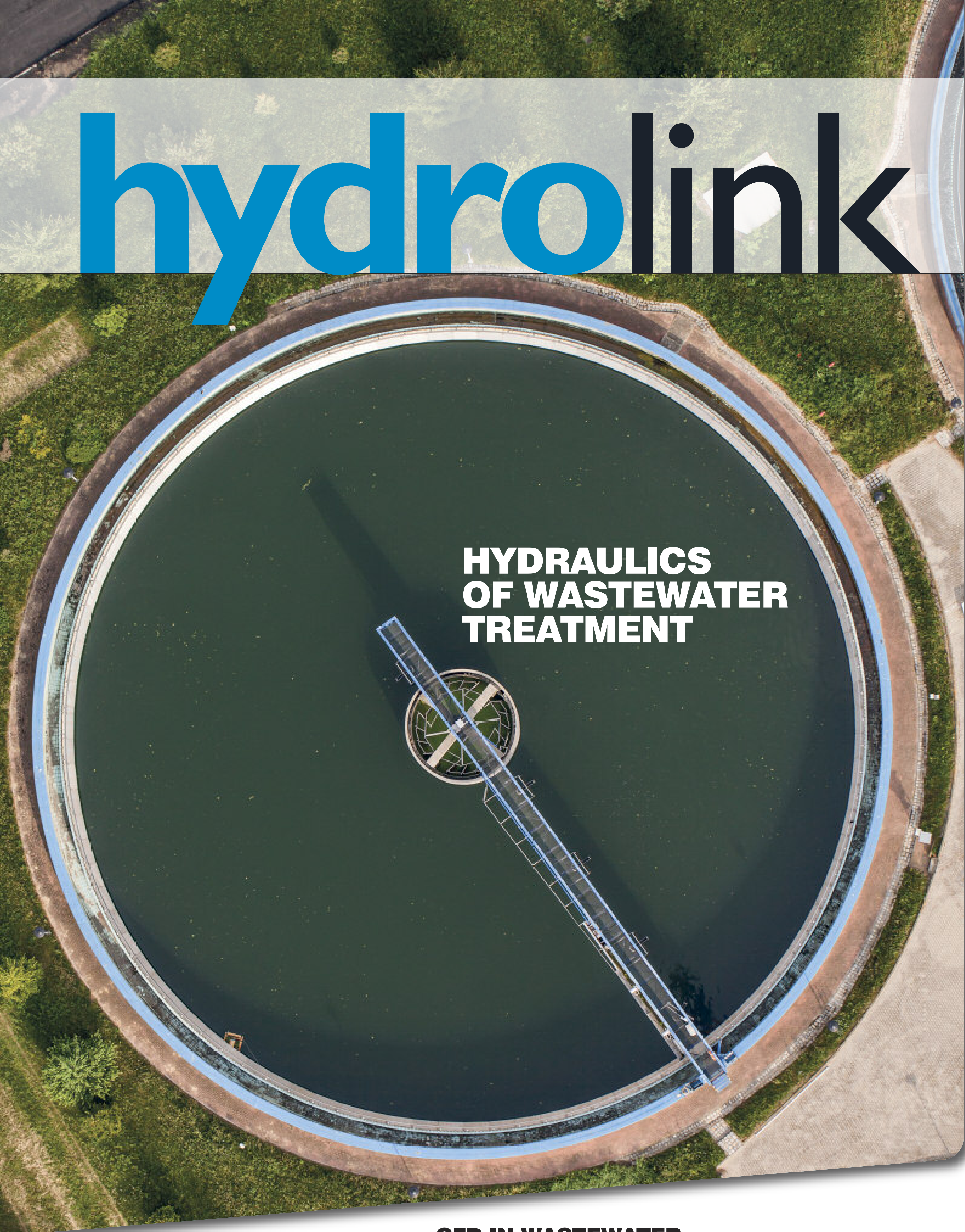
hydrolink
NUMBER 2 / 2016



HYDRAULICS OF WASTEWATER TREATMENT

IAHR AIRH
International Association for Hydro-Environment Engineering and Research
Supported by
Spain Water and IWHR, China

THE CONTRIBUTION OF ADVANCED SIMULATION TOOLS TO THE HYDRAULIC DESIGN OF WATER AND WASTEWATER TREATMENT FACILITIES

EDITORIAL BY ANGELOS N. FINDIKAKIS

Wastewater treatment has evolved over the last two centuries, starting with the use of trenches and pits for the settlement of solids in the early nineteenth century, and continuing with the development of biological filters in the 1890s. The first experiments on activated sludge in 1913, were followed shortly afterwards by the construction of full-scale activated sludge plants, and eventually by the design and use of methods for the removal of nitrogen and phosphorus later in the twentieth century. Through most of this period the emphasis was naturally on the development of efficient methods for the removal of organic matter and nutrients from municipal wastewater. Early on, the design and sizing of the tanks and conduits used to move the wastewater through the different stages of treatment was based mostly on relatively simple traditional hydraulic calculations. Later on, the design and optimizing of the operation of different hydraulic structures, such as pump stations and settling and storage tanks, sometimes called for more in-depth understanding of flow conditions in these elements of the plants with the aid of physical or numerical models. The exponential growth in computer power over the last few decades has made possible the application of sophisticated Computational Fluid Dynamics (CFD) models to the design of different parts of wastewater treatment plants. The development of these models has advanced to the point where they can account for the rheological properties of municipal and industrial wastewater, which often behaves like a non-Newtonian fluid, and can also incorporate the simulation of chemical and biological processes. The use of well-validated models supported by powerful user-interfaces that facilitate pre- and post-processing of model data, makes it possible to evaluate multiple design options and explore alternative operation modes within much less time than required for the same level of analysis with physical models.

Angelos N. Findikakis
Hydrolink Editor

This comes at a time when the need for increased water treatment capacity in response to the rapid growth of many urban centers calls not only for the construction of new or the expansion of existing facilities, but also for their optimal operation and the use of increasingly more efficient designs. This issue presents several articles on the use of modern hydraulic analysis and simulation tools to improve the design and performance of water and wastewater treatment facilities.

Over the last few years, CFD has been used in the hydraulic design of many individual processes and components of wastewater treatment plants, including disinfection contact tanks, distribution chambers, pump stations, flow measurement flumes, separation of solids, filtration, and mixing. The broad range of CFD applications in wastewater problems around the world is illustrated in an article that discusses the design of primary settling tanks for a wastewater treatment facility in Chicago, the study of dead zones and short-circuiting in wastewater stabilization ponds in Bolivia and the optimization of the design of an anaerobic digester treating livestock waste at a farm in Costa Rica.

A common problem whose solution has been facilitated by computational hydraulics is that of recirculation and insufficient mixing in various tanks. One of the articles in this issue presents three examples of the use of CFD for this type of problem, starting with the case of a distribution chamber where special site conditions prevented a conventional design and required the evaluation of different alternatives for the location of the introduction of the return activated sludge in the chamber in a manner that would produce a uniform load distribution. A second example is the simulation of the flow in an activated sludge process reactor exploring various design modifications that would prevent short-circuiting of the flow between the inlet and the outlet of the reactor. The third example in the same article is from a study aimed at producing an effective inlet design for the final settlement tanks to achieve uniform flow distribution and dissipation of the inlet momentum. In this case the evaluation of different alternative designs using CFD helped determine the required clearance between the sludge bed and the effluent weir, leading to reduction of the concentration of solids in the effluent, and an improved performance of the system in response to flow surges during storm events.

Advanced numerical methods in combination with physical tests are also used in the hydraulic design of parts of drinking water treatment facilities. Another article in this issue describes the use of a free-surface numerical model to optimize the placement and location of baffles and orifices in a storage tank in order to minimize the detention time and avoid recirculation and stagnant areas. The same article presents an example of a two-phase (air and water) flow simulation in an ozonation tower of a water treatment facility, which helped assess the ozonated air transfer to water from differently positioned diffusers.

The treatment of industrial wastewater streams presents special challenges. One such example is the simulation of the mixing of a slurry with ten percent solids by weight, both dissolved and suspended, as part of an industrial wastewater treatment process, described in another article. The analysis of this case, which was performed in parallel with a physical model of the problem, required the use of an advanced CFD model to account for six solids phases representing different particle sizes and densities in a non-Newtonian carrier fluid and for the gas in the upper part of the vessel.

Dynamic simulation models offer new opportunities for integrating hydraulics, controls, and process treatment, in order to improve design coordination, identify more efficient solutions, enhance system understanding, and optimize operations. One of the articles in this issue describes the development of such a tool which can be used not only to confirm equipment sizing, explore control strategies, evaluate different operating parameters such as costs, water loss, and energy and chemical usage, but also to assist operator training and to optimize the overall operation of a treatment plant.

Another article discusses how secondary clarifiers can be the limiting factor for the hydraulic load in wastewater treatment plants especially during rain events, when the plant is located downstream of a combined sewer system, and presents the development of a combined controller for real time control of secondary clarifiers, which makes it possible to increase the hydraulic load during rain events without affecting the operation and control of the upstream biological processes.

I believe that the articles included in this issue will give our readers a glimpse into the possibilities offered by advanced simulation tools supporting the development of more efficient designs and optimal operation methods of drinking water and wastewater treatment facilities.

IAHR
International Association for Hydro-Environment Engineering and Research

IAHR Secretariat

Madrid Office
IAHR Secretariat
Paseo Bajo Virgen del Puerto 3
28005 Madrid SPAIN
tel +34 91 335 79 08
fax + 34 91 335 79 35

Beijing Office
IAHR Secretariat
A-1 Fuxing Road, Haidian District
100038 Beijing CHINA
tel +86 10 6878 1808
fax +86 10 6878 1890

iahr@iahr.org
www.iahr.org

Editor:
Angelos Findikakis
Bechtel, USA
anfindik@bechtel.com

Editor Assistant:
Elsa Incio
IAHR Secretariat
elsa.incio@iahr.org

ISSN 1388-3445

Cover picture: Aerial view of sewage treatment plant, thinkstockphotos.com

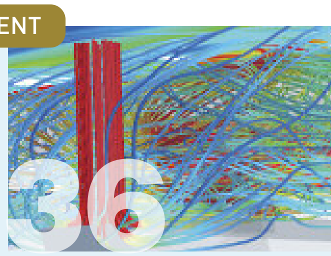
IAHR AIRH

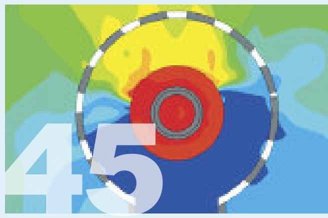
Supported by
Spain Water
and IWHR, China

NUMBER 2/2016

IN THIS ISSUE

THEMED ISSUE: HYDRAULICS OF WASTEWATER TREATMENT





CFD IN WASTEWATER TREATMENT PLANTS

BY PEDRO FONSECA & NELSON MARQUES

Computational Fluid Dynamics (CFD, please see Ferziger and Peric, 2012) generally allows a comprehensive analysis of the hydraulic behaviour of a design. CFD can address many problems that historically were studied using scale physical modelling. Both CFD and physical models are typically used to study three-dimensional flow problems, both can generally handle steady and transient simulation scenarios, and both require a relatively high level of specialised resources. However, there are also important differences between CFD and physical model studies, as discussed later.

Closely tied to developments in computer technology, CFD enjoys the benefits of the exponential growth of computational power. Most of the progress in recent years is not so much in the understanding of the underlying physics of the flows modelled, but more in the easiness of use, manageable case dimensions, computer response time, cost, etc.

As an example, we can consider a well-known water treatment process, the disinfection contact tank, and more specifically, the determination of its residence time distribution, which is probably the simplest hydraulic analysis problem for the application of CFD in water treatment. It is very easy to understand the results of this analysis (shortcuts, recirculation or dead volumes, etc.), and imagine possible optimization. A lot has already been explored on this particular topic. Either profiting from the experience gained with existing tanks or reservoirs (field based tracing), or from systematic R&D work done on reduced scale physical models, and, more recently, on CFD models.

Several reasons make this an interesting case of discussion:

- The computational effort needed for the CFD analysis of this problem is relatively small, especially when compared with the effort needed for a physical model study.
- Under certain circumstances, a 2D analysis is an acceptable simplification of this problem, allowing for a large number of simulations to be ran in a very short amount of time.
- It is relatively simple to conceive a design driven by parameterization (e.g. number, length and position of baffles), allowing for the automatic search of an optimum design within the given constraints.
- Despite the long history of accumulated experience on this problem, a custom-made design seems to be always preferred, either because a unique geometry is imposed by particular site constraints (refurbishments and extensions, etc.), or simply because a particular concept or design compromise has not been characterised before.

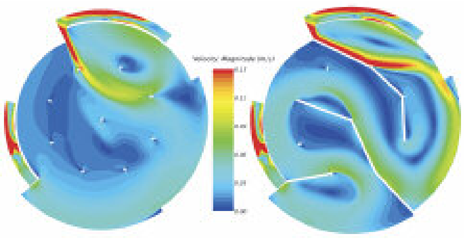


Figure 1. Contact tank with two distinct baffles arrangements (the arrangement on the left has baffles only at the entrance, but not in the interior of the tank). Velocity distribution on a horizontal section plane at half height. (Volume: 100 000 m^3; flow rate: 2 m^3/s)

Figure 1 shows the simulated velocity distribution in a disinfection contact tank for two different baffle configurations. This example illustrates well the advantages offered by CFD in the hydraulic design of water treatment plants. Specifically:

- There are clear basic benefits when compared to physical modelling, like time and cost;
- The simulated physics, for the most basic analysis, are relatively simple;
- The potential for automated design optimization exists, while there is no such possibility in an equivalent physical scale model study;
- The modelled physics can promptly be extended to the limits of our needs or knowledge, allowing the designer to go beyond the purely hydraulic aspects, into a more comprehensible water treatment analysis (integrating pathogen inactivation laws, by-product formation estimation, etc.).

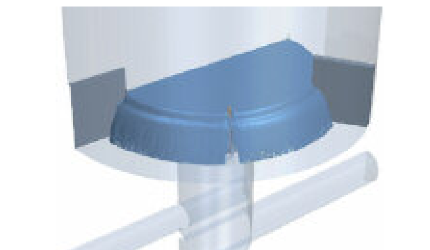


Figure 2. Discharge weir with water injection at the bottom (Diameter: 8 m; flow rate: 4 m^3/s). This injection pattern gives rise to non-uniform flow distribution across the weirs since the flow is not guaranteed to be symmetrical

If CFD can be of great assistance in the design and optimization of what seems to be a simple and long-mastered water treatment process, its application to the rest of the treatment plant has literally no limits.

In developing and using CFD models (Casey and Wintergerste, 2000), productivity is mostly controlled by the ease with which the a user can develop the numerical grid for the representation of the geometry of the problem domain and the application of boundary conditions (using, for example, a modern GUI like the one provided by STAR-CCM+®), and by how powerful the computer resources performing the CFD analysis actually are. The latter, in particular, have to be adequately prepared for the size and type of intended analysis. A steady-state analysis, for example, is typically limited by RAM memory whereas a transient analysis is mostly limited by the actual time it takes to perform the simulation. It is also true that there are still several limitations in physical models and data needed by CFD, e.g. models adequate for multi-fluid, multi-phase flows and data for non-Newtonian fluids or even, at a more practical level, the absence of a given feature from the capabilities of the employed CFD code (SIAMUF and Sommerfeld, 2008), which prevent a more widespread use of the tool. CFD analysis exhibits a huge variation in what regards physical complexity, mesh size, CPU time and, consequently, cost. However, within the purview of large structures, there are several typical water and wastewater treatment components where CFD analysis is now easily deployed. A few examples follow. Analysis have been performed with STAR-CCM+® (mostly) and OpenFOAM®.

Distribution Chambers

This type of structure aims to split the flow between distinct branches. Typically made out of concrete, they can be up to several dozen meters in diameter or length. Their design is influenced by process equipment located upstream and downstream, which, more often than not, translates to space constraints, especially in retrofits. Also because of this, these structures tend to be dealt with a case by case design, thereby proving CFD essential. Static structures are preferred and layouts range from circular (Figure 2) to linear (Figure 3). Flow rate control outlet is generally performed by free-fall sharp crest weirs. This approach decouples flow distribution from downstream influences but renders the outlet flow rate very sensitive to local perturbations.

Pumping Stations

Pumping Stations perform an essential function. The overriding concern in their design and operation is to ensure uniform flow approach to the pumps. Moreover, for each pump, a certain set of flow parameters must be met in order to insure that the pumps themselves operate efficiently and reliably. These requirements go against the local geometry and flow conditions, non-withstanding the conventional design practices. CFD provides the necessary approach to assess the aforementioned concerns once boundary conditions are properly set in the analysis on the pump side. In particular, the pumps themselves are not modelled. In modelling these problems the computational domain extends at some length inside the pumps inlet ducts. In this setup the model equations make it possible to capture the inlet pre-swirl angles, or the velocity distribution at the pumps suction (seen in Figure 4 in the form of streamlines coloured by velocity magnitude) and, in case of a non-conformity with the American National Standards Institute/Hydraulic Institute (ANSI/HI) guidelines, study the effect of design changes to improve them. Even on a stricter adherence to current ANSI/HI guidelines, CFD analysis allow for the identification of the most adequate dimensions for physical model testing.

Flow Rate Measurement

Measurement of flow rates is much needed for both process control and for economic reasons. It goes without saying that much effort is continuously devoted to develop reliable and accurate flow measuring devices and techniques. Static structures like flumes, however, continue to be a popular approach given their relatively low-cost and reliable operation. However, the flume may also be under the influence of upstream structures, beyond standard recommendations, that end up affecting the incoming flow pattern and, thus, risk rending the calibration curve meaningless. Another potential issue in the use of the calibration curves can arise in the installation or the in-situ manufacturing of the flume, since both can lead to geometrical deviations in lengths and angles from the standard. The effect of these deviations on the flow measuring capability has then to be assessed and, if necessary, new calibration curves determined. All of these concerns can be easily assessed with CFD if the computational model is made big enough to include the influence of said disturbances (Figure 5) and/or produced with CAD data that includes in-situ geometrical measurements.

Separation

Separation of suspended solids is one of the most common operations in a wastewater treatment plant. However, the range in solid sizes usually found in wastewater has led to a diverse set of separation approaches, which also has led to a diverse set of simulation methods when using CFD to study this subject (Wicklein et al, 2016). The basic distinction between methods is in how the suspended elements are considered: either as a continuum (Eulerian approach) or as a discrete field (Lagrangian approach). The former essentially implies a mixture of several components – forcibly including water and at least one type of suspended solids – whereas in the latter each particle is tracked individually. Both have limitations regarding the type of physical processes that can be accurately modelled and simulation cost are usually high. However, choosing one over the other approach may be based on the water concentration relative to the suspended solids: for low values an Eulerian approach is advisable (Figure 6), for high values Lagrangian is possible (Figure 7).

Filtration

Filtration is a separation process but entails an active element that stands in as a filtration element. Analysis of pressure loss effects and associated flow distribution are common practice, but the actual filtration effectiveness can only be assessed case-by-case due to a usually wide range of size and time scales involved in the process. Numerically, this disparity in scales almost always amounts to costly simulations, unless the problem can be reduced to a setup which is still economically viable and produces results which are statistically significant. For example, retention rates of

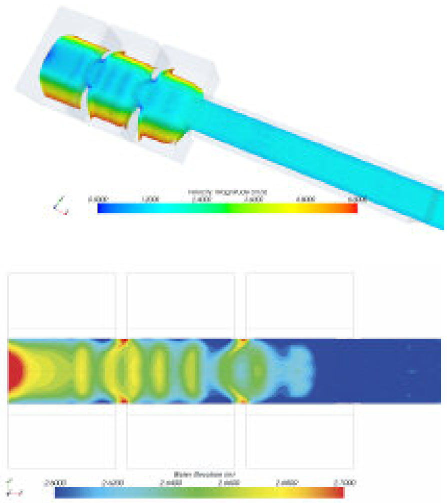


Figure 3. Water approaching through open channel (channel width: 2.5 m; flow rate: 6.5 m^3/s). Velocity distribution in impinging flow at closed extremity composes non-uniformity in the flow caused by lateral discharge weirs. Lower image displays free-surface height relative to datum

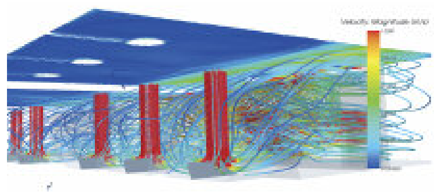


Figure 4. Pumps inlet at pump station. Flow rate: 5 m^3/s, distributed through 5 submersible pumps. Flow pattern shown through streamlines to assess admission requirements for safe pump operation

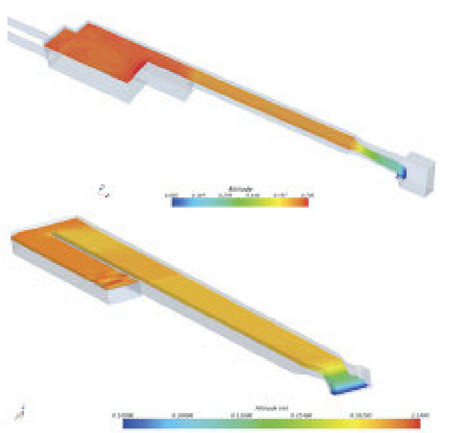


Figure 5. Flumes operating under the influence of discharge chamber and channel bend. Flow rates approximately 1 m^3/s

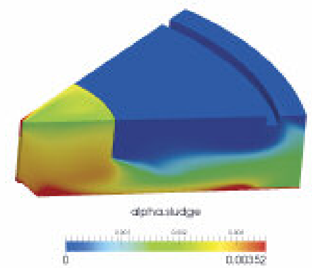


Figure 6. Secondary clarifier (18m radius) sludge blanket modelling. Flow rate ~1.2 m^3/s. Colours represent sludge concentration

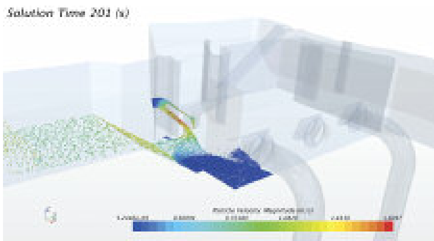


Figure 7. Grit-trap optimization. Flow rate: ~2 m^3/s. Each particle represents a grit element

individual particles, solid, liquid or gaseous, can be accurately estimated with a fraction of their actual flow rate (e.g., Figure 7).

Mixing

Mixing is always accomplished through kinematic means, i.e., fluid motion. However, the way that fluids are set in motion may vary. Simple fluid direction change may be achieved by baffles, by impellers, or through gas injection. All these options can be handled with CFD. However, when using impellers, the simulation can be carried out in the reference frame(s) of the rotating blades. This approximation allows a steady-state simulation at the expense of obtaining a time-averaged flow field (Figure 8). In gas based mixing (Figure 9), on the other hand, there is a need for a two-step approach because the actual gas-injection and gas-transport processes need to be captured properly on every single case through an adequate two-phase flow model. The mixing is a consequence of this process. For the former, it is essential to possess a 3D CAD representation of the actual impeller blades, whereas in both cases it is important to use accurate fluid properties.

Non-Newtonian Flows

Non-Newtonian fluids are usually found in some

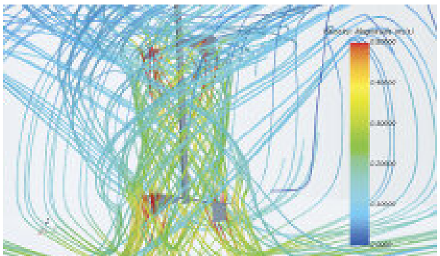


Figure 8. Streamlines across mixers simulated with rotating reference frames

Figure 9. Central injection gas based mixing. Gas volume fraction iso-surface and streamlines coloured by velocity magnitude

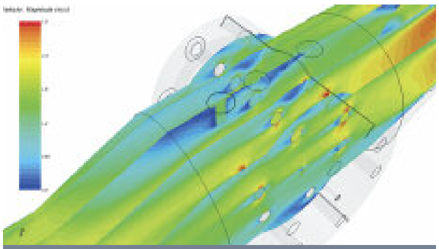


Figure 10. In-line UV reactor with lamps placed perpendicularly to the fluid flow. The intensity of the germicidal irradiation is computed throughout the domain and its eventual absorption monitored through particles released in the flow

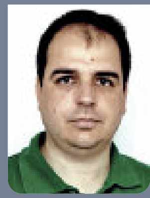
of the wastewater treatment plant components due to the high concentration of suspended matter in the water which alters the physical properties of the fluid. From a CFD point of view, this variation in rheological properties can be properly accommodated. However, the actual rheological properties are usually highly uncertain. To overcome this uncertainty, it is possible to perform sensitivity studies whereby rheological key properties are varied with a view to assess the impact of their variation.

Chemical and Biological Processes

Some processes where chemical reactions occur can be tackled based simply on the characterization of their residence time. In other cases, the reaction's locus and rates matter and must be studied via direct modelling like in Nitrification-Denitrification processes. However, some of these processes have a biological basis, which should be accounted for – at a cost – depending on whether the process will be studied directly, i.e., with local reaction rates, or indirectly, through residence time. The development of highly integrated, 3D simulation methodologies is still an ongoing job in such cases.

Physical Processes

The range of physical processes present in a water or wastewater plant is very broad. CFD can tackle most of them, either physical (e.g., UV disinfection, see Figure 10, where the total amount of germicidal radiation to which a pathogen is exposed can be estimated) or chemical (Ho et al, 2011). Operational concerns in wastewater treatment plants also matter, since transient effects (for example while opening or closing valves but not necessarily leading to water-hammer effects) are as relevant for the control and command part of things as they are for the process itself. CFD in such cases may provide a viable alternative to model testing or provide better estimates to sustain procurement of specific models of process equipment (e.g., back pressure regulators).



Pedro Fonseca **is Suez treatment infrastructures hydraulic discipline manager. He started working at Degrémont, in 1999, while still finishing his degree in Environmental Engineering (sanitary specialization), at Universidade Nova de Lisboa (Portugal). Having started his career participating in both potable and wastewater treatment plants start-up and day-to-day operation activities, he proceeded with an international career mostly focussed on the detail engineering of water treatment plants, always within Suez group, having been based in France, USA and Portugal. Managing the hydraulic discipline since 2012, he participates in the research and development of new products, as well as in the basic and detail design of water treatment plants, all around the world.**
pedro.fonseca@suez.com

Nelson Marques **is the CEO of blueCAPE, a company specialized in Computer Aided Engineering, particularly CFD. blueCAPE provides consultancy and develops CFD software to accommodate internal needs and customers' requests across the spectrum of industries served, from energy to wastewater treatment. Clients can be found all around the globe. He holds a PhD in CFD and a Mechanical Engineering degree from IST, Portugal. Since 2000, his professional life has been spent performing CAE in Portugal, USA and the UK. Currently, Nelson Marques is an invited lecturer at Instituto Superior de Engenharia de Lisboa and has published some of his work.**
nelson.marques@bluecape.com.pt

Other topics of interest to the design and operation of water and wastewater treatment plants that can be studied using CFD models include ventilation and odour control, sludge after-treatment – including drying and, eventually, incineration – erosion, biogas separation and conditioning or condensing two-phase flows can all be tackled, benefitting from the track record gathered by CFD in other engineering fields. ■

References

Ferziger, J.H. and Peric, M., 2012. Computational methods for fluid dynamics. Springer Science & Business Media.
Casey, M. and Wintergerste, T., 2000. Best practice guidelines for industrial computational fluid dynamics of single-phase flows. Lausanne: ERCOFTAC (European Research Community on Flow, Turbulence and Combustion).
Swedish Industrial Association for Multiphase Flows (SIAMUF) and Sommerfeld, M., 2008. Best Practice Guidelines for Computational Fluid Dynamics of Dispersed Multi-Phase Flows. European Research Community on Flow, Turbulence and Combustion (ERCOFTAC).
Ho, C., Khalsa, S.S., Wright, H.B. and Wicklein, E., 2011. Computational Fluid Dynamics Based Models for Assessing UV Reactor Design and Installation. Water Research Foundation, Denver, CO.
Wicklein, E., Batstone, D.J., Ducoste, J., Laurent, J., Griborio, A., Wicks, J., Saunders, S., Samstag, R., Potier, O. and Nopens, I., 2016. Good modelling practice in applying computational fluid dynamics for WWTP modelling. Water Science and Technology, 73(5), pp.969-982.

COMPUTATIONAL FLUID DYNAMICS: A PROMISING TOOL FOR ANALYSIS AND DESIGN OF WATER AND WASTE-WATER TREATMENT

BY JIE ZHANG, XIAOFENG LIU, ANDRES TEJADA-MARTINEZ & QIONG ZHANG

Computational Fluid Dynamics (CFD) technology has been widely applied to modeling flow in water and wastewater treatment in recent years, e.g., water intake infrastructures, sedimentation basins, disinfection reactors, and activated sludge systems. The increasing interest in CFD applied to water and wastewater treatment is partly due to the rapid advancement of computer technology making the intensive computing affordable; and partly due to the demand for rapid modeling of the hydraulics in physical, chemical and biological treatment processes. Based on an improved understanding of these processes afforded by CFD, treatment facilities can be improved in terms of efficiency, cost, or regulatory compliance.

CFD applications to water and wastewater treatment have led to the development of various practical models (Zhang et al, 2016) and the exploration of various emerging applications (Verbyla et al, 2013; Kinyua et al, 2016). Beside hydraulics, attempts have been made to model chemical and biological treatment processes. However, due to uncertainties in chemical and biological kinetics models and difficulty in modeling turbulence-chemistry interaction, the primary focus of most current applications is still on the hydraulic performance (Zhang et al, 2014). This article showcases three recent studies of CFD applications in wastewater treatment with the hope to spur more interest in this research area and encourage wider adoption in the industry. At the end, we discuss the technical hurdles and offer some perspectives on the future directions.

Primary settling tanks

Primary settling is one of the key processes in wastewater treatment to remove particular matter. The efficacy of primary settling has a great impact on the downstream treatment units. Often the primary settling tanks consume a significant portion of the construction and operation cost. Thus a proper design to maximally utilize primary tanks is of great interest. Computational models have been used as an economical alternative to more costly and time consuming physical tests. In the settling tanks literature, there exists a hierarchy of computer models with various complexities, ranging from simple lumped models to sophisticated 3D models. In general, the inclusion of more physics, e.g., hindered settling and rheology of sludge, is always associated with more complexity. Among many other things, the most crucial processes in the primary tanks are the turbulent flows and the transport of particles. It is important to note that all the processes are coupled together and thus they need to be modeled as such. For the case of settling tanks, the tank will continuously and slowly accumulate particles until an equilibrium is reached. This aspect further increases the computational cost because most of the modeling exercises have to reach steady effluent concentration to evaluate the tank performance.

A CFD modeling was conducted to help the design of 12 large primary settling tanks for the Metropolitan Water Reclamation District of Greater Chicago (MWRDGC; Liu and Garcia, 2011). The tanks had a fixed diameter of 155 ft and a target removal rate of 50%. The sludge was withdrawn from the bottom hopper by an air-lift pump (on a 30-min on and 30-min off cycle). Each tank was designed to treat a flow of 22.58 million gallons per day (MGD) (average) and 40 MGD (maximum). A 3D computational model was developed using OpenFOAM (2010), an open source CFD platform, to optimize the tank depth, inlet feedwell dimensions, and inlet pipe arrangement. The particles were divided into size groups and the shear-induced flocculation was considered. The details of the model and its implementation in OpenFOAM can be found in Liu and Garcia (2011). The CFD model was validated with experimental data from a physical tracer study. The simulated concentration field within the tank at the end of a 1-hr sludge pump cycle is shown in Figure 1 (b). A sludge blanket

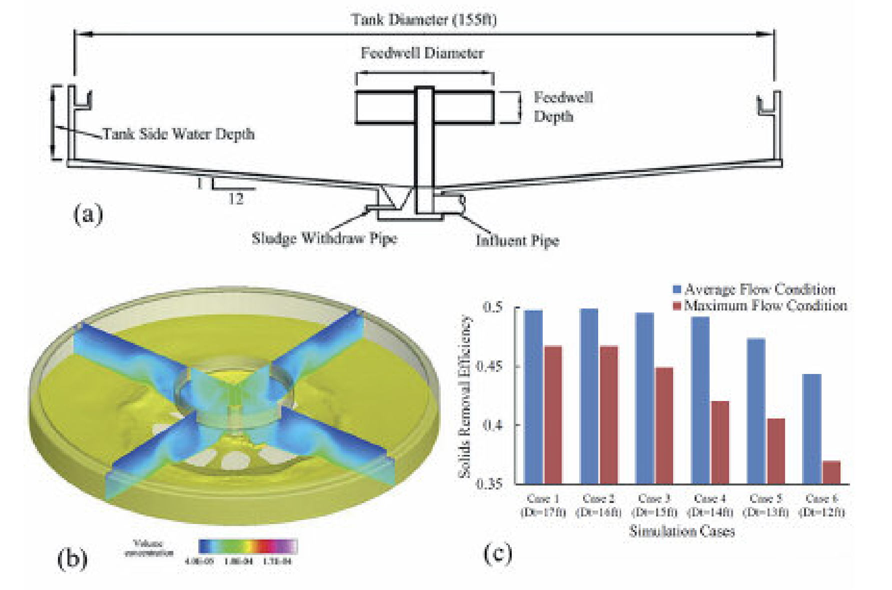


Figure 1. (a) Schematic of the primary settling tank; (b) volume concentration distribution in the tank; (c) solids removal efficiencies of different tanks under average and maximum flow conditions for different tank depths (D_t) (from Liu and Garcia, 2011. Reprinted with permission of ASCE)

can be observed on the bottom of the tank. The effect of the center well on re-distributing the flow and sediment can also be seen ("holes" on the bottom). Due to the blocking inlet piers, the sediment layer depth is not uniform. Instead, some of the sediment has been "blown" away and the bottom of the tank is exposed. Figure 1 (c) shows the simulated removal efficiency as a function of tank depth. In general, the tank removal efficiency decreases with the reduction of tank depth. This result makes physical sense because a deeper tank means a longer flow path and the particles have more time to settle out.

Stabilization ponds

Wastewater stabilization ponds (WSP) are a widely used and economically viable wastewater treatment technology that is critical for sanitation provision throughout the world. Importantly this technology may be more sustainable than mechanized methods of wastewater treatment and can be readily integrated with agricultural water reuse to improve food security, especially for smaller cities facing increasing population and urbanization (Verbyla et al, 2013). In addition, such an approach can offset the negative impacts of eutrophication while recovering valuable nutrients required for crop growth. However, there are challenges in managing a WSP that must continuously protect water quality and human health associated with parasite, bacteria, and virus removal (Verbyla et al, 2013). The level of pathogen removal is highly dependent on the hydraulic performance of a WSP, which also affects the removal of suspended solids (SS) and biochemical oxygen demand (BOD) (Verbyla et al, 2013). Therefore, improving the hydraulic performance of a WSP is an important management strategy not only for ensuring protection of public health and the environment, but also in maximizing the potential to reuse the treated effluent. Verbyla et al. (2013) conducted a CFD simulation on a facultative pond in San Antonio, Bolivia and found that the dead zones and short-circuiting observed in CFD results can help to explain the measured low helminth egg removal of the pond since these flow structures would decrease hydraulic efficiency. Ouedraogo et al (2016) further investigated the impact of sludge layer geometry on hydraulic performance of the facultative pond using a CFD model based on OpenFOAM (2010). Results indicate that sludge distribution and volume have a significant impact on the pond's wastewater treatment efficiency and capacity. Although treatment capacity is reduced with accumulation of sludge, the latter may induce a baffling effect which causes the flow to behave closer to that of plug flow reactor and thus increase treatment efficiency. Ouedraogo et al (2016) also investigated the impact of water surface level on hydraulic performance. Findings show that an

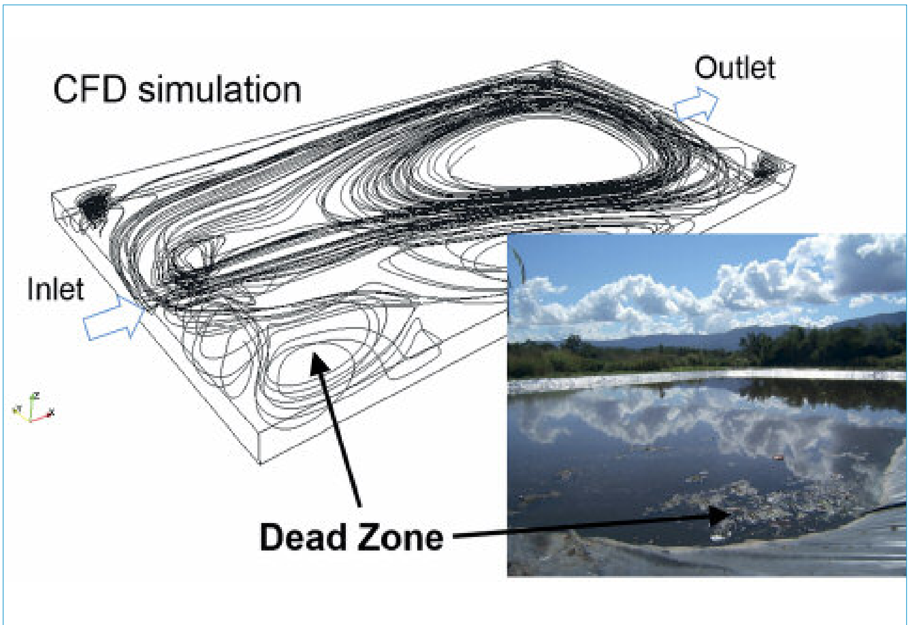


Figure 2. Dead zone in a wastewater stabilization pond, San Antonio, Bolivia modeled in Verbyla et al. (2013)

Jie Zhang **is a postdoctoral researcher and adjunct instructor in the Department of Civil and Environment Engineering at the University of South Florida, Tampa, Florida. He is serving as the chair in ASCE EWRI computational fluid dynamics (CFD) task committee. His current research interests include sustainable water and wastewater systems, modeling turbulence-chemistry interaction in incompressible flows, environmental flows in natural systems, and integrated modeling framework of CFD and sustainability assessment models.**

Qiong Zhang **is an associate professor in the Department of Civil and Environment Engineering at the University of South Florida. She has sponsored research projects in the areas of sustainability assessment and system modeling, with applications in water systems, wastewater and resource recovery systems, bioenergy systems, and the water-energy nexus. She has received a CAREER award from the NSF's Environmental Sustainability Program and her research has been supported by the Water Reuse Foundation, Florida Energy Systems Consortium, various NSF programs, and EPA programs.**

Andres Tejada-Martinez **is an associate professor in the department of Civil and Environment Engineering at the University of South Florida. At USF Tejada-Martínez has received a National Science Foundation (NSF) CAREER Award and various others NSF collaborative research awards for his work in Large Eddy Simulation (LES) of turbulent mixing in shallow shelf coastal regions and in the upper ocean mixed layer. He is also currently involved in various other computational research projects.**

Xiaofeng Liu **is an assistant professor in the Department of Civil and Environment Engineering at Penn State University. He is serving as Associate Editor to the Journal of Hydraulic Engineering, ASCE and vice-chair of the ASCE EWRI computational CFD task committee. His research interests include environmental fluid mechanics, sediment transport and erosion control, land surface process and morphodynamics, multiphase flow, water quality modeling.**

increase in water level while keeping a constant flow rate can result in a significant decrease in the hydraulic performance by reducing the sludge baffling effect, suggesting the need for careful monitoring of sludge accumulation and water surface level in WSP systems.

Anaerobic digestion

Anaerobic digestion is a method for treating the solid waste in wastewater. It can also assist in reducing water pollution by decreasing the concentration of organic matter in the waste. In addition, it can produce biogas for cooking, heating water for buildings, and generating electricity. Polyethylene tubular anaerobic digesters are commonly used in household-scale applications to recover energy from livestock waste in the developing world. These systems do not require a high level of skilled labor to install; they are easy to operate, low in cost and can operate under a range of temperature conditions (Kinyua et al 2016). However the studies on the physical and biochemical processes in tubular anaerobic digesters are rare.

To further understand these processes and consequently optimize digester design and operation conditions, Kinyua et al (2016) investigated the performance (biogas production and effluent quality) of a tubular digester treating livestock waste in the Monteverde region of Costa Rica through experimental studies and combined CFD and bioprocess modeling. CFD simulation was used to visualize the transport and mixing mechanisms in the digester. Based on the CFD analysis, a reduced-order model was developed for estimation of hydraulic and mean cell retention time, providing a practical tool for analyzing the performance of tubular anaerobic digester.

Perspective

CFD applications in wastewater engineering have become more prominent due to the advancement of computing power and numerical methods. However comparing to the success of CFD in other engineering areas (e.g. aerospace engineering and mechanical engineering), the use of CFD techniques for wastewater engineering is still limited. Several limitations are summarized as follows: 1) limited knowledge about the physical, chemical and biological processes, e.g., multi-phase turbulence, flocculation, and sludge rheology. The improvement of CFD modeling for wastewater treatment depends on the progress made in these sub-disciplines; 2) the lack of and uncertainties within the measurement data. Validation is critical for the fidelity of computational results. Lack of high quality and publically accessible data to validate computational models hinders the adoption of CFD models and to some extent erodes the confidence in the results. A community-based effort, to collect, sort, and evaluate existing data and establish a validation case pool, will be beneficial; 3) limited modeling techniques. For example, the modeling of the interaction between chemical reactions and turbulence in water flows is still a grand challenge; 4) The last but not least limitation comes from the steep learning curve of CFD and the requirement for the modeler to have comprehensive knowledge on a wide spectrum of disciplines, such as fluid mechanics, multi-phase flow, physical and biochemical processes, numerical methods, and computer science. The inclusion of introductory and advanced CFD modeling in undergraduate and postgraduate curricula will definitely help. Recently an ASCE EWRI CFD committee has been formed to promote CFD applications in water and wastewater engineering. Some of the activities planned by the committee include developing a primer of CFD applications in water and wastewater engineering and organizing a related technical session at the World Environmental and Water Resources Congress in West Palm Beach, Florida in May 2016. ■

References

Kinyua, M. N., Zhang, J., Camacho-Céspedes, F., Tejada-Martinez, A., & Ergas, S. J. (2016). Use of physical and biological process models to understand the performance of tubular anaerobic digesters. Biochemical Engineering Journal, 107, 35-44.
Liu, X. and García, M.H. (2011). Computational Fluid Dynamics (CFD) Modeling and Optimal Design of Large-Scale Primary Settling Tanks. Journal of Hydraulic Engineering, 137(3):343-355.
OpenFOAM Version 1.7.1 [Computer software]. (2010). OpenCFD Ltd., Bracknell, U.K.
Ouedraogo, F., Zhang, J., Cornejo-Warner, P., Tejada-Martínez, A. E., Zhang, Q., Mihelcic, J. R. (2016) Impact of Sludge Layer Geometry on Hydraulic Performance of the Waste Stabilization Pond in Bolivia. Water Research, 99, 253–262.
Verbyla, M. E., Oakley, S. M., Lizima, L. A., Zhang, J., Iriarte, M., Tejada-Martinez, A. E., & Mihelcic, J. R. (2013) Taenia eggs in waste stabilization pond systems with poor hydraulics: concern for human cysticercosis?, Water Science and Technology, 68(12), 2698-2703.
Zhang, J., Tejada-Martínez, A. E., & Zhang, Q. (2014) Developments in Computational Fluid Dynamics-Based Modeling Framework for Disinfection Technologies in Last Recent Two Decades: A Review, Environmental Software & Modeling, 58, 71-85.
Zhang, J., Tejada-Martinez, A. E., & Zhang, Q. (2016). Rapid Analysis of Disinfection Efficiency through Computational Fluid Dynamics. JOURNAL AWWA, 108, 1.

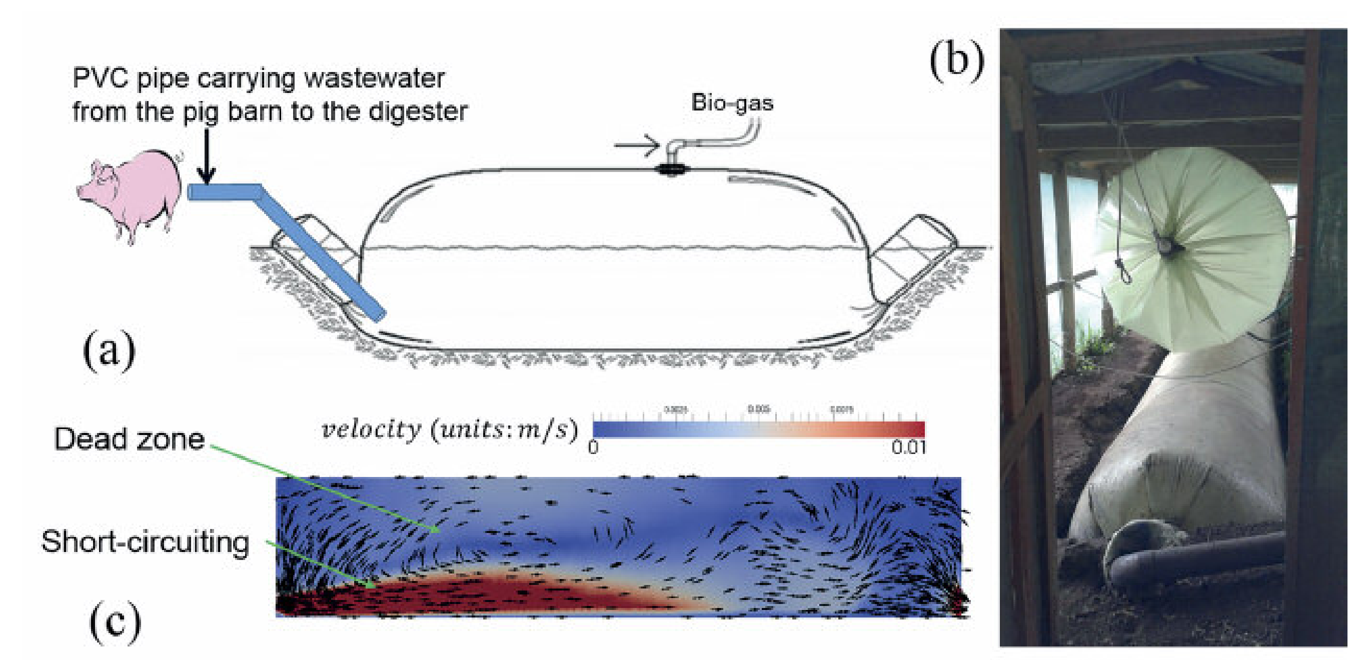


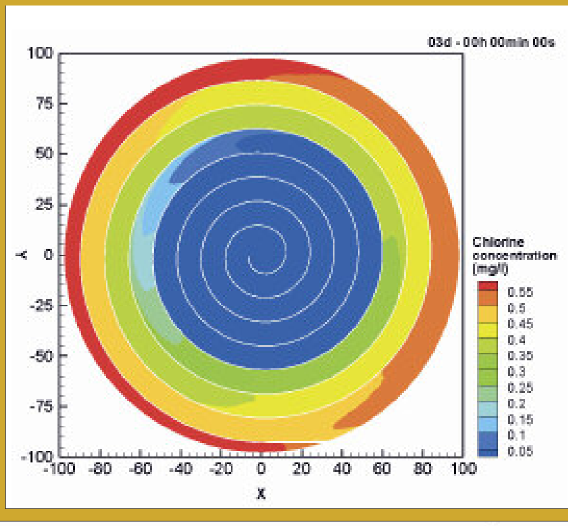
Figure 3. (a) Schematic, (b) photo, and (c) simulated internal flow of the anaerobic digester in Costa Rica studied by Kinyua et al. (2015)

NEW TRENDS IN MODELING APPLIED TO WASTEWATER TREATMENT AND POTABLE WATER

BY OLIVIER BERTRAND, JULIEN SCHAGUENE, BERNARD MAZAUDOU & PATRICK SAUVAGET

Modern and environmentally developed cities use large sewage network systems to collect and transport all types of wastewater from homes to wastewater treatment facilities. At the treatment plant, different processes are used to remove most of the pollutants from the wastewater. The efficiency of the treatment depends not only on the degree of purification applied to wastewaters, but also on the ability of the facilities to treat the effluent homogeneously and without temporal variations.

Artelia, as an independent engineering, project management and consulting group, operates in nine markets including industry, water, environment and urban development, providing services to private clients (industrial groups, developers, investors, building contractors, etc.) as well as to public clients (government departments, local authorities, international funding agencies, etc.). This article provides a brief overview of Artelia's use of modelling in support of the design of water storage, transfer structures, treatment installations and the development of solutions for decontaminating wastewater.



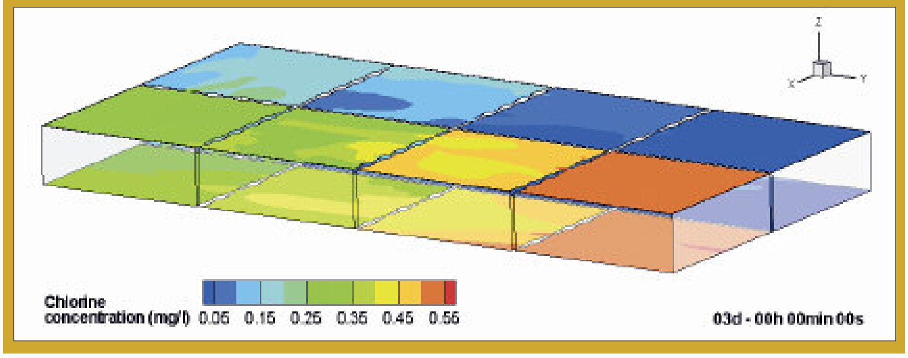


Figure 1. Simulation of water renewal within a circular/rectangular storage mega reservoir

From simple to more complex model
Most of the structures involved in water treatment or storage are large, but their performance is affected by small details in their design (orifices, distribution weirs, baffles, etc.) and is highly sensitive to the definition of the water surface, a few centimeters of water surface variation having significant influence.

This can be illustrated by the following project (Figure 1) the aim of which was to design an alternative back-up system (reservoirs, pumping stations and interconnection with the existing network) in order to secure continuous water supply. This involved a total storage capacity of 17 Mm^3 broken down into 5 storage sites. This system was susceptible to be used in case of sea water pollution or major power shortage for a country where water supply currently relies on desalinated water source plants. Mega Reservoirs structures were designed in order to achieve a 7-Day strategic water stock at any time within the network system. The dimensioning of the reservoirs was one of the main aspects of the project.

This objective was achieved thanks to a 3D non-hydrostatic numerical model with advection-diffusion of a conservative tracer to quantify the renewal time, elaborated on the basis of open source TELEMAC modelling system. The flow was free surface only. Various baffle geometries and positions of orifices were simulated and compared to minimize detention time in the water domain and to avoid recirculation or stagnant areas. Internal baffles were placed in the water storage tanks in order to direct and control the flow of water and to reduce water stagnation.

Interacting physical and numerical models
Physical and numerical modelling approaches are very complementary in the global study of the hydraulic structure of a water treatment plant and for many projects we performed comparison of both tools.

Physical scale models remain an incomparable resource when it comes to the analysis, communication and discussion of most complex development projects. Their experimental and practical nature provides guidance for engineers to understand various phenomena and help them determine high-performance solutions to manage projects in full compliance with specific requirements. They make it easier to explain phenomena by presenting the existing situation and changes to be expected once the project is implemented. Physical

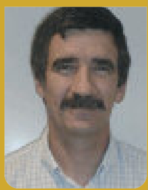
models allow visualization and quantification of flow and solute transport by directly simulating these phenomena.
On the other hand, mathematical models replicate these physical processes through mathematical governing equations, boundary conditions, and initial conditions. Mathematical models and computer simulations are also essential to describe, predict and control the complicated interactions of the processes. The number of reactions and organism species involved in wastewater treatment may be very large. An accurate description of such systems can therefore result in highly complex models, which may not be very useful from an operational point of view. Numerical modelling has the advantage of allowing flexibility in selecting problem parameters, being this way better suited for predicting results under different scenarios.

As an example of numerical application, water treatment plants are generally classified by implemented processes. Treatments could be physical (filters, settlers…) and/or chemical (coagulation, biological…). The use of ozone in waste water as chemical treatment is expanding. It is an accepted fact that drinking water is disinfected when a residual of ozone has been maintained for few minutes. The design of ozone generators and treatment chambers can be improved with the use of modeling.

In this application case (Figure 2), a model was set up and used to assess the behaviour of the ozonation tower. The entire tower was represented in the model and ozonated air diffusers positioned at the apron were included in the model. Two-phase modeling thus allowed simulating both the fluid phase and the gas phase. No exchange between the phases was considered. The rise of gas bubbles tended to block the water flow along the upstream walls of each compartment, but the whole mass of water was in contact with the gas phase. The ozonated air transfer to water to be treated was more important for the diffusers positioned upstream of the compartments.

Open source software and a distributed computer environment
With the explosive growth of mesh discretization there is a shift from the use of single desktop computers to using computer clusters. At the same time the use of Open Source software (as OpenFOAM) is gaining

**After having obtained general training in fluid mechanics and transfers, *Olivier Bertrand* developed numerous computation codes in such varied fields as crystallogenesis, compression of petroleum fluids, oceanic and river modelling, transport of oil slicks. As Project Manager/Director, he participates in particular in expert appraisal and complex hydraulic. He provides his experience in hydraulic development and its impact on the environment at different levels, from simple advisory services to design and innovative modelling studies. In particular, he runs and coordinates research and development projects associated with these problems.
In February 2015, he became Leader of the Numerical Hydraulic Modelling team and Project Director in the Maritime business unit.**



***Bernard Mazaudou* participated in the development and maintenance of the CAREDAS computer model for simulating unsteady flows in urban sewerage networks, which constitutes the hydraulics module of the CANOE system that has been co-developed since 1992 by Artelia and INSA (engineering college in Lyons, France). He also manages complex hydraulic studies for major water and waste water transmission schemes and pumping facilities, covering both steady and unsteady flow conditions. He is currently a project manager for water and wastewater studies within the Water & Solid Waste - International Activities business unit of Artelia Ville & Transport.**

Following a general training in fluid mechanics, *Julien Schaguene* has specialised in numerical modelling applied to the mechanics of unconfined surface flows. Julien Schaguene is a hydraulic modelling project manager in the fields of maritime, river and structural engineering. In this framework he is developing particular skills in the implementation of three-dimensional numerical models, notably using the CFD OpenFOAM computation code.

***Patrick Sauvaget* has 36 years of professional experience in the field of numerical modeling applied to the water environment. After obtaining an engineering degree in Hydraulics (INPG, Grenoble, France), he passed a master's thesis in civil and environmental engineering (University of Iowa, USA) and a PhD thesis in fluid mechanics (INPG, Grenoble, France). He is presently head of the Hydraulics department of Artelia Eau & Environnement, Grenoble, France. He acted as project leader or project director of water and environmental studies in various domains: hydraulics and water quality in rivers, coastal hydrodynamics, flood risk management, water resources management, water distribution, etc.**

Contact:
olivier.bertrand@arteliagroup.com

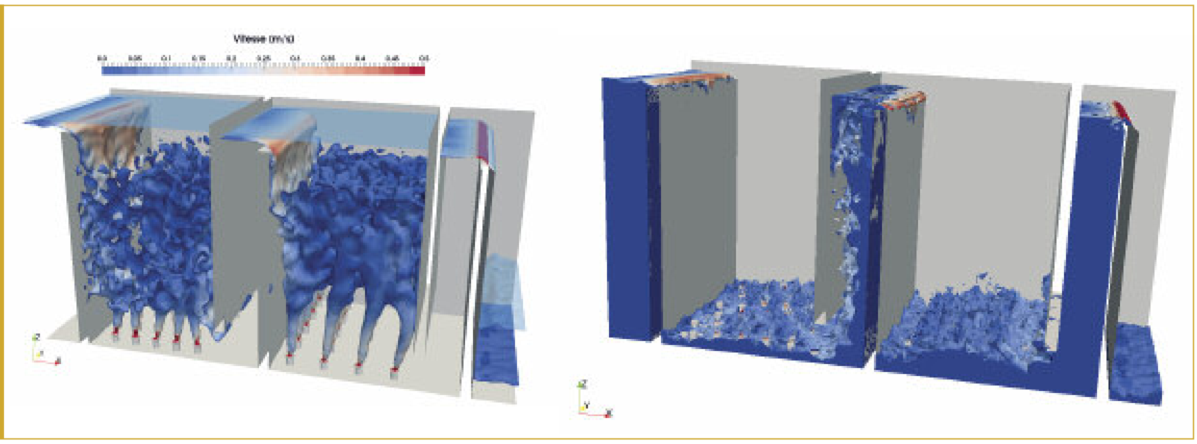
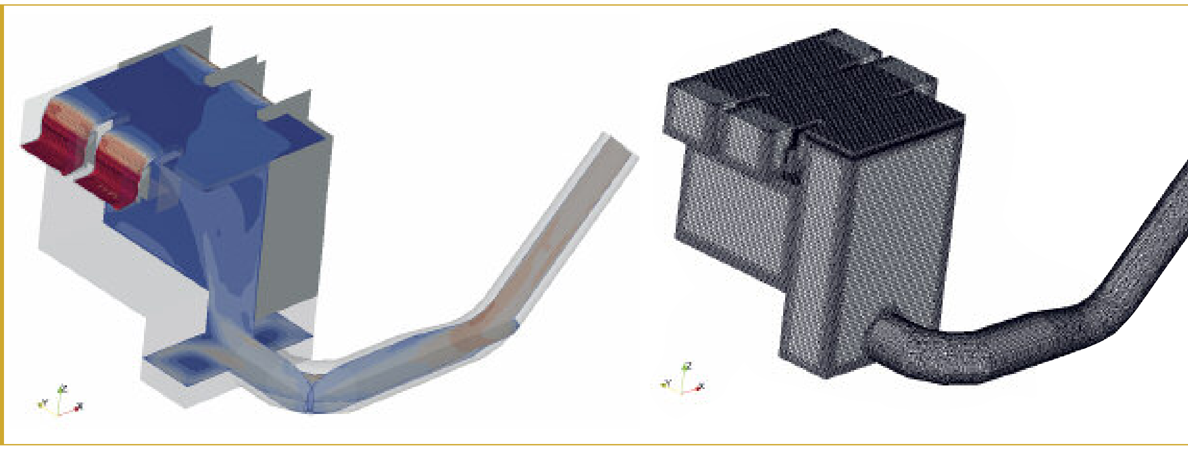


Figure 2. Analysis of the correct mixing in an ozonation tower (left: plume of water/ozonated air (99% water) and free surface; right: water volume out of contact with the ozonated air

Figure 3.
Flow analysis of an upstream distribution chamber



popularity as it provides free access to computer programming codes allowing experienced users to modify equations, to program new processes, or to optimize numerical solvers. These choices allow in the end to build numerical models totally adapted to the engineering problem to be solved.

These computational hardware and software facilities made it possible for instance to study the flow in a distribution chamber with weirs which is used to avoid unequal distribution (Figure 3). This kind of geometry required the computation of a mixed free surface and confined flow. The Volume Of Fluid (VOF) method that was used is simple, but it allows very complex free surface tracking. In this case the mesh (spatial discretization) was refined near the walls, at the free surface and in potentially highly turbulent areas.

For all these studies, expertise on core business and Computational Fluid Dynamics is extremely important, as well as improvements in the speed of scientific computations. They facilitate the use of more complex models, produce better designs by the multiplication of tests, reduce the time of the development phase and at the end the costs of new projects. ■

References

Bertrand O., Lavedrine A., Rebai A., David E., Pierrefeu G., Roumieu P. 2011 – Numerical modelling of the Vaugris reservoir on the Rhône with Telemac 3D. Proceeding of the XVIIIth Telemac & Mascaret User Club, Chatou, France, October 19–21.

Daou, M. P., Blayo, E., Rousseau, A., Bertrand, O., Tayachi, M., Coulet, C., & Goutal, N. 2015. Coupling 3-D Navier Stokes and 1-D Shallow Water. La Houille Blanche, (6), 29-33.

Loisel, P.E., Schaguene J., Bertrand O., Guilbaud C., 2015. Comparative study between numerical and physical hydraulic modelling of flood spillway structures. ICOLD 25th Congress, Stavanger, Norway, June 17-19, 2015.

HYDRAULICS OF WASTEWATER TREATMENT

USING CFD TO OPTIMISE UNIT TREATMENT PROCESSES IN A WASTE WATER TREATMENT WORKS

BY DARRELL EGARR & DAVID BURT

Introduction

Computational Fluid Dynamics can be used to generate a computational model of a treatment unit in order to calculate the process performance. The advantage of a computational model developed during the design stage is that optimisation can be undertaken by assessing either geometric changes or process conditions.

In its most basic form, the Activated Sludge Process (ASP) comprises a reactor, in to which waste water continually flows, followed by a settling tank to remove the biomass. Some of the biomass is surplus and removed from the system and some is returned back to the reactor. A waste water activated sludge plant includes several operational stages where the hydraulic behaviour is key to achieving optimum efficiency. This includes distribution chambers, anoxic zones and Final Settlement Tanks (FSTs).

Distribution Chambers

In a particular study, due to bed rock, a fairly shallow distribution chamber was designed, which meant that the treatment flow entered through the side of the chamber rather than from underneath as in conventional designs. Initially it was proposed that the Return Activated Sludge (RAS) was introduced at the centre. In order to calculate the flow distribution, a free surface model was used, which resolved the fluid-air interface. The boundary conditions specified at the outlets ensured that none of the weirs were drowned and hence, each was under free discharge. In order to model the activated sludge, the algebraic slip model was used with a hindered settling velocity defined by the Takács [1] equation. A rheology model was used to represent the increase in apparent viscosity with increasing solids concentration. The Stirred Specific Volume Index (SSVI) was conservatively set to 80 mL/g, which represents a good settling sludge and which is therefore harder to mix.

Figure 1. Solids concentration contours and solids distribution before and after optimization

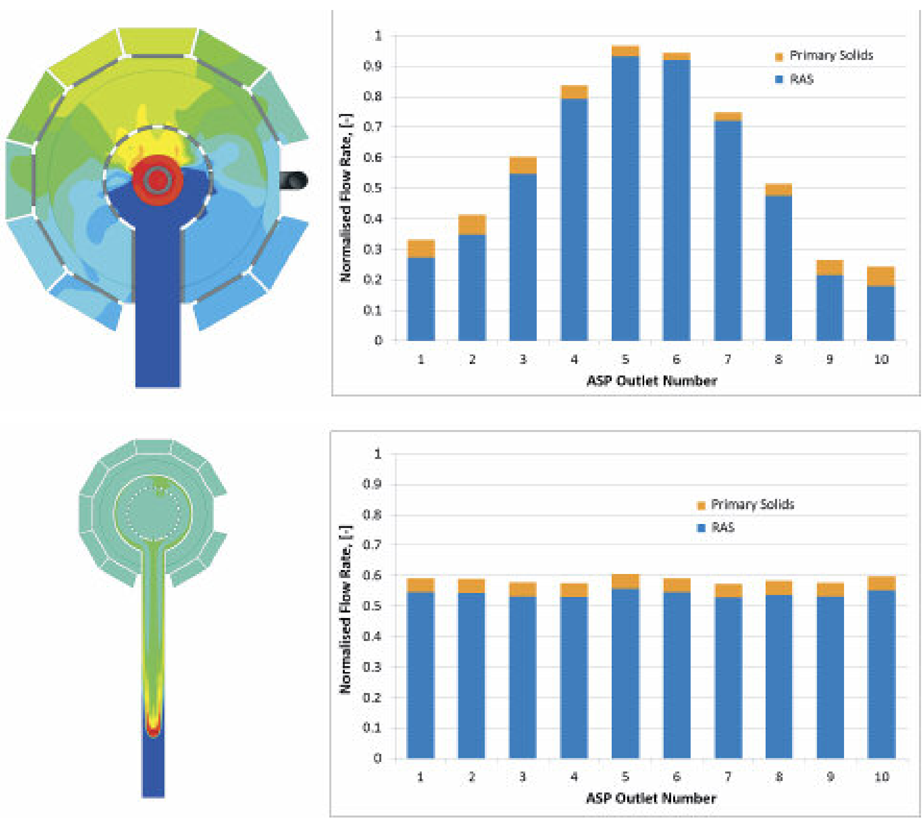


Figure 1 presents the load distribution and the contour plot shows that, due to poor mixing of the RAS, there was an uneven distribution with RAS biased to one side of the chamber. With the RAS inlet repositioned to introduce the RAS into the channel upstream of the chamber, the turbulence within the channel promoted effective mixing, and a good distribution of both flow and solids load was achieved, as shown in Figure 1. It should be noted that good distribution of flow and solids load is important at all stages of treatment. If, for example, there is poor distribution to the FSTs, there may be loss of sludge blanket at higher loadings.

Anoxic Zones

In the ASP, the reactor is typically sub-divided into discrete pockets in which the biological processes can be described as aerobic, anoxic or anaerobic. In anoxic zones, there should be

Dr Egarr **is a Senior Engineer in MMI Engineering, applying Computational Fluid Dynamics to analysing and optimising waste water treatment processes. Today his work involves managing the technical delivery of projects and supervision of staff. He continues to undertake research and development and has recently worked on the development of a solver for modelling the flocculation of particulates.**

Dr Burt **is a Principal Consultant with over 20 years of experience in the application of hydraulic and fluid dynamic modelling codes. He has developed and applied several multi-phase models in the CFX code including, boiling, condensation free surface tension and drift flux settling. He has applied these CFD techniques to the analysis of systems across industry sectors, including Water, Nuclear, Aerospace and Defence.**

no dissolved oxygen. This encourages micro-organisms to consume oxygen bound in the nitrates, resulting in denitrification and the release of nitrogen gas. Thus, amongst other requirements, it is usually intended that in an anoxic zone, there is no short circuiting of the flow between inlet and outlet.

An example of an anoxic zone is presented in Figure 2(a). In the original design there were two successive high level weirs, in blue labelled Weir (1) and Weir (2), each at entry and exit to the anoxic zone. To identify short-circuiting of the flow, a 'dye trace' experiment was replicated. The advantage of using CFD for dye tracing is that very low concentrations can be measured over very long time scales, which is not easily achieved by similar physical experiments at full scale. Additionally, the average concentration can be monitored over the region of interest, in this case at Weir (2), whereas experiments are limited to monitoring at point locations. The result of the dye trace is presented in Figure 2(b), which shows a significant short circuit. This is shown in the initial concentration which is very high, followed by very low concentration, indicating that the majority of the dye exits very quickly.

In order to improve the design, baffles were introduced in the anoxic zones, as shown in red in Figure 2(a), to divert the flow to low level, therefore generating a longer flow path between the inlet and outlet weirs. The flow paths are annotated in Figure 2(a) with the blue dashed arrow indicating the short-circuit in the absence of the baffle, and the longer flow path with the red dashed arrow which is generated with the baffle. With the baffle, although the dye trace does not show perfect mixing behaviour within the first 0.5 residence times, thereafter, the dye trace curve does follow that of a Completely Stirred Tank Reactor (CSTR), and therefore shows a significant improvement in the hydraulic behaviour.

Final Settlement Tanks

After the aeration lane, the waste water undergoes secondary treatment in a clarifier. The activated sludge settles and the effluent passes over a side weir.

Due to the quiescent environment within a FST, effective design of the influent arrangement is key to controlling the hydraulics around the influent to minimise interaction with the sludge bed. Effective inlet design achieves uniform distribution of the flow into the FST and dissipation of the inlet momentum.

As an example, Figure 3 presents views from a series of models of circular FSTs. The first is where the tank design incorporates a stilling drum only. The second includes a McKinney baffle, which is a horizontal baffle below the stilling drum that feeds the flow into the tank in a radial direction at a fixed elevation. The right most view presents a design incorporating an

Figure 2.Anoxic zones and residence time distributions

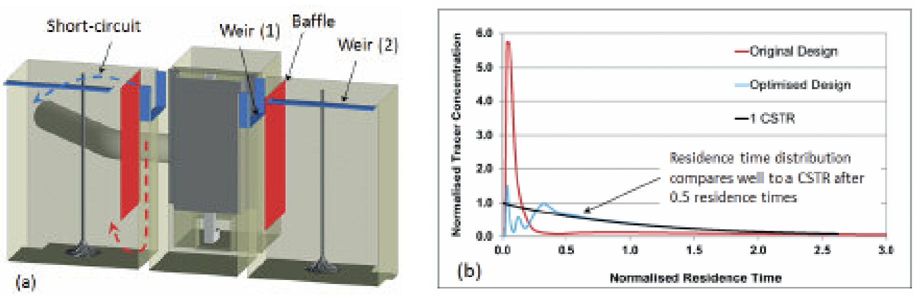
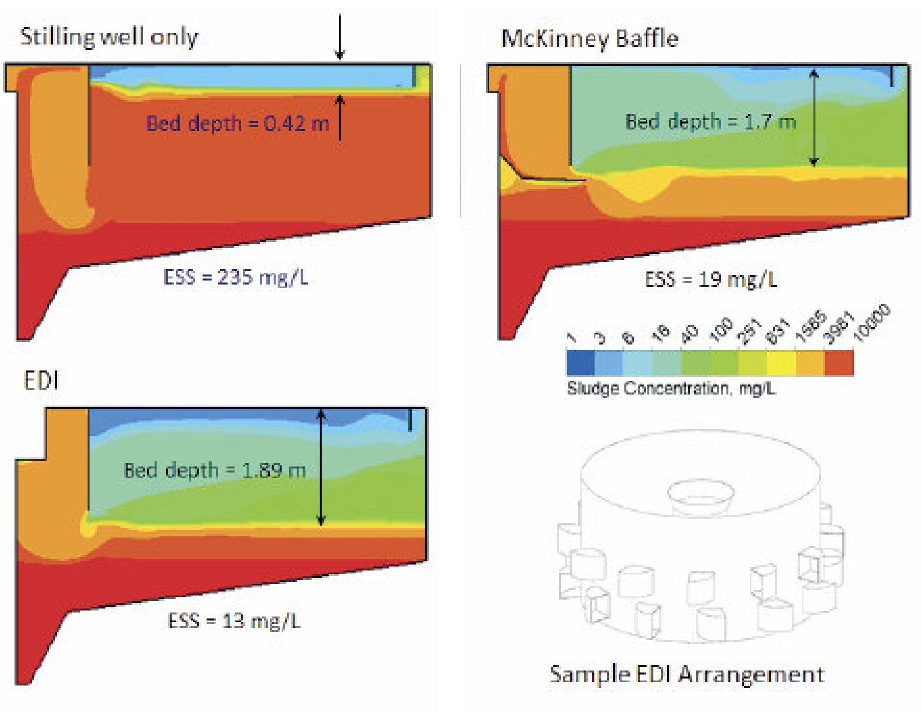
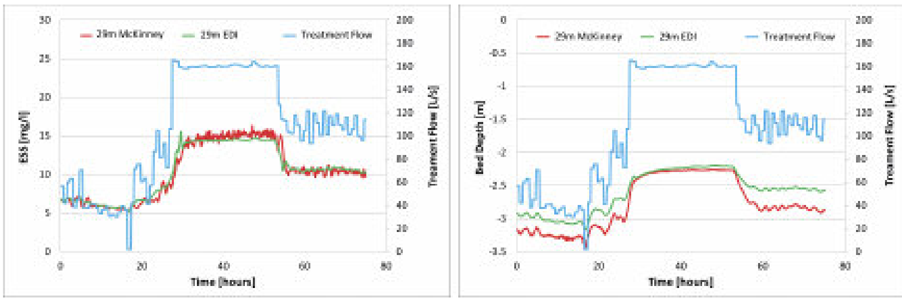


Figure 3. Comparison of tank performance for different influent designs



Energy Dissipating Influent (EDI) which is a drum in to which the influent flows and incorporates a number of ports to swirl the flow on exiting, which assists in dissipating the influent momentum. A schematic of an EDI is presented in Figure 3. Although the analysis was undertaken for steady state conditions, the results clearly demonstrate that, for appropriate influent design, significant improvements in tank performance can be achieved. In analysing the position of the sludge bed, the contour at around 900 mg/L is assessed and it can be seen in Figure 3 that at around this concentration, there is a significant concentration gradient, indicating that this is the approximate position of the sludge bed where a high concentration fluid-solid mixture exists below this contour and a dilute supernatant above. In the latter two designs (McKinney and EDI), there is a significant clearance between the sludge bed and the effluent weir. This not only assists in reducing effluent solids concentrations, but also generates resilience of the tank to surges in flow such as storm events. As an example, Figure 4 presents a hypothetical storm surge received by a FST. In this case, the influent concentration was held constant although in reality, there may be some dilution of the solids. Figure 4 also presents the effluent concentration and the position of the sludge bed over the duration of the storm. (with 0 m being Top Water Level). This shows that, although there was an increase in Effluent Suspended Solids (ESS) concentration during the storm, loss of sludge blanket over the weir did not occur. ■

Figure 4. Response of effluent concentration and sludge bed during storm surge



References

1. Tákacs, I., Patry, G.G., Nolasco, D., 1991, A dynamic model of the clarification-thickening process, Water Res., 25(10), pp. 1263-1271.
2. Vesilind, P.A., "Theoretical considerations: design of prototype thickeners from batch settling tests.," Water & Sewage Works, vol. 115, pp. 302-307, 1968.

COMPUTER SIMULATION OF LIQUID-SOLIDS SLURRIES FOR WASTEWATER TREATMENT

BY L. JOEL PELTIER, KELLY J KNIGHT, BRIGETTE ROSENDALL, SANJEEB PAL, ANDRI RIZHAKOV, ANDREI SMIRNOV & CHANTHY IEK

Applied Simulation and Analysis; Bechtel Nuclear, Security, and Environment; Reston, Virginia 20190

Processing of liquid-solids slurries for wastewater treatment involves handling of dissolved solids and undissolved solids with readily suspended to rapidly settling behaviors. Given a significant loading of dissolved or readily suspended solids, the effective carrier-fluid rheology may exhibit complicated non-Newtonian effects. A simulation-based assessment of wastewater treatment requires a sophisticated computational fluid dynamics (CFD) flow code with submodels sufficient to address this potentially diverse range of physics. Reynolds-Averaged Navier-Stokes (RANS) models are the current workhorse.

Simulation is always limited by available computational resources and physics parameterizations. With advances in computational engineering in parallel processing environments and physics submodel development for computer simulation codes, many limitations are either being removed or are being moved to higher-order details. CFD-RANS models are now able to meet challenges for simulating liquid-solids slurry flows in complicated configurations.

Industrial wastewater may contain a significant fraction of undissolved solids with potentially broad particle size and density distributions. Granular-Eulerian multiphase modeling is an example of a CFD-RANS technology that has been formulated to handle this kind of application. In a Granular-Eulerian multiphase model, each gas, liquid, or solids constituent is treated

Figure 1. Multiphase mixing in an industrial process vessel: Flow (left), Velocities (Right)

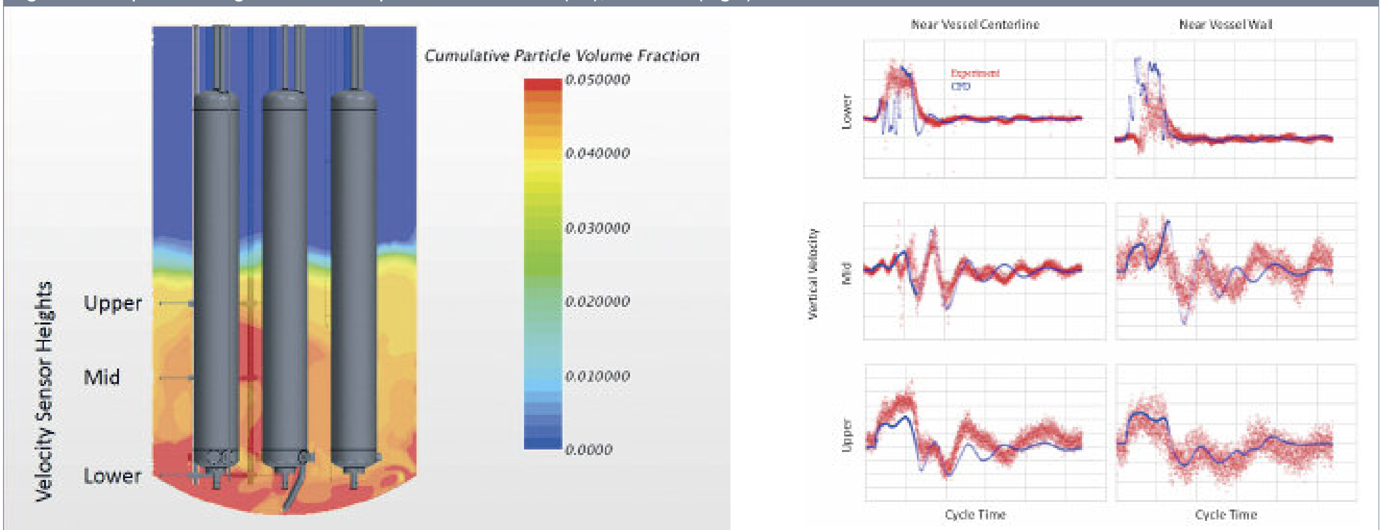
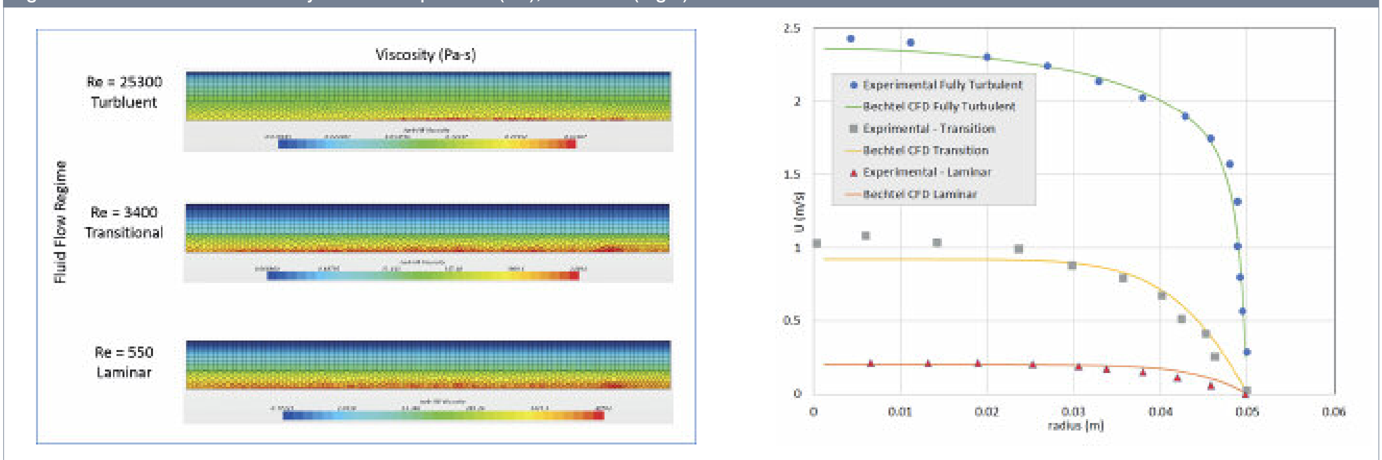


Figure 2. Flow of a Herschel-Bulkley Fluid in a Pipe: Flow (left), Velocities (Right)



as a separate continuous modeling phase. Submodels are used to parameterize interactions and behaviors at boundaries.

Comparisons of computational results (from a commercial CFD code CD-adapco/Star-CCM+) to experimental data show the fidelity that can be achieved. Figure 1 (left) is an instant from a simulation of mixing of a polydisperse loading of undissolved solids in a Newtonian carrier fluid. The mixing is performed in a vessel prior to the next step of the treatment process. The total solids loading in the vessel is 10% by weight. Approximately half of the solids are readily suspended. The upper part of the vessel is gas. The particle distribution is characterized by 6 solids phases with representative particle sizes and densities. The simulation presented in this article models an existing physical model experiment of the mixing of the waste in the vessel. The simulation geometry is derived from a CAD model of the experimental apparatus. In both the simulation and the experiment, time histories of velocity are sampled at six points in the bulk flow with the velocity sampling locations at lower, mid, and upper levels. Three locations provide velocities near the vessel centerline. Three locations provide velocities near the vessel outer wall. Comparisons of the CFD-RANS predicted velocities to the experimental data, Figure 1 (right), confirm model fidelity to real-world physics.

Dissolved and undissolved readily-suspended solids in industrial mixing vessels and other liquid-solids slurries may be modeled using an effective fluid rheology and density. Contemporary CFD solvers include a broad range of rheology submodels, a non-Newtonian Herschel-Bulkley fluid being an example.

In a Herschel-Bulkley fluid, the apparent viscosity of the fluid depends on the local shear rate. In regions of high local shear rates, a Herschel-Bulkley fluid behaves like a Newtonian fluid. As local shear rates reduce, a Herschel-Bulkley fluid becomes more viscous. The local shear rate in a turbulent flow occurs in the dissipation range of turbulence. CFD-RANS solutions provide energy-containing-range (mean-field) statistics, not dissipation range statistics. Without an appropriate model linking local shear rates to mean-field statistics, CFD simulations of Herschel-Bulkley fluids are well defined only for laminar flows where the dissipation range can be resolved explicitly.

A method to extend CFD modeling of Herschel-Bulkley fluids into the turbulence regime was recently presented at the Star-CCM+ Global Conference (Peltier et al, 2016). This model extension uses turbulence theory to estimate representative local maximum shear rate magnitudes from CFD-RANS data enabling simulations of Herschel-Bulkley fluids in the turbulence regime.

Figure 2 (left) shows CFD predicted viscosities for flow in a pipe of a Herschel-Bulkley fluid in the laminar, transitional, and turbulence regimes. The slice shown is from the pipe centerline to the upper outer wall. Comparisons of the CFD predicted velocities to experimental data, Figure 2 (right), confirm model fidelity to real-world physics.

The examples shown for simulation of liquid-solids slurries underscore that capabilities of contemporary commercial CFD flow codes are rapidly advancing and support a conclusion that a simulation-based assessment of wastewater treatment is possible with an expectation for fidelity to real-world physics. ■

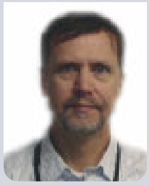
References

Peltier, J., Rizhakov, A., Rosendall, B. Inkson, N., and Lo, S., 2015: Evaluation of RANS Modeling of Non-Newtonian Bingham Fluids in the Turbulence Regime using STAR-CCM+®, STAR-CCM+ Global Conference Location: San Diego, CA, March 15-18, 2015.

L. Joel Peltier **is a Principal Engineer and manager of Computational Fluid Dynamics (CFD). He specializes in applied fluid mechanics, CFD, and numerical heat transfer applied to nuclear waste treatment and industrial processes. He is a member of ASME and Vice-Chair of the ASME Verification & Validation 20 Code and Standards Committee.**

Andri Rizhakov **is a Senior Engineer in Computational Fluid Dynamics. He specializes in nuclear waste treatment, multiphase flows for industrial applications, and analytics work via machine learning paradigms. Andri's leadership activities include team guidance involving writing of CFD quality process procedures. He is a member of the ASME.**

Kelly J Knight **is a Principal Engineer and manager of Advanced Simulation and Analysis (ASA). Kelly specializes in simulation of fluid and structural mechanics including modeling of blast protection, structural stress/strain and fatigue, multiphase flows, shock and blast wave events and thermal heat transfer. She is a member of the ASME.**



Andrei Smirnov **is a Senior Engineering Specialist in Computational Fluid Dynamics. He specializes in design of nuclear reactors, nuclear waste treatment, multi-phase flow analyses for industrial applications, and combustion software development. He is a member of the ASME and the ANS and a holder of one US patent.**

Brigette Rosendall **is a Computational Fluid Dynamics Specialist for Solar Turbines in San Diego. She specializes in solving problems involving multi-phase flow, combustion, radiation, species transport, free-surface, and moving boundaries. She is a member of the AIChE.**

Chanthy Iek **is an Engineering Specialist in Computational Fluid Dynamics. He specializes in shock wave dynamics and radiation/boiling/particulate deposition for chemical processes. His engineering experiences include CFD, turbomachinery, aero-acoustics, and air-breathing and rocket propulsion. He is a member of the ASME.**

Sanjeeb Pal **is a Senior Engineering Specialist in Computational Fluid Dynamics. He started working for Bechtel Corporation in 2008. His work in the ASA group is focused on Computational Fluid Dynamics application to address engineering challenges on Bechtel EPC Projects. He is a member of the ASME.**

A FLIGHT SIMULATOR FOR WASTEWATER AND DRINKING WATER TREATMENT FACILITIES

DYNAMIC MODELLING OF OPERATIONS

BY JASON CURL

Figure 1. A 3D rendering of a municipal drinking water treatment plant

Computational power continues to increase and data are prolific. The best method for leveraging both of these is combining them within a single platform to: (1) increase understanding of design conditions to inform engineers, (2) improve operators' understanding as to how the system performs, and (3) test worst-case scenarios in a realistic simulator. The Replica™ platform, developed by CH2M, accomplishes all of the above and has been successfully applied on dozens of projects around the globe.

Achieving Multiple Goals under Many Constraints

Variability of water availability, reduced capital and operational budgets, and tightening regulations all combine to present real challenges to our water conveyance and treatment infrastructure. We strive to improve operational efficiency and ensure the highest water quality levels at all times to meet these challenges. The issues are not new, but the stakes seem to be higher in recent years due to the 24-hour news cycle and potential public relations concerns. "Big data" is a widely used term to describe large, complex data sets that present difficulty to analyze, digest, and use. Now, new tools are available to make use of big data to address the water supply challenges facing the industry. Dynamic simulation software leverages powerful computational capabilities and is founded on fundamental hydraulics, controls, and water quality calculations. With Replica™ software, we can make use of the abundant data that exist so we can fully understand the changing water quality, hydraulics, and controls in an operational facility.

Combining simulation power, abundant data, and the ability to evaluate long-time series of data, Replica™ truly operates like a flight simulator for the full-scale facility. The benefits of the software are many, including the ability to:

(1) test extreme water quality or flow scenarios in a safe environment, (2) evaluate new control schemes, (3) train operators, and (4) improve communication among operators, engineers, and utility management with the intuitive interface.

Dynamic Simulation of Hydraulics, Controls, and Water Quality

Robust dynamic simulation, which integrates hydraulics, controls, and process treatment, allows for improved design coordination, more efficient solutions, improved system under-standing, and optimization of operations. CH2M's dynamic simulation model, Replica™, can be applied at various phases of a project, which can start with a theoretical model during design and can be calibrated during the opera-tions phase.

Replica™ models are typically set up for one-second time steps. Each hydraulic component (e.g. pipes, tanks, open channels, pumps, etc.) are modelled individually and linked together in the model to pass information between blocks. All types of water and wastewater unit processes have been successfully modelled with the system. Examples include: settling basins, membrane filtration, granular media filtration, screening, aeration basins, mixing basins, disinfection chambers, and dewatering processes. Figure 2 shows a Replica™ model overview of a drinking water treatment plant. This model includes the following:

- Raw water pump station for water conveyance
- Flash mix facility to rapidly mix chemicals
- Sand ballasted clarification to settle out solids
- Ozone contactors for disinfection
- Granular media filtration for particle and pathogen removal
- In-plant pump station for water conveyance

During the design phase of a project, the dynamic simulation model can be used to confirm equipment sizing and selection, evaluate system pressure and gravity hydraulics, and develop fundamental control strategies and preliminary control set-points. Further evaluation and optimization can be performed on the design for various parameters such as operating costs, power consumption, chemical usage, water loss, and process control tuning set-points.

During the commissioning phase of a project, a dynamic simulation tool, such as Replica™, can aid control system testing and operator training. The process control logic programmed in the actual programmable logic controllers (PLCs) being installed in the field are tested and tuned against the hydraulic simulations in the model prior to field installation to minimize programming changes in the field. Next, the model can also be linked to the supervisory control and data acquisition (SCADA) software to provide a realistic "flight simulator" of the system for operator training. The benefit is that

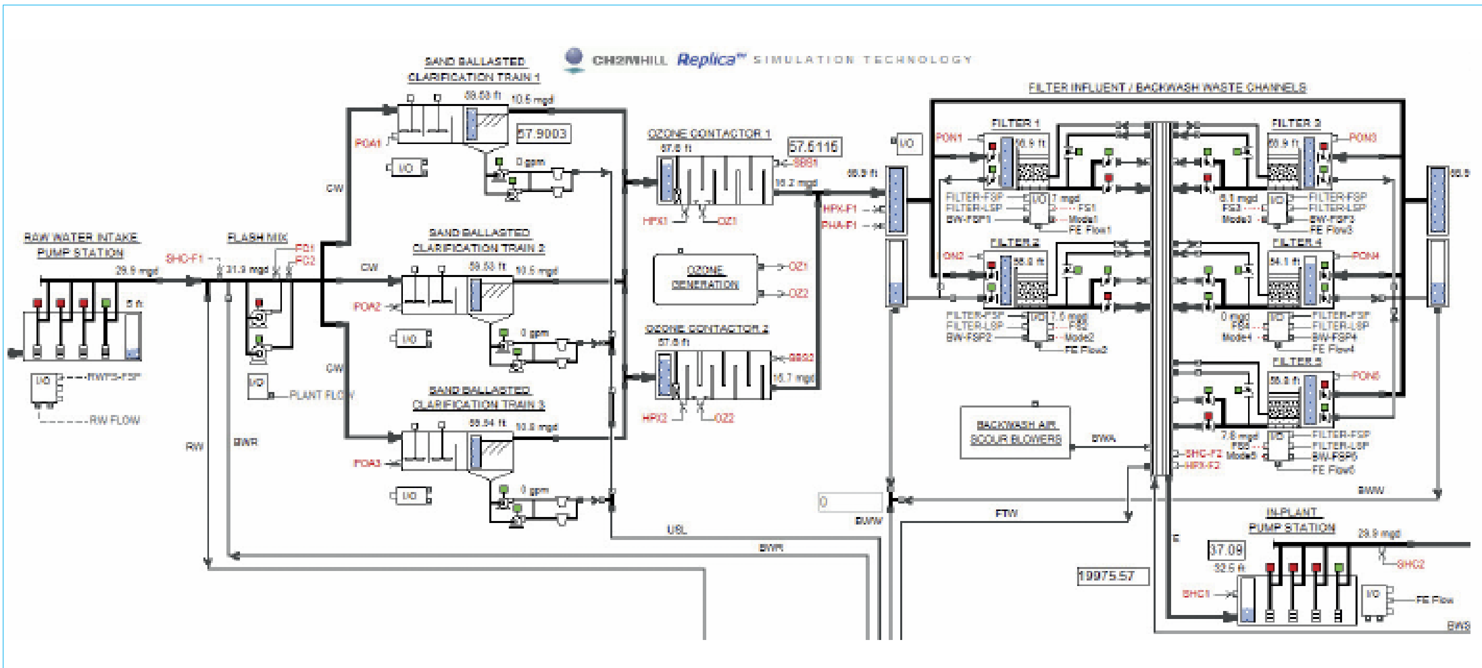


Figure 2. The Replica™ model's process flow diagram interface is intuitive to navigate

the operator can interface with the real SCADA system and PLC controls, but the hydraulic performance is being simulated in the Replica™ model. This operation provides operators insight in operating the system under simulated hydraulics and water quality conditions with the actual PLCs, prior to actual facility operations. Finally, but maybe most importantly, a dynamic simulation model can be used after a project has been in operation to evaluate and optimize operations since "what if" scenarios can be tested in the simulator environment. The dynamics of the system, including changing water quality, recycle flows, pump operations, and hydraulics, are all modeled so the system operation feels real time. The process flow diagram interface improves communication to help capture operations knowledge. Additionally, new operator training is further enhanced with a realistic facility simulator such as this.

Successful Project Applications of Dynamic Simulation Models

Replica™ has been built from over 30 years of hydraulic modelling, process design, and process controls knowledge, in a fashion that is both flexible and complies with industry standard calculations, and can be customized for a specific application. It has been used on over 100 projects of varying sizes around the world consisting of water conveyance and distribution, water treatment, wastewater treatment, and wastewater collection systems. Some example project applications are described in the following paragraphs.

Project 1

Project 1 took place at a municipal wastewater treatment plant (WWTP), which has a daily flow capacity of 340 mega liters per day (MLD) and a peak wet weather flow of 1,660 MLD. An intermediate pump station (IPS), including three 2,250-horsepower pumps, sends flow to secondary treatment, which is permitted to treat a maximum of 1,100 MLD. All additional flow above 1,100 MLD bypasses secondary treatment at the flow diversion structure via two weir gates and travels directly to the chlorine contact basin for disinfection. The WWTP was being cutover from its existing control system to a new control system. The dynamic simulation model was used to analyze and optimize the hydraulic performance of the IPS to alleviate operational issues associated with its small volume wet-well and impacts on water levels in the flow diversion structure. Using the model, CH2M simulated and tested various flow conditions for various scenarios, including pump failure, pump start/stops, and pump sequencing. To further ensure a robust control scheme, the team simulated 12 months of historical flow through the IPS to assess energy consumption reduction with the optimized control algorithm.
Based on using a robust model to simulate hydraulics and controls, CH2M reduced wet-well deviation from level set-point during pump transitions both in the Replica™ model and real-world application of the new control algorithm. As a result of the strong understanding achieved with the model application, there was a smooth cutover from the existing control system to the new.

Figure 3. Process performance can me predicted live in SCADA systems based on historical performance data

Compliance Reporting - Sheet 1

PARAMETER	COMPLIANCE PERIOD	COMPLIANCE VALUE	ENG UNITS	ALLOWED VARIANCE	Current Value	High Alarm
Combined Filter Effluent Turbidity	Monthly Average	0.15	NTU	95%	NTU	96%
Filter 1 Turbidity	Monthly Average	0.15	NTU	95%	NTU	96%
Filter 1 Turbidity	15 Minute Average	0.30	NTU	15 Minutes Until Next Sample		N/A
Filter 2 Turbidity	Monthly Average	0.15	NTU	95%	NTU	96%
Filter 2 Turbidity	15 Minute Average	0.30	NTU	15 Minutes Until Next Sample		N/A
Filter 3 Turbidity	Monthly Average	0.15	NTU	95%	NTU	96%
Filter 3 Turbidity	15 Minute Average	0.30	NTU	15 Minutes Until Next Sample		N/A
Filter 4 Turbidity	Monthly Average	0.15	NTU	95%	NTU	96%
Filter 4 Turbidity	15 Minute Average	0.30	NTU	15 Minutes Until Next Sample		N/A
TTHM	Monthly Average	0.04	mg/L	0	mg/l	> 0.03
HAA5	Monthly Average	0.03	mg/L	0	mg/l	> 0.025
Nitrosamines	Monthly Sample	0.00	ng/L	0		> 6
Total Manganese	Monthly Sample	0.02	mg/L	0		> 0.01
Total Iron	Monthly Sample	0.24	mg/L	0		> 0.2
Total Aluminium	Monthly Sample	0.16	mg/L	0		> 0.12
Thiobencarb	Monthly Sample	0.001	mg/L	0		> 0.0006
MIB	Agency Request	5.00	ng/L	0		> 3.00
Geosmin	Agency Request	5.00	ng/L	0		> 3.00
pH	Daily Average	7.00	pH	+ / - 0.2	pH	+ / - 0.18
LSI	Monthly Average	0.00	LSI	+ / - 0.1	LSI	> 0.08
Phosphate	Weekly Average	0.10	mg/L	+ / - 0.2	mg/l	+ / - 0.18

***Jason Curl* is a principal technologist with CH2M's Water Business Group and has been based in Denver, Colorado, for more than 13 years. Jason has bachelor's and master's degrees from the University of Illinois and is a licensed professional engineer in the States of Colorado and Texas. He is CH2M's Global Service Leader for Software Applications and Integration, leading a team of engineers who develop software that enhances analysis and delivery of solutions to our clients' most challenging projects. He has a wide variety of process engineering experience on water treatment, membrane technologies, and reuse projects as well as implementation of software to solve client challenges.**

Project 2

Project 2 is a 115 MLD greenfield river intake, raw water pump station, municipal drinking water treatment plant, and high service pump station project that used Replica™ to aid design, startup, and operations. The model was used throughout the design phase to facilitate coordination among design staff. Control strategies, which included full consideration of both hydraulics and controls, were modeled to assist development of control narratives. Water quality was also included in the dynamic simulation model to predict facility performance to meet stringent contractual requirements on more than a dozen water quality parameters. The predictive water quality performance was proven in the dynamic simulation model and then included in the design of the plant SCADA system. Now the operators receive water quality alarms that are predictive of full treatment facility performance in advance of the parameters actually falling out of contract compliance. Figure 3 is a screen capture of the operator interface for these predictive alarms. For example, the alarm for TTHM represents total trihalomethanes. This datapoint is continuously updated with a programmed predictive equation based on online measure parameters such as water temperature, total organic carbon, finished water turbidity, and free chlorine residual. ■

REAL TIME CONTROL OF SECONDARY CLARIFIERS – ENHANCING HYDRAULIC CAPACITIES DURING RAIN

BY ANDERS LYNGGAARD-JENSEN

Secondary clarifiers are usually the limiting factor for the hydraulic load on wastewater treatment plants (WWTP). This is specifically the case during rain, when the WWTP is located downstream a combined sewer system. The hydraulic capacity, Q_{biomax}, of the WWTP is given by the sludge settling velocity multiplied by the secondary clarifier area. Therefore, efficient real time control of the secondary clarifiers can increase the hydraulic capacity of the WWTP during rain by increasing the sludge settling velocity.

Introduction

The sludge settling velocity, V_{sed}, decreases exponentially with increasing suspended solids concentration, SS, in the inlet to the clarifier. In order to increase the sludge settling velocity the suspended solids concentration to the clarifiers therefore has to be reduced. This article describes a methodology where the clarifiers themselves are used as a well-controlled sludge storage, in order not to return all the sludge flushed to the clarifiers during start of rain to the process tanks again immediately.

The designed and implemented controller is combined of a feed forward control for balanced (minimum) return sludge flow and a feedback control of the distribution of the return sludge flow between clarifiers. The feed forward control is based on an estimate of V_{sed} (therefore also calculates Q_{biomax}) and uses a flux balance to calculate the minimal return sludge flow, $Q_{r,min}$.

The feedback control uses sludge blanket measurements, SB (the level where sludge is separated from clear water), in clarifiers to distribute the pumping of $Q_{r,min}$, in order to make all sludge blankets follow each other. The feedback control therefore ensures full storage capacity, or in other words ensures that the entire clarifier area is equally useable, as the controller compensates for any skewness in the load distribution to the clarifiers.

Depending on the design of the wastewater treatment plant, i.e. if more secondary clarifier lines are present – it is possible to extend the controller to accommodate them by including a feedback controller using the average sludge blankets for the different lines to control the distribution of the sludge. A control handle (gate/weir) is thus required downstream of the process tanks in order to carry out the right distribution of the flows to the clarifier lines. The controller selectively flushes the sludge to a specific clarifier line when the flow at the treatment plant increases due to rain. This procedure increases the hydraulic capacity of the WWTP almost instantly, and secures the controller does not need any lead time for handling a fast increasing flow.
Finally, in dry weather situations, the clarifiers are controlled to give a more stable performance, and thereby give a higher suspended solids concentration in the return sludge, by using the optimal and balanced return sludge flow $Q_{r,min}$.

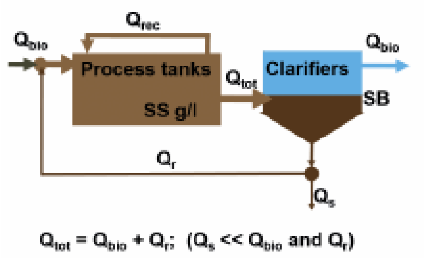


Figure 1. Overview of the biological treatment and the used nomenclature

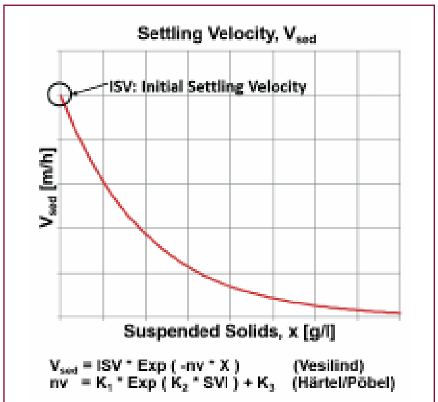


Figure 2. Sludge settling Velocity, V_{sed}

The controller is in operation on several WWTPs, including Marselisborg WWTP operated by Aarhus Water, Denmark, which is used in the examples below.

Sludge settling velocity

The relationship between the suspended solids concentration in the inlet to the clarifiers and the sludge settling velocity in the clarifiers can be represented by the well known Vesilind exponential equation (Vesilind, 1968) – see figure 2, with the exponent being equal to the suspended solids concentration multiplied by a factor, *nv*. The factor *nv* is a function of the Sludge Volume Index, *SVI*, which is a sludge characteristic, and therefore only changes slowly with the sludge age of the WWTP.

Several functions for the factor *nv* have been suggested. Here the function suggested by Härtel and Pöbel is used, as the determined constants (K_1, K_2 and K_3) do not change much from WWTP to WWTP.

The *SVI* can be determined in the laboratory as the sludge volume after 30 minutes, *SV30*, divided by the suspended sludge concentration in the sample. In the equation for the sludge settling velocity the exponential is multiplied by a factor known as the Initial Settling Velocity, *ISV*, which is, like *SVI*, a sludge characteristic. *ISV* can be determined in the laboratory from a measurement of V_{sed}, but it can also be estimated in real-time by a proper control of the clarifiers in dry weather.

Clarifier State Diagram

The operation of clarifiers can be described using a state diagram (figure 3) consisting of the functional relationship between fluxes of suspended solids in the clarifiers and the suspended solids concentration. The flux is given by the amount of sludge crossing a square meter of the clarifier per unit time. The settling flux, can be expressed as the suspended solids concentration multiplied by the settling velocity, where the settling velocity is described using the Vesilind equation.

In the Clarifier State Diagram the area below the settling flux curve describes the area for stable operation of the clarifier, i.e. the state point has to be located under the settling flux curve. The state point is defined as the point where the returned flux, equals the upward flux. The suspended solids concentration in the sludge arriving to the clarifiers (the same as the *SS* in the process tanks), can be read on the x-axis vertically under the state point.

Optimal return sludge rate

Usually values for the return sludge flow, Q_r, are calculated as a percentage of the inflow to the WWTP. Typically too high percentages are chosen, despite the fact that besides using too much energy for pumping, it also gives a varying and too low suspended solids concentration in the return sludge. This again can lead to an unnecessary use of polymer in the pre-dewatering of the surplus sludge taken from the return sludge.

In the Clarifier State Diagram the returned flux is a straight line having the slope $-Q_r/A$, and it crosses the x-axis at the concentration of the suspended solids in the return sludge, SS_r. As the line is fixed to the state point, manipulating Q_r will make the line turning around the state point – increasing Q_r the line will become more vertical, and the SSr will decrease – decreasing the Q_r the line will become more horizontal and the SS_r will increase.

Increasing the Q_r too much will thus cause the clarifier to move towards a fully mixed tank and the sludge blanket will disappear. Decreasing the Q_r too much, making the returned flux line cross the settling flux curve, will cause the clarifier to function as a thickener, and the clarifier will fill up with sludge causing the sludge blanket to move towards zero.
The optimal Q_r (which also will be the minimal Q_r) will thus be obtained where the settling flux just equals the returned flux (figure 4) – in this point the two fluxes will also have the same slope, and therefore the first derivatives will also become equal. Solving the equations for the fluxes and their derivatives then gives the optimal value for Q_r, which then is used in the feed forward controller. Further, because the optimal Q_r balances the fluxes it also creates a steady sludge blanket.

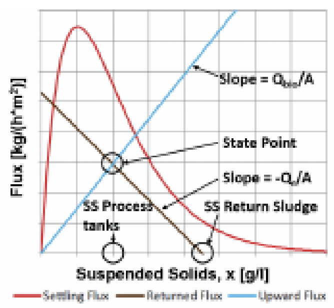
Dynamic maximum hydraulic load

The upward flux is a straight line passing through the origin and having the slope Q_{bio}/A. Therefore, an increase in Q_{bio} will cause the line to become more vertical and move the state point upwards (still located vertically over the value for *SS* in the process tanks). When the state point reaches the curve for the settling flux, the upward velocity in the clarifier equals the settling velocity (figure 5), and the hydraulic load of the WWTP has reached its maximum, Qbiomax. The clarifier is overloaded and the sludge blanket will move upwards – no matter the size of Q_r - and eventually sludge will escape directly into the effluent or the next step of the WWTP (e.g. a sand filter, which will clog). However, if the sludge from the process tanks is allowed to flush to the clarifiers – keeping Q_r at its minimum – *SS* will decrease in the process tanks, as it will be stored in the clarifiers. Therefore, *SS* in the inlet to the clarifiers will decrease causing Vsed to increase. In the Clarifier State Diagram (figure 5) this will be reflected by a movement of the state point to the left – sliding down the line for the upward flux – also causing the controller to decrease Q_r, and giving the hydraulic load, Q_{bio}, the possibility to increase further. The result is that a controlled flush of sludge to the clarifiers and a subsequent controlled storage of the sludge in the clarifiers will increase the maximum hydraulic load, Q_{biomax}, during rain.

Anders Lynggaard-Jensen **(M.Sc. Chemical Engineering, Technical University of Denmark) is Chief Engineer for Automation and Processes at DHI in Denmark. He has for more than 35 years worked with instrumentation, control and automation related to water and wastewater. The work has included development of sensors, software for real time data processing and control algorithms. His research and consultancy work is focused on process optimization on wastewater treatment plants incl. biological Nitrogen and Phosphorous removal, clarifier hydraulics and integrated control of combined sewer systems and WWTPs during rain.**

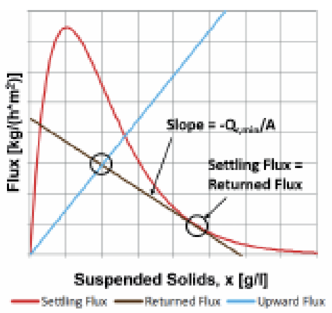
Distribution of return sludge rates between clarifiers

Most WWPTs have several secondary clarifiers or even more lines of clarifiers including several secondary clarifiers (figure 6). In practice it is a well-known problem that equal distribution of the load to the single clarifier (or line of clarifiers) sometimes through manually operated weirs in the distribution constructions is impossible to obtain – the weir positions (if any) are dependent on the hydraulic conditions in the distribution constructions. This in fact reduces the overall hydraulic capacity of the WWTP as such, because the clarifier with the highest load during



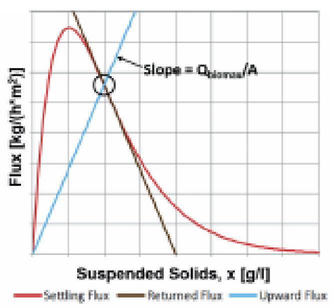
$$F_{settling}(X) = X * ISV * Exp(-nv * X)$$
$$F_{returned}(X) = ((Q_{bio} + Q_r) * SS - X * Q_r) / A$$
$$F_{upwards}(X) = X * Q_{bio} / A$$

Figure 3. Clarifier State Diagram



$$Q_r = ISV * A * (nv * X - 1) * Exp(-nv * X)$$

Figure 4. Clarifier State Diagram – $Q_{r,min}$



$$Q_{biomax} = V_{sed} * A \text{ or}$$
$$Q_{biomax} = ISV * Exp(-nv * SS) * A$$

Figure 5. Clarifier State Diagram – Q_{biomax}

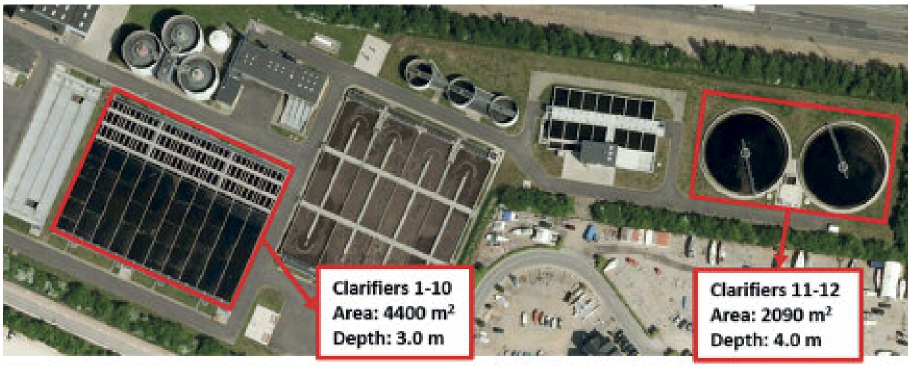


Figure 6. Marselisborg WWTP, Aarhus, Denmark is equipped with two clarifier lines

rain will produce the fastest growing sludge blanket level, and therefore set the limit to the inlet flow of the WWTP in order to avoid sludge washout.

In other words the operator experiences a too low value of Q_{biomax} compared to the dimensioned hydraulic load, and as this has nothing to do with V_{sed}, it can be compared to a loss of clarifier area, A, as the effective area is less than the built area. This can be quite a serious problem – an effective area as low as 2/3 of the built area has been observed. The straightforward solution could be real time control of all inlet weirs – if weirs exists, but on most treatment plants that will require investment in automatic weirs. Instead it is possible to compensate the skew distribution of the clarifiers using dynamic control of the distribution of the overall required return sludge flow, to the single clarifier based on each clarifier's sludge blanket level, SB, compared to the average value of the levels of all sludge blankets, SB_{Avg}.

The feedback controller aims at equalizing the sludge blankets in the clarifiers (having the same area and depth) to the same value in order to distribute and control the sludge storage capacity. The hydraulic load is not distributed, but the load variations, (measured as the variations in the sludge blankets) are compensated by higher or lower return sludge flow values from each of the clarifiers.

At the Marselisborg WWTP with 10 clarifiers in one line (figure 6), an equal distribution of the hydraulic load to the clarifiers in this line would require that the return sludge flow from each tank should be 10% of the calculated setpoint for Q_r, called Q_rSP. The controller adds or subtracts to this percentage for each clarifier keeping Q_rSP at the same value. The resulting return sludge flow values for each clarifier are sent to local control loops for the pumps or valves in each of the clarifiers.

The resulting movements of the sludge blankets are shown in figure 7 (left) together with the percentwise distribution of the return sludge pumping (right). The plots cover a period of two days, and the two peaks in the sludge blanket levels are caused by a small rain and a somewhat bigger rain respectively. It is clear that the distribution of the load to the 10 clarifiers is quite different in dry and wet weather, because the distribution of the return sludge pumping, which compensates for the skewness in the load distribution, is very different.

Distribution of load between secondary clarifier lines

Often more lines of secondary clarifiers at a wastewater treatment plant are a result of an investment in an extension of the WWTP in order to handle an increasing load. The extension of the secondary clarifier capacity is often done as an "add-on" exercise, and a proper distribution of the load between the old and the new clarifier line has to be done.

A control handle (gate/weir) is thus required downstream the process tanks, in order to control the flow to each of the clarifier lines. As the control handle is a part of the extension, it will typically work on the flow going to the new clarifier line (line 2), via a local control loop based on a measurement / set-point for the flow to the new clarifier line and divert the remaining part of the total flow to the old clarifier line (line 1).

It has to be remembered that the flow going to the clarifiers Q_{tot} is the sum of the inlet flow to the treatment plant and the return sludge flow, and as the control of the return sludge already is dependent on the inlet flow, Q_{tot} is as well. So any set-points applied to $Q_{tot,i}$ for each of the clarifier lines (i=1 and i=2) not only have to respect that $Q_{tot} = Q_{tot,1} + Q_{tot,2}$, but also have to respect the set-point to be calculated for the return sludge flow Q_r ($= Q_{r1} + Q_{r2}$).

In other words, the distribution of the flow to more clarifier lines has to be done in terms of the inlet flow, and the distribution shall reflect the ratio of the clarifier areas in the clarifier lines (as the possible load is dependent of the area).
In order to actively control the sludge distribution, the distribution according to clarifier areas is extended with a term including the sludge blanket averages: SB_{Avg1} and SB_{Avg2}, which then can be controlled to follow each other by compensating the area related return sludge flows with a percentage calculated from the distances of the sludge blanket average to the overall sludge blanket average for the WWTP. Calculating the overall sludge blanket average requires that the sludge blankets averages for each of the clarifier lines are comparable. For clarifiers of different depths the same sludge load per m^2 gives a comparable steady state sludge level, but as sludge blankets are measured from the top of the clarifier, they need to be compensated for the difference in clarifier depths. The clarifiers at the Marselisborg WWTP are 3 and 4 meters deep respectively, which requires an offset of 1 m on the sludge blanket average for clarifier line 2. This offset forms together with the measured sludge blanket average for clarifier line 2 a virtual sludge blanket, which can be directly compared to the sludge blanket average for clarifier line 1.

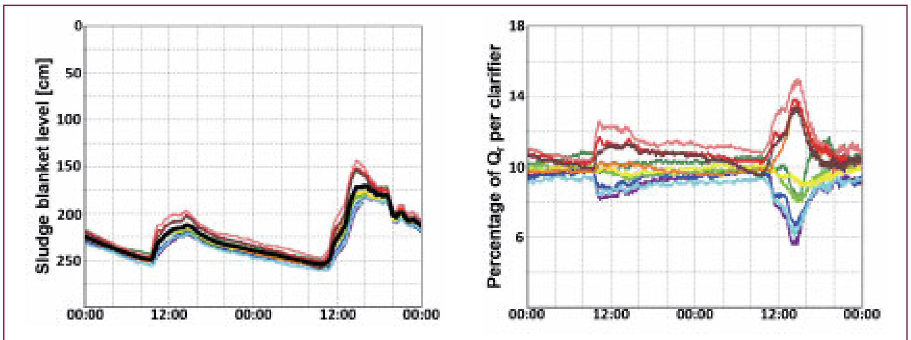


Figure 7. Marselisborg WWTP clarifier line 1 – Sludge blanket measurements resulting from the distributed return sludge pumping. Colors on plots follows the color spectrum – violet for clarifier 1 and dark red for clarifier 10. The average sludge blanket level is black

The controller is here described for two clarifier lines (at the Marselisborg WWTP), but obviously it can be extended to accommodate more clarifier lines. Sludge blanket offsets shall then be defined for each depth of clarifiers – the reference being the shallowest clarifiers.

Selective sludge storage control during rain

As the distribution of load between clarifier lines as described above is controlled by the average sludge blankets for each line, and as the average sludge blanket in the deeper clarifiers is virtual (measured sludge blanket level plus an offset), it is possible to force the load to a given line by manipulating the virtual sludge blanket (figure 8, bottom). When the inlet flow to the WWTP increases during the start of a rain, the offset can be set to zero.

When the offset is set to zero, the sludge blanket average in line 2 is suddenly increasing (note the inverted axis), which abruptly will increase the flow to the to the deeper clarifiers, as the controller will compensate a sudden increase in the average sludge blanket by a sudden increase in distribution percentage for line 2. As a result, sludge from the process tanks will be flushed to line 2, which will be heavily overloaded and therefore cause the average sludge blanket to decrease until the average sludge blankets (both being real) in the clarifier lines are balanced again. When it stops raining, the sludge blanket average in line 2 will again be virtual, and the average sludge blanket will suddenly decrease, which abruptly will decrease the flow the line 2, as the controller will compensate a sudden decrease in the average sludge blanket by a sudden decrease in the distribution percentage for line 2. As a result, the amount of sludge coming from the process tanks to the deeper tanks will be minimal, and the return sludge pumping will pump more sludge out of the clarifiers than arriving, and therefore cause the average sludge blanket to increase until the average sludge blankets (one real and one virtual) in the lines are balanced again. In figure 8 this is repeated at the next rain.

Combined value for max. hydraulic load

Figure 9 shows the same two rain events as in figure 8, and it can be seen that Q_{biomax} is increasing each time sludge is flushed to the clarifiers – from 1000 l/s to 1600 l/s and further on to 2800 l/s, which is far more than the WWTP can handle. The Qbiomax curve suggests that the settling velocity and the area are not the limiting factors.

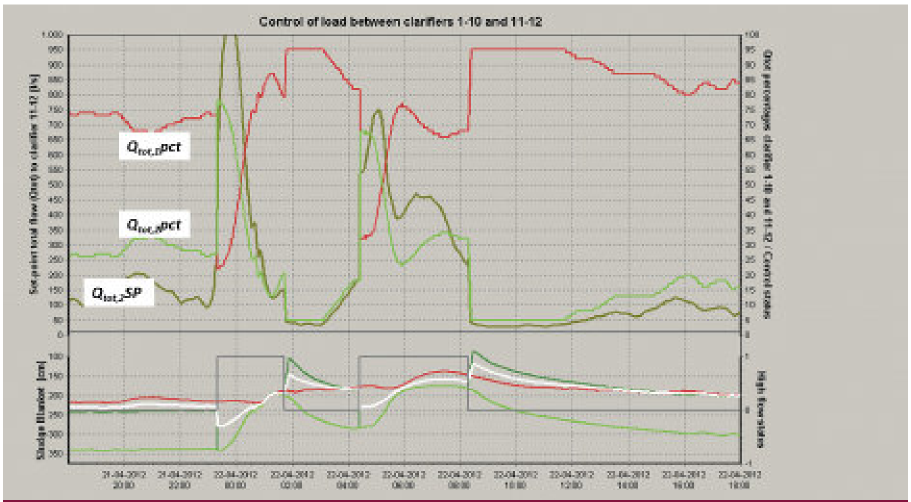


Figure 8. Marselisborg WWTP – sludge storage control using a virtual sludge blanket. Bottom: $SB_{Avg,1}$: red; $SB_{Avg,2}$: light green; $SB_{Avg,2\text{-}virtual}$: green; $SB_{Avg,WWTP}$: white

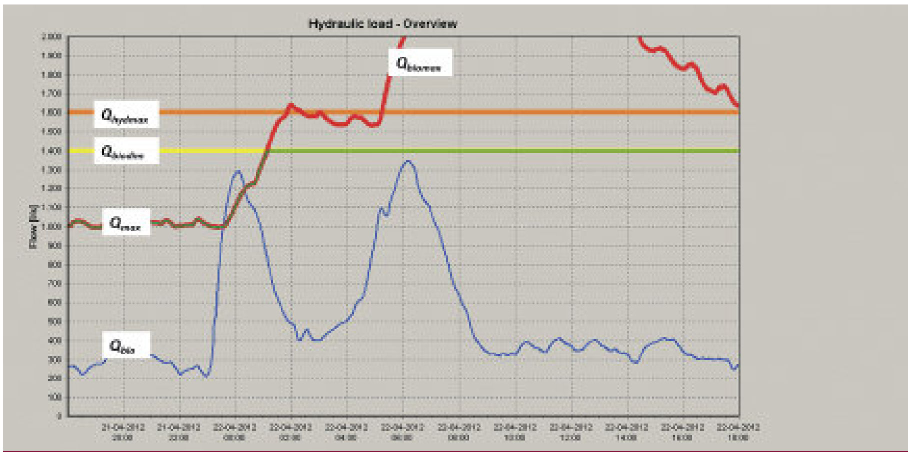


Figure 9. Marselisborg WWTP – maximum hydraulic load, Q_{max}

Figure 9 shows how the maximum hydraulic capacity, Q_{max}, is calculated as a "safety wrapped" value:

If $Qbiomax <$ selected limit then $Q_{max} = Q_{biomax}$

If $Q_{biomax} >=$ selected limit then $Q_{max} =$ limit

where the selected limit can be either the dimensioned maximum inlet flow, Q_{biodim}, or the absolute maximum flow, Q_{hydmax}, that the WWTP can handle due to internal limitations. The limit can be selected by the WWTP plant manager. As the figure shows, Q_{biomax} can be less than the actual dimensioned capacity – especially during winter, where suspended solids concentrations in the process tanks need to be quite high, in order to be able to comply with the effluent standard for total-Nitrogen. The idea of Q_{max} is to protect the wastewater treatment plant when needed (low Q_{biomax}) and on the other hand to put more than the dimensioned capacity into service, if Q_{biomax} shows it is possible (selected limit = Q_{hydmax}).

Conclusion

A combined controller for real time control of secondary clarifiers has been developed and implemented on several WWTPs, and is here demonstrated by presenting the function of the controller at the Marselisborg WWTP. The results show:

- Efficient control of secondary clarifiers makes it possible to increase the hydraulic load during rain considerably above the dimensioned hydraulic load
- The presented controller does not have any lead time – which often is the case for this type of controller
- The controller does not affect the operation and control of the upstream biological process ■

References

Härtel, L., Pöbel, H.J.: A dynamic secondary clarifier model including processes of sludge thickening. Wat.Sci.Tech. Vol 25, No. 6, pp. 267-284. (1992)

Lynggaard-Jensen, A.: Increased capacity of wastewater treatment plants during rain. Climate Change, Water Supply and Sanitation – Risk Assessment, Management, Mitigation and Reduction. ISBN 9781780404998, IWA Publishing, London, UK. pp 223-233 (2015)

Vesilind, P.A.: Design of prototype thickeners from batch settling tests. Water and Sewage Works. 114 (7) pp. 302-306 (1968)

Prince Sultan Bin Abdulaziz International Prize for Water

Recognizing Innovation

Invitation for Nominations

8th Award (2018)

Nominations open online until 31 December 2017

www.psipw.org e-mail: info@psipw.org

CLIMATE CHANGE
DRIVEN ACTIVITIES AT THE COASTAL AND MARITIME HYDRAULICS CENTER OF CEDEX, SPAIN

BY JOSE M. GRASSA, RAMON M. GUTIERREZ-SERRET, ANTONIO LECHUGA & ANA LLORET

Coasts and seas pose some of the most challenging problems in the global analysis of Climate Change (CC) key risks and adaptation needs. This article describes the activities of CEPYC-Centro de Estudios de Puertos y Costas *(Center for Coastal and Maritime Hydraulics)* of CEDEX-Centro de Estudios y Experimentación de Obras Públicas *(Center for Studies and Experimentation of Public Works)*, related to direct effects of CC or to growing CC-induced response of developments on coastal and marine systems. Specific problems of concern in Spain that drive the research efforts at CEPYC are described, as well as the research subjects, facilities and other infrastructure needed to support the society in dealing with these issues.

Issues related to direct effects of CC
One of the less-uncertain, direct effects of CC is Sea Level Rise (SLR). Sea level rise affects directly the overtopping of maritime protection structures and, in general, coastal flooding both in natural and heavily modified, man-made, coasts. A direct, high confidence effect of SLR is coastal erosion induced by beach profile adaptation to sea-level changes. Summarising, more frequent coastal flooding events and increased rates of sustained coastal erosion are two main effects of CC on coastal zones. Coastal flooding and erosion (figure 1) require coastal and maritime engineering expertise supported by field measurements and experimentation (numerical and physical) in applied studies and research and development (R&D).

With 46% of the Spanish population living within a short distance from the shoreline in low-lying land (figure 2), and with one of the world's largest coastal - oriented tourism industries that represents about 10% of the national GDP, there is high social and economic sensitivity to increased flooding risks and erosion trends. Coastal land use is changing with people moving from larger coastal cities to less dense, smaller cities, which results in linear coastal cities on long stretches of the Spanish coast. These urban developed coasts, lack most of the natural adaptive response of natural beaches to SLR, and demand significant defense and / or mitigation efforts that should be guided by science and technology. Another special issue is that most major commercial ports have been built on open coasts (figure 2), due to the lack of natural havens. A significant number of quays are located on the lee – side of breakwaters which increases the risk of overtopping due to SLR, and is becoming a major issue to be studied.

Figure 1. Coastal flooding and erosion

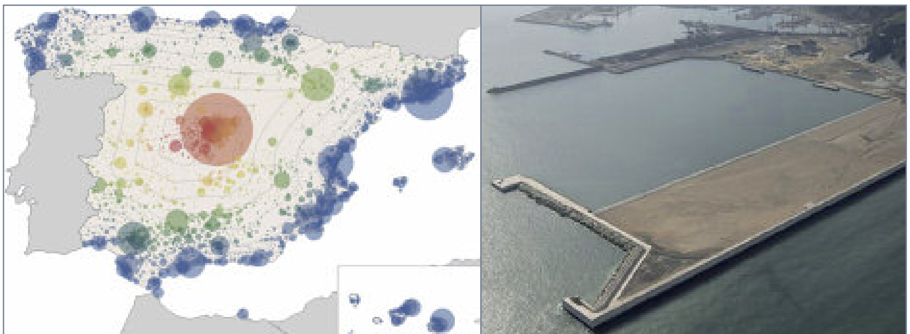


Figure 2. Spanish population density distribution and outer port of Gijón (Atlantic north coast)

Issues related to CC response
Adaptation measures to cope with CC involve new uses of the coasts and seas. CC-related plans for the reduction of fossil energy consumption and the reduction of greenhouse gas emissions 'fuel' the effort to develop different forms of marine renewable energies, such as offshore wind, tidal and wave energy. Also, more frequent and longer drought periods provide a motivation for the development of coastal seawater desalination plants. These innovative activities require expertise in coastal, maritime and environmental engineering and sciences supported by field measurements, laboratory characterization and experimentation The development in Spain of marine renewables involves mostly deep water applications, given the extremely short width of the continental platform in most coastal areas of Spain, which creates the need to advance knowledge in the operation of non-gravity based solutions for offshore wind, namely floating wind turbines. Also, due to the growing hydrological deficit mainly on the South East coasts of Spain, a major national desalination plan is under development, which requires the study of gravity currents, water quality and ecosystem impacts on the very important kelp prairies in the vicinity of the disposal points, as well as engineering measures to increase effluent dispersion in the near field.

Projects, Working Groups and Facilities
Coastal flooding and overtopping, SLR-induced coastal erosion, marine energy and desalination studies involve all the working units at CEPYC, in many cases within coordinated multidisciplinary efforts. Research and applied studies on overtopping (deep and shallow water structures) and coastal flooding, as well as on marine energy (waves and wind) make use of the major facilities in the Maritime Hydraulics Laboratory: the large-scale wind and wave flume, the short crested wave basin and, in the case of marine energy the ship simulators for the study of maritime operations for the deployment of

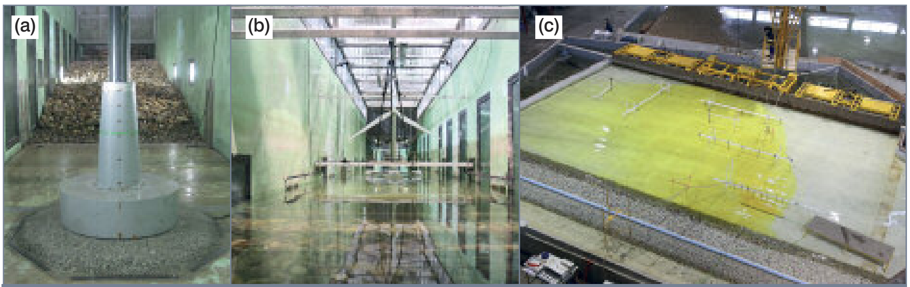


Figure 3. Physical model tests for (a) Scouring around the footing of offshore wind turbines; (b) Wind turbine test and (c) desalination plant effluent discharge

Figure 4. (a) Large wave and wind flume, (b) real time maneuvering simulation for transportation of a wind turbine foundation and (c) CFD model for wave forces on a wind turbine

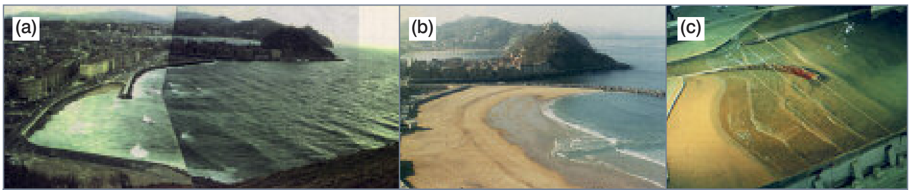


Figure 5. Zurriola beach. (a) in 1990; (b) in 1995; (c) physical model 1992

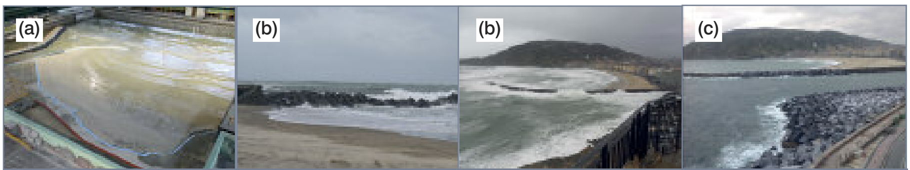


Figure 6. Zurriola beach. (a) Physical Model study (2013). (b) Damage in the 2013-2014 storms and (c) Situation in 2015 after repairs

equipment at sea (figure 4). Regional strategies for adaptation to SLR-induced coastal erosion including coastal defenses and beach nourishment are developed by the Coastal Engineering Department, based on field work and are supported by physical and numerical experimentation. Collaborative research projects, Environmental Impact Assessment (EIA) supporting studies and applied work on desalination are led by the Marine Environment Department and its Marine Environment Quality Laboratory with the support of other units in field work, physical and numerical modelling (figure 4).

A case study on the effects of future sea states. Zurriola beach damage and restoration

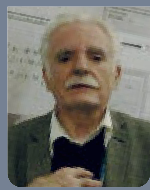
Zurriola Beach, located in the city of San Sebastian, is an urban beach built in the 1990's in front of the original barrier beach that was fully reclaimed by the city at the beginning of the twentieth century. The construction included an artificial replenishment with offshore sand and a low-crested rubble mound groin with a curved design for providing a uniform width beach from the 'hard' urban boundary (figure 5).

After construction, the beach, exposed to north-western Atlantic storms, suffered no significant damage for nearly 20 years, with the exception of some initial damage to the groin due to extreme events at the end of the last decade. Then in 2013, the Spanish Coastal Authority commissioned CEDEX to study with a physical model the stability of the slightly damaged groin and any needed repairs (Figure 6) and to check the monitored recorded trends of beach evolution.

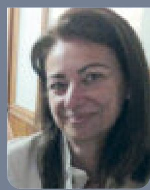
As the study was proceeding, a series of major storms attacked the beach during the winter of 2013-2014. The February 2, 2014 storm produced considerable damage to the groin, interestingly not in its more exposed deep water mouth, but in a shallower wave-height limited zone. While severe, the storm was not of an extremely high return period in terms of significant wave height; however, the significant wave periods were high, up to 20 s, and above all, occurred during extremely high tide with water levels reaching 4.5 m, therefore allowing the arrival of less depth-limited waves. The damage was reproduced well in the physical model and a repair scheme was subsequently developed and successfully tested (figure 6).

Another storm, with similar extreme water levels occurred on March 3, 2014. Both this and the February storm resulted in considerable damage to shallow water fishing harbours, urban beaches and promenades. These cases are seen as exemplifying typical situations which will occur in the future, producing damage to shallow water city coasts due to CC-induced SLR and which therefore will require detailed study.

Jose M. Grassa, Director, has been working at CEPYC for nearly 30 years. His personal research interests are coastal wave phenomena modelling and wave structure interaction.



Antonio Lechuga is the Director for Coastal Engineering. He is a coastal researcher interested in all phenomena and events related to the littoral zone, as climate change and its coastal effects, sea level rise and extreme waves.



Ana Lloret, Director for Marine Environment, has been working at CEPYC since 1990. Her personal research interests are the assessment and management of human activities on the marine environment to improve the protection of the sea.

Ramón Gutiérrez-Serret, IAHR Secretary General, is the Director of the Maritime Hydraulics Laboratory. He is currently involved in studies of physical models in the maritime domain and ship maneuvering simulation. Previously, during 20 years, he was committed to hydraulic works.

Conclusion

Either directly, or indirectly, CC now represents a major driver of the activities in Maritime Hydraulics Research Centers, in some cases imposing new functional and structural requirements on traditional issues and in others motivating new areas of work. To preserve coastal and marine environmental quality in view of CC there is a need for a broader application of risk analysis, and greater emphasis on incremental flexible solutions and forecasting of disruptive situations. ■

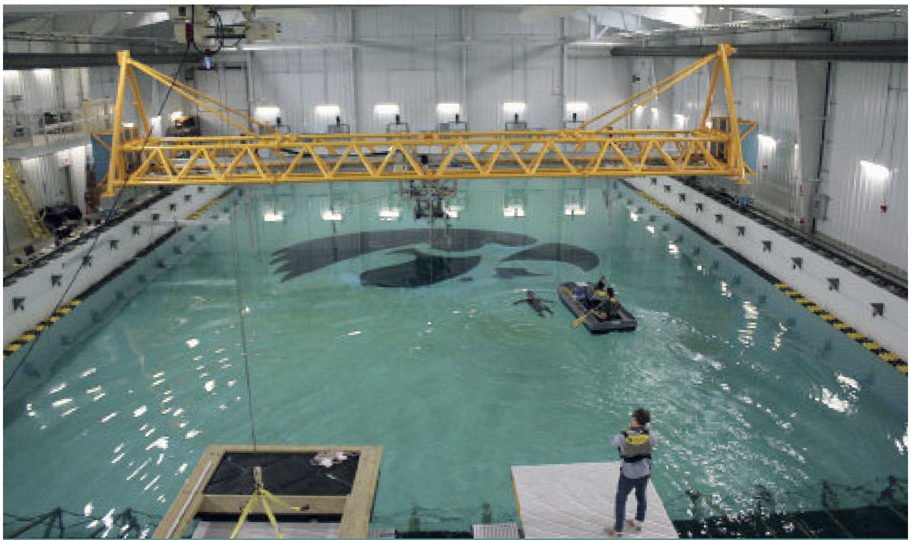
RIDING THE WAVE OF THE FUTURE

IIHR – HYDROSCIENCE & ENGINEERING: A DISTINGUISHED PAST, FOCUSED ON THE FUTURE

BY TROY LYONS & FRED STERN

IIHR – Hydroscience & Engineering (IIHR) has been a world leader in the field of hydraulic engineering and research for nearly a century. This world-renowned institute for hydraulic research and fluid mechanics is constantly evolving and expanding its scope to remain at the forefront of the field.

Based in the C. Maxwell Stanley Hydraulics Laboratory on the Iowa River, IIHR is a unit of the University of Iowa's College of Engineering. With remarkable computational capabilities and extensive experimental facilities—including a state-of-the-art wave basin, a new wind tunnel, and space for large models for fish passage studies and stormwater management structures—IIHR researchers are able to study fundamental processes and their ever-diversifying real-world applications.



CalWave is one of several organizations that tested wave energy technologies at the IIHR Wave Basin. The teams are competing for a U.S. Department of Energy prize; IIHR was selected as one of five sites nationwide to serve as a test facility

DOE Wave Energy Prize Comes to IIHR

In 2015, the U.S. Department of Energy (DOE) selected IIHR as a test facility for its prestigious Wave Energy Prize competition. Wave energy devices convert energy from ocean waves into electricity. The DOE launched the competition to stimulate development of innovative wave energy converter (WEC) technologies. Twenty teams competed for finalist status and the opportunity to test at the Naval Surface Warfare Centre's Maneuvering and Seakeeping basin. First, however, they had to pass proof-of-concept small-scale testing. Several did so at the IIHR Wave Basin.

The IIHR Wave Basin is a state-of-the-art facility completed in 2010 that accommodates free-running model testing in an open body of water. Researchers use the 40x20x3-meter wave basin to test captive or radio-controlled model-scale navy ships under a variety of real-life conditions, created by the basin's six wavemakers. The free-moving models can maneuver straight ahead, zigzag, full circle, and capsize.

A custom eight-ton overhead carriage tracks the radio-controlled ships using indoor global positioning and two-camera vision, shadowing the vessels to within +/-100 mm. A 3D particle image velocimetry system measures fluid velocities around the ships, facilitating the collection of detailed flow data. IIHR's wave basin is the first to include local flow measurement capabilities, critical for continued development of simulation-based design tools. Unlike towing tanks using captive ship models, which typically allow only straight line movement with very limited side to side motion, the wave basin facility with its local flow measurement instruments can test ships under many different real-world conditions, measuring the water flow and wave patterns around the ship, including breaking waves, bubbly ship wake flows, unsteady hull surface pressure, and more.

The wavemaker system consists of six wedge-shaped plungers aligned end-to-end with minimal clearance between ends. Each plunger is 1.2 meters high, 3.3 meters wide, and 0.8 meters thick and submerged 0.7 meters in calm water. The plunger stroke is adjustable up to 250 mm for plunger frequencies less than 0.62 Hz, where the maximum stroke is restricted to 77.5 mm at the maximum plunger frequency of 2.0 Hz. The wavemaker system has two operational modes. The first mode generates regular waves by using pre-set and fixed plunger amplitude and frequency values. The waves are generated in this mode with all six plungers moving simultaneously with the same amplitude and frequency and the same initial phase. The second mode generates irregular waves, where a train of analogue voltage signals of arbitrary wave form are input to each plunger. The wavemakers are calibrated to meet the International Maritime Organization (IMO) requirements (Bottiglieri et al., 2015).

IIHR's wave basin is also well-equipped to test wave energy converters under real-world conditions, making it a valuable tool in assessing whether the WEC devices will be able to prove their worth on the open ocean. Devices were tested under specified regular and irregular wave conditions designed to replicate sea conditions. Measurements included ultrasonic measurements of wave elevations, pressure fluctuations below the WEC, six-degrees-of-freedom (6DOF) motion capture of the WEC body, mooring forces, and resistance and displacement of the power take-off devices used to calculate power. In addition, underwater video was recorded for selected cases. Typical tests included 120 seconds of synchronized data collection. Judges chose the finalists based on their potential to double the energy produced by current WEC technologies. IIHR received funding from global engineering firm Ricardo PLC to conduct the testing of WEC devices entered in the DOE prize competition. The University of Iowa was one of five test sites for the small-scale WEC models. The testing process enabled IIHR to expand the capabilities of its advanced wave basin facility by matching specific wave frequencies and durations while controlling different conditions.

Private industry companies Teams Sea Potential, CalWave, and Waveswing America conducted WEC testing at the IIHR Wave Basin. All three companies are now among the nine finalists and two alternates announced in March 2016. The highest-ranked team after the 1:20 scale testing at the U.S. Naval Surface Warfare Center will receive a total prize purse totaling more than $2 million. The second-place team will receive $500K and third place $250K.

The WEC testing process was a collaborative team effort that included IIHR researchers, shop staff, and others—particularly researchers Yugo Sanada, Hyunse Yoon, Lyons, and Alan McCarville.

IIHR Ship Hydrodynamics

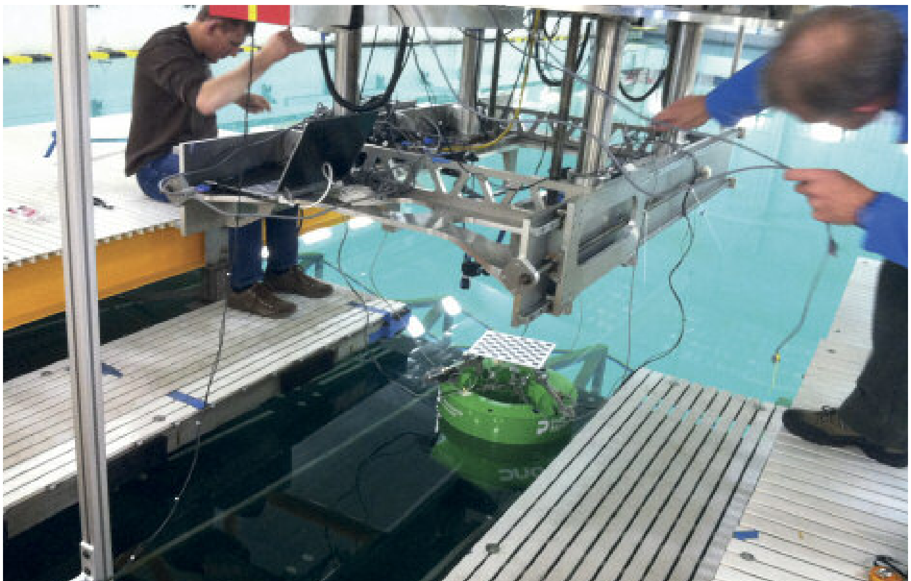
IIHR has played a major role in the evolution of fundamental ship hydrodynamics for decades. Since its founding, the Office of Naval Research (ONR) has continuously funded IIHR's ship hydrodynamics research. IIHR's work with the U.S. Navy began during World War II, focusing on resistance, turbulence, cavitation, and developing hydraulics into a rigorous engineering discipline firmly based on fundamental fluid mechanics.

Today, IIHR's advanced facilities and expertise continue to support the U.S. Navy, while new enhanced capabilities in wave generation promise exciting new avenues of research. The IIHR ship hydrodynamics program is a leader in the area of simulation-based design (SBD), which is revolutionizing naval ship hydrodynamics. Computer simulations guide model-scale physical experiments conducted in the IIHR Wave Basin and towing tank.

The towing tank is 100 m long, 3 m wide, and 3 m deep, equipped with a drive carriage, a planar motion mechanism (PMM), and wall-side wave dampeners and wave-dampening end-beach. The drive carriage is instrumented with data-acquisition computers, a speed circuit, and signal conditions. The drive carriage pulls the PMM carriage, which is used as a contact point of attachment for certain models. The wave-dampeners and the end-beach enable 12-minute intervals between carriage runs. This combination of computer simulations and experiments, in conjunction with sophisticated uncertainty analysis and optimization, puts IIHR at the cutting edge of research in ship hydrodynamics.

Over the next decade, IIHR hopes to build on the strength of the current program, emphasizing international collaborations and focusing on second-generation SBD tools, supported by physical experiments in IIHR's towing tank, flumes, and wave basin.

IIHR's unique combination of resources, facilities, and people promise an ongoing role for IIHR at the front lines of naval ship design. ■



Team Sea Potential prepares its WEC device prior to testing

IIHR Director of Engineering Services *Troy Lyons* joined the institute in 2001 as a staff engineer, after earning BS and MS degrees in civil engineering from the University of Iowa. His role at IIHR has evolved and grown, and today he bridges the gap among clients, engineers, researchers, and shop staff. His research spans several areas, including:

- **Hydraulic structures and dropshafts**
- **Hydropower and fish passage**
- **River hydraulics**
- **Laboratory measurement methodology**
- **Field measurement systems**

Lyons has been instrumental in the success of many multimillion-dollar proposals and collaborative efforts at IIHR.

***Fred Stern* earned BS, MS, and PhD degrees in naval architecture and marine engineering from the University of Michigan. He joined IIHR–Hydroscience & Engineering (IIHR) is 1983, bringing expertise in propellers, free surface effects, and numerical methods for viscous flows. Today, Stern heads up the ship hydrodynamics program at IIHR. Under his leadership, researchers at IIHR have successfully integrated experimental fluid dynamics, computational fluid dynamics, and uncertainty analysis to create simulation-based design. IIHR researchers also developed a groundbreaking computer code, CFDShip-Iowa, the most advanced in the world for ship hydrodynamics. Stern is also the first George Ashton Professor in Hydroscience and Engineering at the University of Iowa.**

STORMWATER MANAGEMENT AND ROAD TUNNEL (SMART)

FLOOD DETECTION SYSTEM, OPERATION AND PERFORMANCE

BY ROSLINA YUSOP, AMIRUDDIN ALALDIN & NOR AZAZI ZAKARIA

Figure 1. Typical Tunnel Cross-Section at Traffic Compartment

SMART is an innovative project of the Government of Malaysia to solve flooding problem in the City Center of Kuala Lumpur. The SMART project has been a great challenge for the local engineers involved in management and construction as it runs below congested roads, near sensitive structures and through varied geological ground conditions. The project also serves to ease the traffic congestion problem between Kuala Lumpur City Center and Southern gateway at Sungai Besi. A unique feature of SMART is the 3 km double-deck motorway in the middle section of the 9.7 km stormwater tunnel which starts near the Kampung Pandan roundabout in the city center, and ends at Kuala Lumpur-Seremban Highway next to the TUDM Airfield at Sungai Besi.

After the major 1971 floods, the Malaysian Government constructed several flood mitigation works such as the Batu Dam, Klang Gates Dam, widened and deepened Klang River and Gombak River including concrete channelization. The projects were completed in early 1990s, but floods continue to occur in the city center of Kuala Lumpur, triggering the government to search for a smarter solution.

As experienced by the residents and businesses on several occasions, Kuala Lumpur get flooded easily even after just a couple of hours of heavy rain. Studies showed that the major flood-prone areas are along the Klang River between the confluence of the Klang River and the Ampang River, and the confluence of the Gombak River and the Klang River. Since it was not possible to widen the flood plain of the river because of developments along the riverbank, the only alternative is to hold and divert the floodwater upstream before it reaches the critical areas. Planning the SMART was against the traditional solutions available at that time after considering factors such as the high land acquisition cost, insufficient space to widen the river channel, and the complex social and environmental issues that would involved in the construction of flood mitigation dam.

The SMART project was implemented jointly by the Department of Irrigation and Drainage Malaysia and Malaysian Highway Authority as the executing government agencies. The construction of the project started 1st January 2003 and was completed on the 30th June 2007 with the total cost of RM 1.933 Billion (0.48 Billion US Dollars).

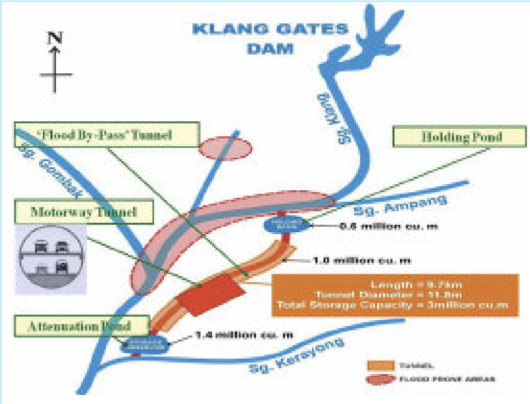


Figure 2. Schematic Layout of SMART

SMART's Flood Detection System

A sophisticated and modern Flood Detection System (FDS) provides real time flood forecasting information. This enables the efficient and safe management at the operation of the tunnel. The SMART FDS Modeling System is comprised of hydrological and hydrodynamic models, a database and scheduler. The hydrologic rainfall-runoff model provides a warning time for tunnel opening using real-time rainfall information from upper catchments and the surrounding areas to predict stream flows. The hydrographs produced from the rainfall-runoff model are automatically input to the hydrodynamic model. The model is fully integrated using scheduler program to extract all relevant data for input to the model and run the model in a seamless fashion. The time series manager database interacts with the SCADA system to achieve all data collected from the monitoring sites (refer to figure 2).

Figure 3. SMART's Catchment Monitoring System

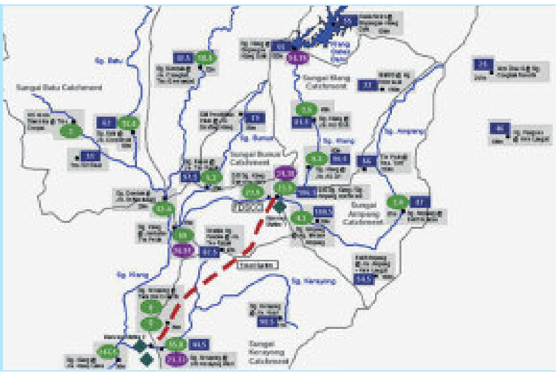
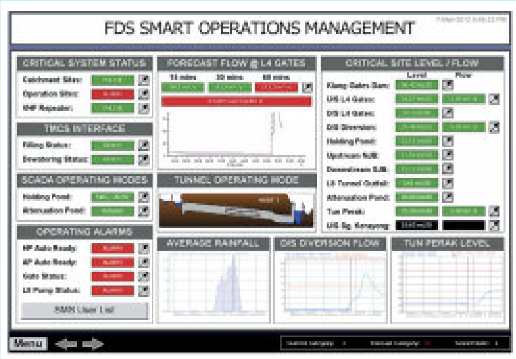


Figure 4. FDS SMART OPERATION MANAGEMENT



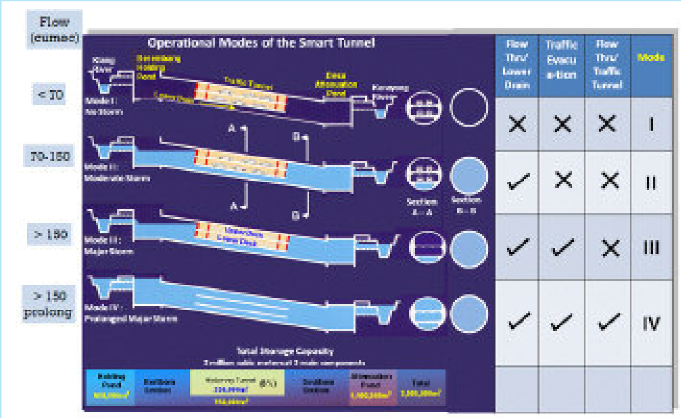


Figure 5. SMART Operation Modes

Based on the predicted hydrograph from the hydrologic model and measured data such as water level, flows and control gate position from the field, the hydraulic modeling components of the FDS predicts flood level and discharges within the SMART system and the surrounding rivers, and also predicts SMART control gate and pump operation. The predictions provide information to aid the tunnel operators in decision making regarding the operation of the SMART system. One hydraulic model is used in the FDS which encompasses both the hindcast and forecast components of the hydraulic model. The model automatically switches from hindcast operation to forecast operation based on a trigger contained in a time-series file which is generated by the FDS. For hindcast operation the model uses measured flows and gate levels up to the "now" time. For forecasting the model uses the forecast flood hydrographs and the SMART gate and pump operation rules.

Event Statistics – 272 Diversion Events until December 2015

Since SMART establishment, there had been 272 heavy rainfall events and flood water diversion operations from July 2007 until December 2015. Five (5) of the events were major flood events and the system was operated under Mode IV of SMART's Standard Operation Procedure.

YEAR	MODE II	MODE III	MODE IV	TOTAL
2007	13	2	0	15
2008	30	21	1	52
2009	20	13	0	34
2010	11	14	0	25
2011	21	19	1	41
2012	25	8	3	36
2013	21	2	0	23
2014	25	2	0	27
2015	19	0	0	19
TOTAL	173	81	5	272

Figure 6. Event Statistics

SMART Has Successfully Performed Under Design Storm Situation

The largest storm event occurred on March 7, 2012. One of the gauging stations in the Ampang River catchment recorded very high rainfall, 227 mm in 4 hours which exceeds the 100 year Average Recurrence Interval (ARI). The Klang River catchment recorded rainfall of 111 mm which is close to the 100 year ARI. The average rainfall for the overall SMART's catchment was 133 mm as shown in Figure 7.

The maximum Flow at the confluence of Klang River and Ampang River reached 475 m^3/s when the flood water was diverted into the holding pond. A total volume of 3.3 Million cubic meters of floodwater was successfully diverted through the SMART structures. Klang River in the city center was overtopped by only 15 cm for 27 minutes during the major storm event.

SMART has successfully performed under design storm conditions and saved millions in flood damage costs, thus serving its purpose in reducing stormwater that flow through the city center. SMART is only one part of the Kuala Lumpur flood mitigation program and operates together with the other programs to reduce flood risk in Kuala Lumpur. ■

References:

JPS SMART, 2008. Functional Specification FDS Modelling System. Hydra Electric Corporation.

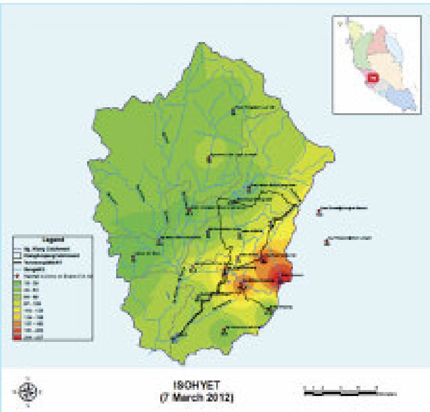
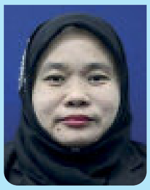


Figure 7. Isohyets Map Showing Rainfall Pattern for Event on 7 March 2012



Ir. ***Roslina Yusop*** **started working at the Department of Irrigation and Drainage Malaysia in 1993. She received Bachelor of Science in Civil Engineering from the University of Hartford, Connecticut U.S.A in 1988, and a Master of Science in Water Engineering from Universiti Putra Malaysia in 2004. She had more than 10 year's experiences in the field of hydrology, hydraulic and flood forecasting. During her career, she was also involved in planning, design and management of drainage and irrigation projects. Her recent position is the Deputy Director for SMART Storm water Control Center.**

Ir. *Amiruddin Alaldin* started working with DID in 1992 and had spent the first 12 years working with flooding issues in the southern and cental regions of Malaysia. He completed his MSc. in Information Management in 2003 thus assuming the head of information Management and Corporate Relations before being promoted as Director of Performance Audit Division where he spent most of the efforts in ensuring the DID core business outputs are delivered with the highest efficiency and effectiveness whilst utilizing economical and optimized resources. His next challenge was to oversee the implementation of 14 mega projects costing RM1.5 billions under the Special Projects Division before being made the Director of Operation for SMART Control Center.

Prof. Dr. *Nor Azazi Zakaria* has served in Universiti Sains Malaysia since 1994. He then established the River Engineering and Urban Drainage Research Centre (REDAC) in 2001 and has since remained as the Director. His main research interests are Sustainable Urban Drainage Systems and River Management. Dr. Nor Azazi is the leading researcher in the innovation of Bio-ecological Drainage System (BIOECODS), and is now an established figure in the field of stormwater management at national and international levels. He sits in the Executive Committee for Malaysian National Committee on Irrigation and Drainage (MANCID) and Malaysia Stormwater Organization (MSO), as well as IAHR APD.

Advance Fluid Measurement

Provide a complete solution to the deep research of fluid mechanics

Large area surface flow field measurement system

- **Sky Canopy**

Super lager scale measurement based on Internet of Things technology

- **Sky Eye**

Rapid surface measurement with unmanned aerial vehicle

@ADV

The first wireless Velocimeter Created in China

Wireless transmission

Sufficient battery life

Multiple point synchronous measurement

☎ +86-10-62988330   www.sinfotek.com   ✉ sinfotek@126.com