HENRY
Hydraulic Engineering Repository

Ein Service der Bundesanstalt für Wasserbau

Article, Published Version

Fonseca, Pedro; Marques, Nelson

CFD in wastewater treatment plants

Hydrolink

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/109322

Vorgeschlagene Zitierweise/Suggested citation:
Fonseca, Pedro; Marques, Nelson (2016): CFD in wastewater treatment plants. In: Hydrolink 2016/2. Madrid: International Association for Hydro-Environment Engineering and Research (IAHR). S. 36-38. https://iahr.oss-accelerate.aliyuncs.com/library/HydroLink/HydroLink2016_02_Hydraulics_Wastewater_Treatment.pdf.

Standardnutzungsbedingungen/Terms of Use:

Die Dokumente in HENRY stehen unter der Creative Commons Lizenz CC BY 4.0, sofern keine abweichenden Nutzungsbedingungen getroffen wurden. Damit ist sowohl die kommerzielle Nutzung als auch das Teilen, die Weiterbearbeitung und Speicherung erlaubt. Das Verwenden und das Bearbeiten stehen unter der Bedingung der Namensnennung. Im Einzelfall kann eine restriktivere Lizenz gelten; dann gelten abweichend von den obigen Nutzungsbedingungen die in der dort genannten Lizenz gewährten Nutzungsrechte.

Documents in HENRY are made available under the Creative Commons License CC BY 4.0, if no other license is applicable. Under CC BY 4.0 commercial use and sharing, remixing, transforming, and building upon the material of the work is permitted. In some cases a different, more restrictive license may apply; if applicable the terms of the restrictive license will be binding.

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

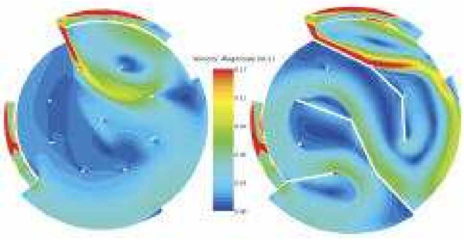

Figure 1. Contact tank with two distinct baffles arrangements (the arrangement on the left has baffles only at the entrance, but not in the interior of the tank). Velocity distribution on a horizontal section plane at half height. (Volume: 100 000 m^3; flow rate: 2 m^3/s)

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 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

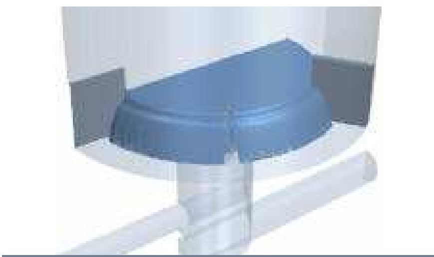

Figure 2. Discharge weir with water injection at the bottom (Diameter: 8 m; flow rate: 4 m^3/s). This injection pattern gives rise to non-uniform flow distribution across the weirs since the flow is not guaranteed to be symmetrical

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.).

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

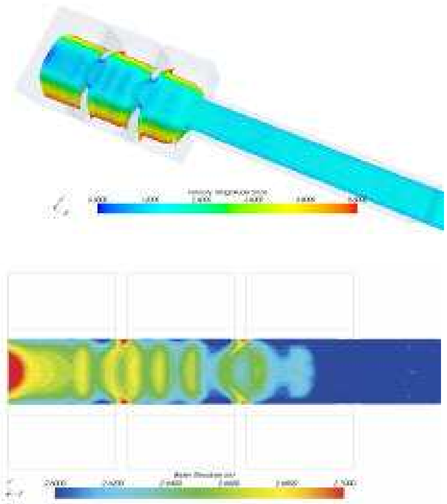

Figure 3. Water approaching through open channel (channel width: 2.5 m; flow rate: 6.5 m^3/s). Velocity distribution in impinging flow at closed extremity composes non-uniformity in the flow caused by lateral discharge weirs. Lower image displays free-surface height relative to datum

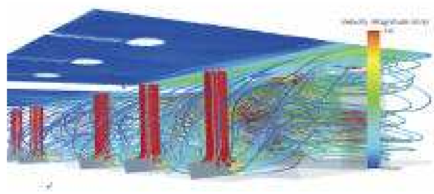

Figure 4. Pumps inlet at pump station. Flow rate: 5 m^3/s, distributed through 5 submersible pumps. Flow pattern shown through streamlines to assess admission requirements for safe pump operation

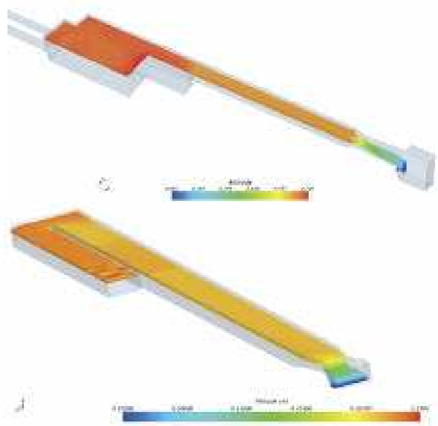

Figure 5. Flumes operating under the influence of discharge chamber and channel bend. Flow rates approximately 1 m^3/s

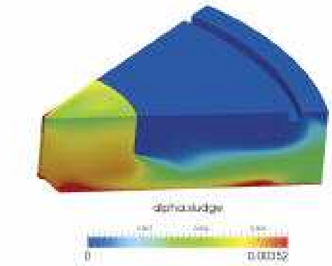

Figure 6. Secondary clarifier (18m radius) sludge blanket modelling. Flow rate ~1.2 m^3/s. Colours represent sludge concentration

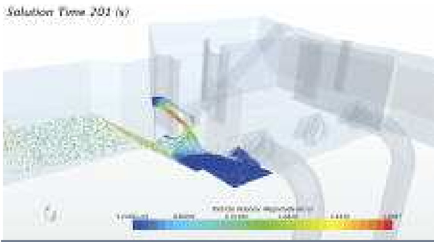

Figure 7. Grit-trap optimization. Flow rate: ~2 m^3/s. Each particle represents a grit element

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

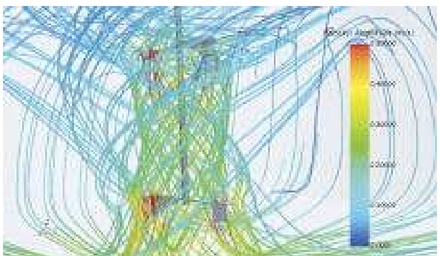

Figure 8. Streamlines across mixers simulated with rotating reference frames

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

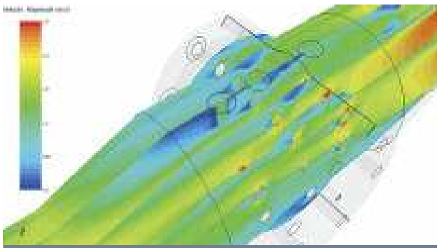

Figure 10. In-line UV reactor with lamps placed perpendicularly to the fluid flow. The intensity of the germicidal irradiation is computed throughout the domain and its eventual absorption monitored through particles released in the flow

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.