Optimization of water treatment plant flow distribution with CFD modeling of an influent channel
Carrie Knatz, Stephen Rafferty and Anthony Delescinskis

ABSTRACT
In the design of water and wastewater treatment plants, proper flow and solids distribution can be as critical as process design considerations. Insufficient treatment and even plant failures can result from unequal and unmanageable flow and solids distribution. Computational fluid dynamics (CFD) modeling is a valuable tool in the evaluation of flow distribution to multiple units within a treatment process. This article reviews the benefits achieved by performing a CFD analysis of an Infilco high-rate dissolved air flotation (DAF) influent channel prior to finalizing the design of the plant. The CFD model was used to optimize the DAF influent channel configuration with respect to flow distribution to 10 identical process units that were inserted into an existing facility footprint. For the initial configurations modeled, the largest deviation of flow rate to an individual DAF unit was over 60%. Using CFD, design engineers developed a DAF influent channel configuration predicted to achieve less than 10% deviation. The upgraded facility is constructed and in service and the results of the CFD model were confirmed using actual turbidity data, which indicate that the solids are evenly distributed to the DAF process trains.

Key words | case study, CFD, flow distribution, influent channel

ABBREVIATIONS
CFD computational fluid dynamics
cm centimeters
DAF dissolved air flotation
m/s meters per second
m³/d cubic meters per day
NTU nephelometric turbidity units
WTP water treatment plant
VOF volume of fluid

INTRODUCTION
A critical component for successful treatment plant process performance is even solids and flow distribution to the multiple treatment process trains. Process performance is often dependent on specific retention times and when there is a significant deviation in flow rates between treatment process trains agencies find it challenging to meet regulations. Influent channels are used to distribute the flow from one process to the next. Engineers have many tools to use when designing influent channels. There are energy-intensive assemblies such as flow meters and flow-regulating valves that are used to achieve equal flow distribution, and there are less energy-intensive options that utilize structural components such as rectangular weirs, V-notch weirs and submerged orifices (Benefield et al. 1984). Energy is often not readily available when upgrading treatment plants and engineers are challenged with finding a design that balances energy, cost and performance.

Distribution channels, or dispersion conduits, were studied by Camp & Graber (1968) where general equations were derived, and broad functional hydraulic interpretations were given for a number of flow phenomena. Following this work by Camp and Graber, Yao (1972) investigated the control characteristics of orifices and weirs and developed general guidelines to select control devices for flow...
distribution applications. Yao emphasized the sensitivities and efficiencies of various control devices. Limited information was available to engineers for the analysis of open channel distribution channels until the ‘step method’ was developed. The ‘step method’ is now a well-known approach for calculating flow distribution in open channels.

The ‘(correction) step method’ proposed by Chao & Trussel (1980) has been widely used as a design method of distribution channels. In this method, flow distribution to each basin is determined by proceeding step-by-step from the downstream end of the channel to the upstream end where the flow enters. Even with this developed method, a review of various plants with multiple process units has shown that with traditional flow splitting, solids in suspension in the flow stream typically migrate to the downstream-most basin in an assembly of multiple basins. Particles in suspension are carried by momentum and pass downstream-most basin in an assembly of multiple basins. It has been proven that an equitable flow distribution could not be achieved in most channels that were designed by the ‘step method’. The maldistribution of flow occurs mainly from the abrupt turn of flow direction and inadequate representation of the inlet geometry. Ultimately this phenomenon impaired the treatment efficiencies seriously (Hudson 1981; Baek & Kim 2000). In recognition of the importance of equitable flow distribution in channels, computational fluid dynamics (CFD) modeling has been identified as a reliable tool to investigate the hydrodynamic behavior in a full-scale channel (Ta 1999; Baek et al. 2005; Dutta et al. 2010; Lin 2012; Nopens et al. 2012). The objective of this study was to optimize the design of the Infilco high-rate dissolved air flotation (DAF) influent channel with respect to the flow distribution to 10 identical process units that were inserted into an existing facility footprint.

**MATERIALS AND METHODS**

**Case study background**

The Haworth water treatment plant (WTP) in Oradell, New Jersey, provides up to 710 m$^3$/d of water to 800,000 users in northern New Jersey. The raw water supply to the plant is from a series of impoundments in the upper reaches of the Hackensack River. The river's watershed is highly developed, and due to the small volume of impounded water, the water quality is variable with a wide range of turbidity from day to day. The existing WTP utilized a treatment scheme of oxidation, sedimentation and filtration followed by disinfection. As regulatory requirements were revised over the last decade the plant experienced shorter filter run times during peak demand periods. Accordingly, treatment plant upgrades were identified in response to the new, stringent drinking water regulations and growing customer demands. To expedite completion of the plant upgrades, the design-build delivery approach was selected by United Water New Jersey. Design-build delivery and successful collaboration allowed major process treatment units to be designed, permitted, constructed and placed in service in only 21 months, significantly faster than using a typical design-bid-build approach.

The upgraded plant treats water from the Oradell Reservoir, the major impoundment on the upper Hackensack River, using ozone for pre-oxidation, polyaluminate coagulant, high-rate DAF, intermediate disinfection, filtration and final disinfection. The plant upgrade was constrained by a requirement to maintain existing operations and the need to insert new processes within the existing plant’s hydraulic profile. Early engineering identified that the new processes would have to function with as little as 0.46 m of available energy at peak flow. Inserting a high-rate DAF process to replace the existing sedimentation basins was the optimal treatment system identified for the raw water characteristics and variability. Haworth’s high-rate DAF is the largest system of its kind in the United States. It removes 90% of particles and algae from source water, saving energy and improving water quality. The efficiency of the DAF system has substantially increased the existing dual media filter’s run time and has reduced backwash water volume by about 60% with a resultant decrease in energy and treatment of backwash costs. High-rate DAF requires one-eighth of the process tank volume needed for traditional sedimentation, which conserved 48 m$^2$ of woodland and saved millions on costly infrastructure.

The challenge to the design team at Haworth was to find a solution that used minimal energy and achieved optimal distribution of solids and flow to the multiple DAF process trains. Traditional methods for flow distribution such as
distribution boxes, flow meters and rate-controlling valves required more hydraulic head (energy) than was available. CFD was identified as a valuable design tool to evaluate low energy flow distribution configurations where the DAF influent channel design was extensively modeled using CFD analysis to identify the optimal flow distribution design. Peak flow was evaluated since the main concern with maldistribution of flow was the degradation of DAF performance where any one DAF unit is overloaded with flow greater than 10% over the design flow rate.

**CFD model methodology**

CFD is the science of predicting fluid flow by solving the mathematical equations that represent the three fundamental principles of fluid flow (conservation of mass, momentum and energy). A finite volume formulation is used where the computational domain is generated representing the geometry to be simulated, and the domain is divided into discrete cells (the mesh) where the fundamental fluid flow equations are solved on each of these discrete cells (Sicilian et al. 1987; Park et al. 2003).

For simulating the hydrodynamic behavior in the Haworth WTP DAF influent channel, ANSYS Fluent (Fluent) commercial CFD code was used. Fluent used in this simulation contains the continuity equation, momentum equation (Navier-Stokes) and volume of fluid (VOF) method suggested by Hirt & Nichols (1981) for free surface dynamics. The time-averaged Navier-Stokes equations for momentum and continuity were solved in this study for steady-state, incompressible, turbulent and isothermal flow. The continuity and momentum equations are as follows, respectively:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}
\]

\[
\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot \mathbf{T} + \mathbf{f} \tag{2}
\]

where \(\rho\) is the fluid density, \(p\) is the pressure, \(T\) is the stress tensor, \(f\) represents body forces and \(u\) is the flow velocity.

Reynolds averaging was used where the solution variables in the instantaneous Navier-Stokes equations are decomposed into the time-averaged and fluctuating components. Mean flow and turbulent characteristics of open channel flows have been studied by experimental measurements and numerical computations. In many engineering problems in fluid mechanics and hydraulics the k-epsilon model has been most widely employed due to well-established empirical coefficients of the model (Lee et al. 2012). Since the Reynolds number in the influent channel is high enough to guarantee turbulent fluid conditions, all simulations run for this case study were simulated by the realizable k-epsilon (RKE) turbulent model. This model is based on the turbulent equations for turbulence kinetic energy (\(k\)) and its dissipation rate (\(\epsilon\)). The term ‘realizable’ means that the model satisfies certain mathematical constraints on the Reynolds stresses and the Schwarz inequality for turbulent shear stresses. This turbulent model has been proved to perform well for a variety of flows (Shih et al. 1994). The RKE model combines the Boussinesq relationship and the eddy viscosity definition to obtain the following expression for the normal Reynolds stress in an incompressible strained mean flow:

\[
\overline{u'^2} = \frac{2}{3} k - 2\nu_t \frac{\partial \overline{u}}{\partial x} \tag{3}
\]

The VOF model is used to simulate two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each fluid throughout the model domain. For this study, water and air are the phases tracked and the water surface is developed as an interface between the phases. For the \(q\)th phase, the volume fraction equation has the following form:

\[
\frac{1}{\rho_q} \left( \frac{\partial}{\partial x} \left( \alpha_q \rho_q \mathbf{u} \right) + \nabla \cdot (\alpha_q \rho_q \mathbf{u} \mathbf{u}) \right) = S_{\alpha_q} + \sum_{p=1}^{n} (m_{pq} - \dot{m}_{pq}) \tag{4}
\]

where \(\dot{m}_{pq}\) is the mass transfer from phase \(p\) to phase \(q\) and \(m_{pq}\) is from phase \(q\) to phase \(p\).

Different solution methods and discretization schemes were evaluated for the first couple configurations and those used developed a stable solution where all residuals were below the convergence criterion of \(1 \times 10^{-5}\).
Geometry of the studied work

Figure 1 illustrates the initial influent channel configuration (Configuration 1) modeled. Note that for clarity in illustration, the model domain was scaled up by a factor of 5 in the y direction.

The design of the DAF influent channel was restricted by site constraints and the geometry of the existing influent channel, making a difficult design condition even more challenging. The initial design of the DAF influent channel consisted of a primary channel that feeds two secondary channels, each delivering flow to five DAF units. From the secondary influent channels, the flow enters each DAF unit through an influent gate opening and then over an influent weir.

The width of the primary channel was designed to be 1.5 m. Similarly, the upstream end of the secondary channels was designed to be 1.5 m, but tapering to 0.9 m at the downstream end. Each secondary channel was 75 m and each DAF unit had a width of 13.4 m corresponding to the length of the weir opening. The influent gate openings from the secondary channels to each DAF unit were 1.2 m by 1.2 m. The mesh was generated with only hexagonal-shaped cells. The mesh cell size was specified as 0.15 m and resulted in a total cell count of 600,000. The mesh quality had a maximum skewness of 0.36 and minimum orthogonal quality of 0.87. Note the ‘rule of thumb’ maximum skewness is 0.8 and minimum orthogonal quality of 0.2; therefore, the mesh was considered to be of high quality.

Boundary conditions

Since the hydraulic control point for the influent channel system was the influent weir for each DAF unit, it was important to allow the model to predict the water surface. Therefore, the VOF multi-phase model was applied to include the air and water volumes. For this multi-phase model, the downstream boundary was defined at the weir openings and the water surface (air/water interface) was predicted by the model. The only specified water surface was at the inlet boundary, the upstream end of the primary influent channel. The upstream boundary condition was located at a point in the primary influent channel where a uniform velocity profile was expected based on the straight length of the channel. The flow develops immediately downstream of the inlet boundary and allows the model to determine the water surface throughout the model domain with the critical prediction at the downstream boundary where flow goes over the influent weirs to the individual DAF units. For this downstream
boundary condition, a two-dimensional plane above each DAF unit weir opening was specified as a zero pressure outlet. An air inlet was specified to allow air to enter or exit the model domain as the water surface (air–water interface) moves and the model solution is developed.

The model simulated the plant design capacity of 710 m$^3$/d. The plant design capacity was selected for the evaluation as it is considered to be the worst-case scenario for solids loading.

Table 1 presents a summary of the boundary conditions specified and input parameters. The hydraulic diameter is calculated using the following formula:

$$\text{Hydraulic Diameter} = \frac{\text{Wetted Area}}{\text{Wetted Perimeter}} + 4$$ (5)

And, turbulence intensity is defined as the ratio of the root-mean-square of the velocity fluctuations and is also related to the Reynolds number, Re, by the following formula:

$$\text{Turbulence intensity} = 0.16 + (Re)^{-1/8}.$$ (6)

RESULTS AND DISCUSSION

Table 2 presents a summary of the different influent channel configurations modeled. The evolution of the design and reasoning for the differences between the configurations is discussed below.

Figure 2 illustrates the changes in each configuration as summarized in Table 2. Similar to Figure 1, for clarity in illustration, the model domain was scaled up by a factor of 5 in the y direction.

Figure 3 presents the model results for the flow distribution from the primary channel to the secondary channels (primary flow split) where DAF Units 1 through 5 are fed by one secondary channel and DAF Units 6 through 10 are fed by the other secondary channel. For the initial configuration (Configuration 1), the model predicted that 59% of the flow would be conveyed to the secondary channel for DAF Units 6 through 10 and only 41% of the flow to the secondary channel for DAF Units 1 through 5. It was apparent that the direction of flow in the primary channel caused the momentum to convey more flow to the secondary channel for DAF Units 6 through 10.

Design change to improve primary flow split

To address the issue of the primary flow split maldistribution, two mixing basins were introduced between the primary and secondary channels (Figure 2(a)). The primary channel flow enters the first of two mixing chambers, goes over a baffle wall to the second mixing chamber, then
Figure 2 | Illustration of different features for each configuration: (a) Configuration 2; (b) Configuration 3; (c) Configuration 4; (d) Configuration 5.

Figure 3 | Primary flow split for various configurations.
under a baffle wall and up where it splits to the two secondary channels. The dimensions of these mixing chambers were determined to be square to achieve complete mixing with the mechanical mixers. As presented in Figure 3, this design change improved the flow distribution from the primary channel to the secondary channels, where the flow split was reduced from ±9% to ±1% at best for Configuration 2 and ultimately ±2% for Configuration 5.

**Design changes to improve the secondary flow split**

To evaluate the flow distribution from the secondary channels to the individual DAF units (secondary flow split), the percentage deviation from the ideal flow was calculated where a maximum of 10% deviation was considered acceptable. Figure 4 presents the percentage deviation in the flow distribution for Configurations 1, 2 and 3; a negative value indicates the flow to that DAF unit was lower than the ideal flow and a positive value indicates a higher flow than ideal. It is apparent from Figure 4 that a significant maldistribution of flow was predicted for Configuration 1 where the majority of the flow conveyed to the downstream-most DAF unit for each secondary channel (i.e. DAF Unit 1 and DAF Unit 10). The downstream-most DAF units received 50–60% more than the upstream-most DAF unit (i.e. DAF Unit 5 and DAF Unit 6). Although the secondary channels were tapered, this configuration does not allow for the even distribution of flow between the five DAF units from each secondary channel. The largest deviation of flow rate was from the primary channel to the secondary channels where DAF Units 8, 9 and 10 saw the majority of the flow (Figure 4).

The maldistribution of flow from the secondary channels to the individual DAF units was a more challenging issue to address than the maldistribution of flow from the primary channel to the secondary channels. As indicated earlier, the design was limited to only 0.46 m of available hydraulic head in the existing plant’s process at peak flow conditions, which eliminated the option of using distribution boxes, orifice baffles, flow meters or rate-controlling valves. The only option was to change structural components such as the channel floor, gate openings and walls.

As discussed above, the main design change after Configuration 1 was the addition of the two mixing basins between the primary channel and the secondary channels (Figure 2(a)). The improvement in the flow split from the primary channel to the secondary channels also improved the flow distribution to the individual DAF units as seen from the significant reduction in flow to DAF Units 8, 9 and 10 for Configurations 2 and 3; the percentage deviation of the secondary flow split reduced by at least 15%.

Although the flow distribution was improved from Configuration 1 to Configuration 2, it was still significantly higher than the ±10% criterion. The low flow to DAF Unit 5...
for Configuration 2 was identified as a significant concern. Figure 5 illustrates the velocity vectors at a longitudinal section through the middle of the secondary channel in the vicinity of the DAF Unit 5 influent gate for Configuration 2. It is apparent that the flow exiting the last mixing basin projects a jet up and over the DAF Unit 5 influent gate, resulting in a non-uniform velocity profile and starving the flow entering the gate opening.

Based on these results, the design was modified for Configuration 3 so that the step up from the mixing basin to the secondary channel was revised from the vertical wall to a sloped floor (Figure 2(b)). In addition, the influent gate opening for DAF Unit 5 was moved off-center, further downstream to avoid the majority of the turbulence resulting from the transition from the mixing basins to the secondary channel. The weir length for each DAF unit was also increased with the expectation that this affordable increase in hydraulic head would improve flow distribution. Figure 6 illustrates the velocity vectors and contours for this revised configuration (Configuration 3). The revised configuration produced a more uniform velocity profile at the DAF Unit 5 influent gate opening and the flow distribution results also reflect this improvement where the percentage deviation of the secondary flow split was reduced by almost 30% from Configuration 1 to Configuration 3 (Figure 4).
Although the flow distribution was improved from Configuration 2 to Configuration 3, it was still higher than the ±10% criterion. The next approach to improving the flow distribution for Configuration 4 was to vary the height of the influent gate openings from largest at the upstream DAF unit and smallest at the downstream DAF unit (Figure 2(c)). Momentum carries the bulk flow to the downstream end of the channel, but with small openings downstream and larger openings upstream more flow will be forced upstream. Varying the height of the influent gate openings was identified to be easily accommodated in the existing design and the simplest solution from a constructability and cost impact perspective. There was also some question as to whether the improvement in flow to DAF Unit 5 for Configurations 2 and 3 was mainly due to the change in the DAF Unit 5 influent gate opening location from centered to off-center rather than the revised vertical wall to a sloped wall. Therefore, Configuration 4 reverted back to vertical wall at the transition from the mixing basins to the secondary channel, but maintained the location of the DAF Unit 5 influent gate opening to be off-center (Figure 2(c)).

Figure 7 presents the percentage deviation in the secondary flow split for Configurations 4 and 5. Configuration 4 included uniform widths for all of the gate openings at 107 cm and varying heights for each gate opening from 60 to 107 cm. As presented in Figure 7, the flow distribution results for Configuration 4 still slightly exceeded the ±10% criterion.

**Optimized design to meet the performance criteria**

A better flow distribution required even more variation in the size of the influent gate openings. Therefore, Configuration 5 was modified to have varying heights and widths for each DAF influent gate opening (Figure 2(d)). DAF Units 1, 2, 3, 9 and 10 included 107-cm wide gates and DAF Units 4, 5, 6, 7 and 8 included 122-cm wide gates. The influent gate opening heights varied from 73 to 122 cm.

Figure 8 illustrates the velocity vectors at the longitudinal section through the middle of the secondary channel at the DAF Unit 5 influent gate opening. It is apparent that the velocity profile in the secondary channel at the location of the DAF Unit 5 influent gate opening was almost uniform and very similar to that for Configuration 3 proving that relocating the gate opening further downstream was the main reason for the improved flow into DAF Unit 5. As presented in Figure 7, the flow distribution was significantly improved overall for Configuration 5 and met the ±10% criterion. Therefore, Configuration 5 was the recommended final design configuration and is what was constructed.

**Turbidity measurements to assess the operation of the designed system**

Upon completion of the treatment plant upgrades, the flow distribution predicted by the CFD analysis was confirmed...
by data obtained by the turbidity analyzer that had been installed at each of the 10 DAF units. The data obtained for turbidity levels in each DAF unit from the first day of startup and ongoing provide the plant’s operators with a clear indication of each DAF unit’s performance and ensures optimal performance. The basis of design for the high-rate DAF was turbidity at the effluent weir of 1 nephelometric turbidity unit (NTU) or less. Typical turbidities are 1 NTU or less, particularly at low plant flows. The real measure of performance has been at higher flows with turbidities consistently 1.5 NTU or less. The true test of the treatment system’s effectiveness occurred during Hurricane Isaac in 2011 and Hurricane Sandy in 2012 when the flow splitting and DAF performance saw extreme solids loading. Unlike many other public utilities in New York and New Jersey, no warnings to the Haworth customers were necessary.

Figure 9 presents the average turbidity levels for 5 months of data collection. The average turbidity over the 5-month period of time was relatively close for all units indicating that solids were evenly distributed and removed. Note that it cannot be expected that these data would match the CFD results perfectly since the plant flow varies on a daily

Figure 9 | Actual average turbidity levels for the various DAF units.
basis and the CFD analysis was only performed for peak flow conditions.

CONCLUSION

Even solids and flow distribution to 10 individual DAF units was identified to be critical for the successful performance of the DAF treatment process at Haworth WTP. Using CFD modeling in an iterative design process, the team was equipped with a tool to develop the design to evenly split the solids and flow with minimal loss of energy. The CFD analysis was critical to identifying the optimal, least cost, most constructible answer to the design challenge. For the initial configurations modeled, the largest deviation of flow rate to the DAF units was over 60%. Using CFD, design engineers developed a DAF influent channel configuration predicted to achieve less than 10% deviation. Operations data have shown that this configuration achieved balanced momentum into each of the openings, realizing even solids distribution.

CFD is a valuable tool in predicting the hydrodynamic behavior in distribution channels. There have been numerous problems in water and wastewater treatment plants due to a poorly designed influent channel that resulted in a significant maldistribution of flow to multiple treatment process trains. Visual inspection, float testing, or tracer testing methods often reveal large inequalities in distribution. Frequently the treatment performance of the process units is seriously impaired by maldistribution of flow. This case study suggests that CFD modeling is a reliable tool that can be used during design development to predict flow distribution from an influent channel to multiple treatment process units. The effort required for a CFD analysis is minimal in comparison to potential implications resulting from a poor design.

REFERENCES


First received 3 October 2014; accepted in revised form 19 October 2014. Available online 8 November 2014