Sustaining Hydropower Energy For Economic Growth And Development In West Africa. Applications of CFD in Integrated Urban Water Management (IUWM)

R.Y.G. Andoh[1]*

Abstract
Computational Fluid Dynamics (CFD) has become an essential tool for the design, development, and evaluation of systems and processes including innovative devices utilized for the control and treatment of Urban Water. CFD is now increasingly being used as a cost-effective rapid prototyping tool for the development, design and optimization of urban water management system components without recourse to several iterations of physical models and prototypes. The ability to accurately predict fluid flow fields in 3-dimensions (including pressure, streamlines and residence time distributions) coupled with the tracking and simulation of particulate and other pollutant behaviour and fate is resulting in insights that are facilitating improved understanding of device and system characteristics; leading to improvements and rapid innovations.

Keywords
Computational Fluid Dynamics–Hydropower–simulation–prototypes

* Prof. Robert Andoh AWD Consult. Inc., 32 Vista Drive, South Portland, ME, 04106, USA Tel: +1 207 450 3670 Email: bandoh@awdconsult.com
[1] **Corresponding author** R.Y.G. Andoh Email: bandoh@awdconsult.com

Contents

1. Introduction and Background

Computational Fluid Dynamics (CFD) is the systematic application of computing systems and computational solution techniques to mathematical models formulated to describe and simulate fluid dynamic phenomena. CFD, being a branch of continuum mechanics, deals with numerical simulation of fluid flow and heat transfer problems.

CFD is increasingly proving to be an effective rapid prototyping tool in the development and assessment of Environmental Technologies and Systems (Faram and Andoh, 2000; Egarr et al., 2005). The near real-life simulation capabilities of this computational tool is leading to a significant reduction in the number of physical prototypes that have to be built for testing and evaluation in the traditional product development and assessment cycle(s); resulting in major savings in time and costs. In the last decade or two, the usage of CFD has dramatically increased in the design process and will continue to grow due to its flexibility and cost-effectiveness.

The paper describes the use of CFD for the development of a low energy non-powered Vortex based Mixer utilized to blend sludge streams and fermented liquors as part of a biological nutrient removal wastewater treatment plant upgrade. CFD predictions are corroborated with post commissioning performance monitoring testing.

On the basis of experiences gained and the practical usefulness of CFD, it is recommended that an Applied Computational Modeling Research Center of Excellence is established at the University of Energy and Natural Resources (UENR) in Sunyani, Ghana. This will involve the setting up, installation and adaptation of high performance 'cluster' of desktop PCs using open source software for the simulation, design and evaluation of environmental systems. It is intended that the research center will ultimately become the leading applied computational modeling center in the sub-region and will collaborate with other internationally renowned centers of excellence. The center will focus on Applied Science benefitting the Environment and Society. It is envisaged that the resulting expertise developed could form the basis of an incubator style consultancy geared towards the commercialization of the resulting knowledge and expertise and ultimately assist in CFD becoming a more widely deployed tool.

2. Development of a Hydrodynamic Vortex Mixer Using CFD

2.1 Design Challenges with Physical Prototypes

There are several problems with the traditional product development and design approach involving the building of several versions of physical prototype. First, the geometry of hydrodynamic separators is typically quite complex, so the cost of building and testing a prototype can exceed $30,000 per unit and take more than eight weeks to fabricate. Second, though verification testing of physical prototypes can easily determine device performance, it provides little insight into why the device performs as it does. This is because it is difficult to visualize fluid dynamics, even when the prototype is made of a clear or transparent material. The only way to obtain even limited visualization is through high-speed video imaging, which adds substantially to the cost of testing. The result is that engineers are frequently left guessing as to why a prototype failed to perform as hoped; instead, they must rely on intuition or guesswork to design the next version.

2.2 Benefits of CFD

CFD is increasingly proving to be an effective rapid prototyping tool in the development and assessment of Environmental Technologies and Systems. The near real-life simulation capabilities of this computational tool is leading to a significant reduction in the number of physical prototypes that have to be built for testing and evaluation in the traditional product development and assessment cycle(s); resulting in major savings in time and costs. In the last decade or two, the usage of CFD has dramatically increased in the design process and will continue to grow due to its flexibility and cost-effectiveness. One major advantage of CFD is that a model can be created and evaluated within a week and at less than 20% of the cost of physical prototyping (Andoh, 2006). Another advantage is CFD provides far more information about reasons behind the performance of a design concept (Andoh et al., 2008). For example, it can provide flow velocity and direction throughout the analytical domain and can also trace the path of particles of different sizes as they flow through the system (Edgarr, 2005). Critical in any simulation, however, is how closely it is able to duplicate reality.

2.3 The Middle Basin Case Example

Johnson County Wastewater's (JCW) design-build project to develop, construct and test a new sludge mixer is a good example of the application of CFD in the design and product development processes. The Middle Basin wastewater treatment plant was among the first in the Kansas City metropolitan area to incorporate Biological Nutrient Removal (BNR) technology to reduce the discharge of phosphorus and nitrogen; thereby ultimately helping to reduce the load of these pollutants into the Gulf of Mexico. Faced with the challenge of retrofitting a new sludge mixing and blending system into their wastewater treatment plant, the decision made was to adapt a non-powered vortex mixing technology typically used for urban water treatment. The objective was to develop a new product that utilized the fluid motion through the device to do all the work so it could operate without any moving parts or external power requirements. The system had to be robust and effective with no filtration systems or small apertures or appurtenances that could potentially clog and require maintenance. Research and development engineers recognized at the outset that this would be a technically challenging project given the constraints of utilizing hydrodynamics only with relatively short timescales for project implementation. There was therefore the need to reduce the traditional product developmental cycle times and costs associated with environmental technology development; which typically involves an iterative process of building successive physical prototypes, testing these to find out how well they work, making assumptions on how their performance could be improved, and then building another prototype to start the process anew.

A simulation of the system modelled with CFD was recognized as a critical stage in this mixer's development. Although empirical and theoretical sizing criteria hinted at the suitability of the solution, there was no actual precedent of the use of the advanced vortex technology in the specific application in relation to mixing or blending of sludge streams. It was therefore decided that the addition of a CFD simulation stage would provide a great deal more insight. From a CFD simulation, it would be possible to identify any low-velocity "dead zones" which might encourage build-up, and also help recognize local areas of re-circulation. More importantly, CFD simulations had the potential to provide direct information about sludge concentrations leaving the mixer for a range of concentrations and flow conditions – something that the theoretical formula and semi-empirical approaches could not predict. Though carbon augmentation in the BNR process can be achieved efficiently and cost-effectively through fermentation, fermenter liquor solids concentrations must be carefully maintained. The process adopted at the Middle Basin WWTP would require a mixing device to thoroughly mix 4% fermented thickened sludge with 0.5% primary sludge before passing a blended sludge stream not exceeding 2% sludge back to the gravity thickener. Because the project was a retrofit and plant operation continued throughout construction, the mixer needed to be unobtrusive and have a small footprint. There was also the need to operate with minimum energy requirements or head loss to enable retrofitting into the existing plant hydraulic grade line.

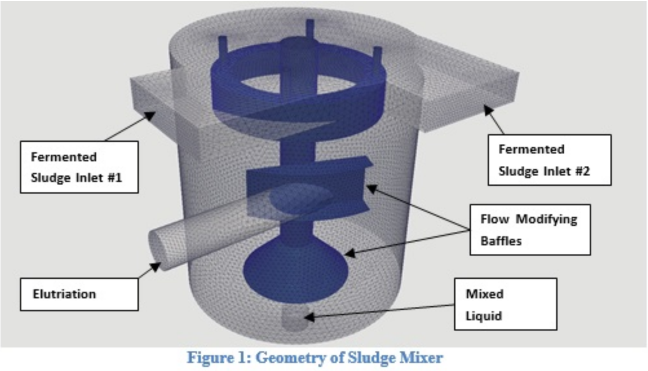

Figure 1. Geometry of Sludge Mixer

2.4 Simulating reality?

The installation at Middle Basin would be the first time that an advanced hydrodynamic vortex vessel was used specifically as a sludge mixer. Although previous experiences and empirical models informed the choice of the appropriate vessel size and configuration, The research engineers determined that CFD analysis would be the best way to verify the design before it was built and installed. The geometry for the mixing vessel was modeled in 3D CAD and meshed with an unstructured mesh comprised of 500,000 tetrahedral cells as shown in Figure 1. Overall geometry and fluid constants are as outlined in Table 1. The inlet conditions are shown in Table 2.

2.5 Simulating reality?

Table 1. Geometry Dimensions and Material Properties

Inlet Channel	Width: 28 in	Height: 14 in	
Elutriation Pipe	Diameter: 8 in		
Outlet Pipe	Diameter:		
Sludge Properties	Conc. 5% dry solids	Density: 1018.164 kg/m^3	Viscosity: 0.0012 kg/ms

Table 2. Inlet Flow Rates in US gals per minute (gpm)

Original Estimate	4% First Sludge	4% Second Sludge	0.5 Elutriation	Total Flow
Minimum	60	60	160	280
Average	60	60	200	320
Maximum	100	100	300	500

Revised Estimate	4% First Sludge	4% Second Sludge	0.5 Elutriation	Total Flow
Minimum	30	30	160	220
Average	30	30	200	260
Maximum	50	50	300	400

2.6 CFD Simulation Results and Discussion

Simulation results included an assessment of path-lines, velocity profiles and predictions of sludge concentrations within the Vortex Mixer Vessel and at the mixed sludge outlet. An example of a plan view of the Vortex Mixer showing velocity profiles in the Fermenter Sludge feed channels and Elutriation influent conduits is shown in Figure 2. Path lines from the feed sources (i.e. First and Second Fementer Sludge Channels and the Elutriation Feed pipe) are shown in elevation view in Figure 3. This illustrates a high degree of interpenetration and suggests that effective mixing of the feed streams can be expected.

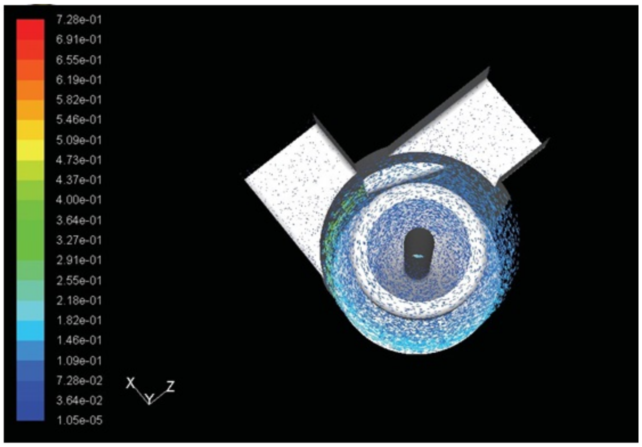

Figure 2. Plan view of Velocity Profile in Vortex Mixer

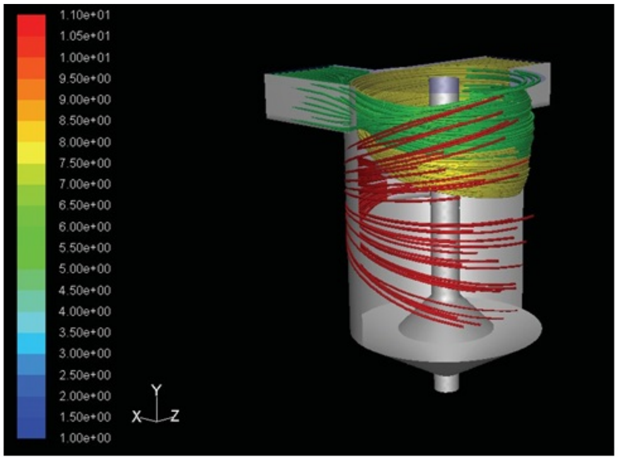

Figure 3. The path-lines for the Elutriation (shown in red)

Figure 4 which shows the sludge concentrations at average flows highlights that effective mixing is predicted to occur in the unit. The simulated mixed sludge concentration at the outlet for the average flow conditions for the initial design simulations is 1.30% dry solids and 1.00% for the revised simulations. The simulation results

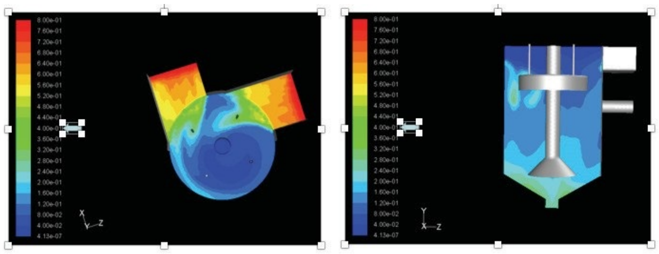

Figure 4. Sludge Concentrations in the Upper Sections of Mixer (plan view on left) and Elevation View through Vessel Body (on right) showing concentrations at less than 2% dry solids at outlet.

for the minimum, average and maximum flow conditions all show sludge concentrations less than the maximum specified concentration of 2% for the outlet; suggesting that the unit as designed should meet the performance objectives at all states of in-service flows.

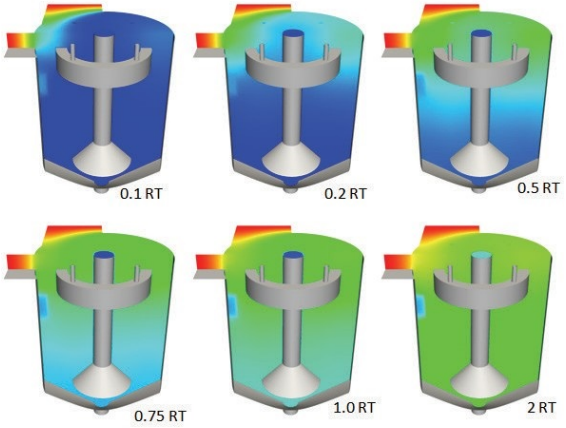

Figure 5. Scalar tracer equivalent to step feed (Residence Times are listed for each image).

Modelling a scalar transport function fed with a step input indicates that the unit achieves homogeneity in less than three residence times (Figures 5). Introducing a scalar is the computational equivalent to placing a dye tracer within the system. Although this later simulation is an oversimplification of the sludge mixing behaviour and disregards the viscous properties of sludge, it does help visualize the mixing zones within the device. This approach gives a rough estimate of mixing times required, particularly when examining relatively dilute mixtures. Even more accurate predictions could be achieved using a time-transient multiphase model. However, this approach is very expensive computationally and was determined to be beyond the scope of the project at the time. The simulations show over 99.6% mixing achieved within three residence times.

2.7 Results of Field Testing of Installed Unit

Following fabrication and installation (see Figure 6), the Hydrodynamic Vortex Mixer unit was tested in the field in-situ to verify its performance. Based on average plant flows and the symmetry of the process streams the test was conducted on one treatment train only. The Fermentation Tank was allowed to fill and reach steady state for a period of 5 days prior to conducting the tests. The unit was left unmixed for 1 hour and then the mixer was run continuously for 15 minutes prior to taking the samples. The mixer was run during the duration of each sampling event to ensure homogenous flows during the test.

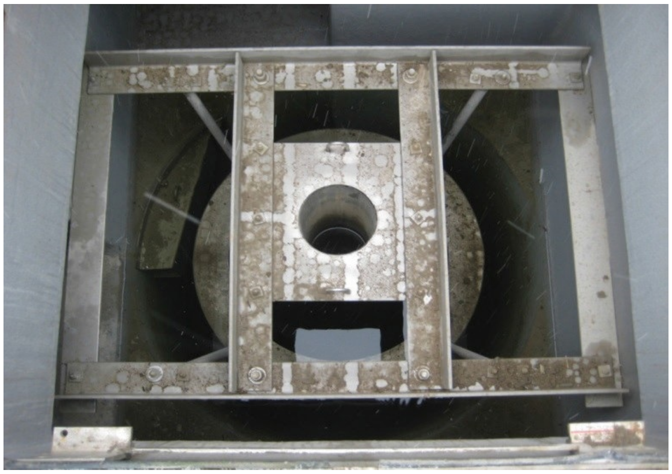

Figure 6. The Installed Hydrodynamic Vortex Mixer Unit

Table 3. Results from the field test of the mixer.

Event	Primary Sludge (gpm)	Overflow Total (gpm)	Primary Flow Conc.	Ferment. Overflow Conc.	Mixer Discharge Conc.	Total Sludge	Total Flow (gpm)	Calculated Sludge Conc.
1	600	27	0.18%	2.11%	0.19%	2.2	627	0.26%
2	600	27	0.22%	1.98%	0.29%	2.41	627	0.30%
3	300	27	0.15%	2.23%	0.43%	1.65	327	0.32%
4	300	27	0.22%	2.09%	0.54%	1.79	327	0.38%
5	160	27	0.33%	2.05%	0.21%	1.63	187	0.58%
8	600	54	0.25%	1.98%	0.38%	3.66	654	0.40%
7	300	54	0.30%	2.12%	0.66%	3.17	354	0.57%
6	160	54	0.15%	2.09%	0.44%	2.5	214	0.64%
		Sample Avg:	0.23%		2.08%	0.39%	Calculated Avg:	0.43%

Samples taken from the underflow (Mixer Discharge) matched the theoretical concentration (Calc. Sludge) within 0.04

3. CONCLUSIONS

Computational Fluid Dynamics (CFD) is a computer simulation tool that has great utility in the water, wastewater and environmental technology industries. As computational power becomes less expensive, CFD studies are becoming fundamental in the design of innovative urban water management and treatment systems. This is especially true when new or adapted technologies are in development, such as the hydrodynamic vortex mixer described in the paper.

CFD has been found to be a powerful tool for the evaluation and comparison of the various types of Environmental Technologies with CFD outputs being corroborated with physical testing and video imaging resulting in insights that are facilitating improved understanding of device and system characteristics; leading to improvements and rapid innovations (Edgarr, 2005; Edgarr et al., 2005; Faram and Andoh, 1999; Faram and Andoh, 2000; Jarman et al., 2007; Jarman et al., 2008).

The paper has described the use of CFD for the development of a low energy non-powered Vortex based Mixer utilized to blend sludge streams and fermented liquors as part of a biological nutrient removal wastewater treatment plant upgrade. CFD predictions are corroborated with post commissioning performance monitoring testing.

There are several CFD outputs which may be good performance indicators for mixing behaviour. For example, the interpenetration of flow path-lines, scalar tracers, and multi-phase modelling all have the potential to provide insights and estimates of the extent and timing of homogeneous mixing.

Due to its unique hydrodynamic flow regime and resulting extended residence times within a compact footprint, an advanced hydrodynamic vortex vessel makes an efficient and cost-effective mixer for water and wastewater applications. These mixing properties can be assessed using CFD simulations, and have been reinforced by results of tests conducted on full scale installations in the field.

On the basis of experiences gained and the practical usefulness of CFD, it is recommended that an Applied Computational Modelling Research Centre of Excellence is established at the University of Energy and Natural Resources (UENR) in Sunyani, Ghana. It is envisaged that the resulting expertise developed could form the basis of an incubator style consultancy geared towards the commercialization of the resulting knowledge and expertise and ultimately assist in CFD becoming a more widely deployed tool.

References

[1] ANDOH, R.Y.G. 2006. CFD Saves $50,000 in design of stormwater separator. Fluent News, Vol. 15, Issue 2: 35-37. EGARR, D.A., FARAM, M.G., O'DOHERTY, T., PHIPPS, D.A. AND SYRED, N. 2005. Computa-

tional fluid dynamic prediction of the residence time distribution of a prototype hydrodynamic vortex separator operating with a base flow component. Journal of Process Mechanical Engineering, Proc. Instn. Mech. Engrs, Vol 219, Part E: 53-67.

[2] EGARR, D.A. 2005. Studies of fluidic systems for environmental applications. PhD Thesis, Cardiff University, UK.

[3] FARAM, M.G., AND ANDOH, R.Y.G. 1999. Evaluation and optimisation of a novel self-cleansing combined sewer overflow screening system using computational fluid dynamics. 8th International Conference on Urban Storm Drainage. Sydney, Australia, August-September: 1107-1115.

[4] FARAM, M.G., AND ANDOH, R.Y.G. 2000. Application of simulation and predictive techniques for the evaluation of hydrodynamic separators. Wastewater Treatment: Standards and Technologies to Meet the Challenges of the 21st Century, CIWEM/AETT Millennium Conf., Leeds, UK, 4-6 April: 223-230.

[5] ANDOH, R. Y. G., EGARR, D AND FARAM, M. G., 2008, "High-rate Treatment and Disinfection of Combined Sewer Overflows using Advanced Hydrodynamic Vortex Separators", 11th Int. Conf. on Urban Drainage, Edinburgh, UK, 31 August – 5 September.

[6] JARMAN, D. S., FARAM, M. G., TABOR, G. AND BUTLER, D., 2007, "A Review of the Opportunities Presented Through the Application of Computational Fluid Dynamics (CFD) to Water Management Challenges", Water Management Challenges in Global Change: CCWI2007 and SUWM2007 conference, De Montfort University, Leicester, UK, 3-5 September.

[7] JARMAN, D. S, FARAM, M. G., BUTLER, D., TABOR, G., STOVIN, V. R., BURT, D. AND THROP, E., 2008, "Computational Fluid Dynamics as a Tool for Urban Drainage System Analysis: A Review of Applications and Best Practice", 11th Int. Conf. on Urban Drainage, Edinburgh, UK, 31 August – 5 September.