IJSAR, 3(4), 2016; 18-24

International Journal of Sciences & Applied Research

www.ijsar.in

Analyzing performance of sewage treatment plant through computational fluid dynamics

Etikaf Hussain*, Shoaib Ahmed, Hadiqa Maqsood and Syed Imran Ahmed

Department of Civil Engineering, NED University of Engineering & Technology, Karachi, Pakistan.

Correspondence Address: *Etikaf Hussain, Department of Civil Engineering, NED University of Engineering & Technology, Karachi, Pakistan.

Abstract

Numerical simulations of fluid flow can enhance mathematical accuracy, understanding, and provide better visuals. For this study, Computational Fluid Dynamics (CFD) method is used to analyze and validate unconventional designed units of wastewater treatment plant at Defence Housing Authority (DHA) Phase VIII, Karachi, Pakistan. The modeling is performed on Upward Anaerobic Sludge Blanket (UASB) reactor, aeration tank, and clarifier tank, respectively. AutoDesk Simulation CFD is used for this study where 3D models of each design treatment unit are generated to illustrate behavior of variables including turbulence intensity, velocity, local mean age, and gas hold up. Results do not show any uncertainty or disturbance for the units in following basic hydraulic principles. In the UASB reactor and aeration tank, baffles and air diffusers, respectively, are creating turbulence and velocity variation which is affirmative for the tank to carry out treatment process. Clarifier tanks show constant movement of water due to scrappers, followed by less local mean age and smooth movement toward disinfection tank. The overall modeling of treatment plant units concludes toward good performance; henceforth, approved for installment without any redesign or modifications.

Keywords: Sewage treatment plant, mathematical modeling, computational fluid dynamics, AutoDesk Simulation CFD

Introduction

Mathematical equations for fluids are complex and difficult to solve water-oriented processes. However, by using computational tools and modeling techniques, results from the fluid equations can be attained quickly with better visualization. Computational Fluid Dynamics (CFD) is a method to analyze various responses of transport of fluid, and other processes by using numerical formulae. Some of the purposes include validation of a design concept and manufacturing process and performance of fluid based processes. It, henceforth, eradicate massive/excess cost for modifications or redesign, need of costly prototypes and destruction tests, reduce in need of modifications in late manufacturing process, design time, and money (Chung, 2010). Modeling using CFD has many advantages over theoretical analysis or experiments as it is cheaper, quicker, repeatable, safer, and provide more information (Anderson and Wendt, 1995). Amini et al. (2013) performed experimental study and modeled simulation on bioreactor and elucidated the comparative results.

To perform CFD, there are various capable models such as ANSYS Fluent (ANSYS, 2011), Autodesk Simulation CFD (AutoDesk, 2013), OpenFOAM (CFD Direct, 2013), and EasyCFD(Antonio G.L., 2013). Amongst these, the most suitable model is chosen as per the requirements and data availability. Le Moullec et al (2010) performed a comparative study using systemic, compartmental, and CFD modelling on biological reactor for a wastewater treatment plant. Similarly, there have been various studies of CFD on sewage treatment plants that elucidates the significance of modeling. Bridgeman (2012) illustrated CFD modeling for sludge mixing for anaerobic digester where biogas yield and velocity gradient are drawn as results. Fayolle et al. (2007) performed CFD to predict oxygen transfer in aeration tanks and Xanthos et al (2013) assessed CFD for clarifier tanks. Lee (2005) and Wu et al. (2007) also simulated the flow pattern of a clarifier with a porous medium as a sludge blanket via 3D CFD. Wu et al. (2008) was the first work to utilize 3-D, multiphase flow simulation for a clarifier. This study attempts to improve effluent water quality of a clarifier by altering its geometric construction. As done by various researchers, CFD tends to develop the significance related to validation and prediction of treatment plant units.

Materials and methods

The model runs on various complex equations that include Navier-Stokes, equation of continuity, equation of momentum and energy equation. In the first phase, geometric modification is done for every unit using variables including gravity, flow axes, initial and final boundary conditions, environmental conditions, fluid properties, and material properties. The computer algorithm runs number of iterations until it meets the convergence criteria (Figure 1).After reaching stability, the systematic methodology of CFD modeling include development of 3D model, creation of 3D mesh using geometry, data input, and simulations. For this study the variables simulated are velocity magnitude, local mean age, and turbulence intensity.

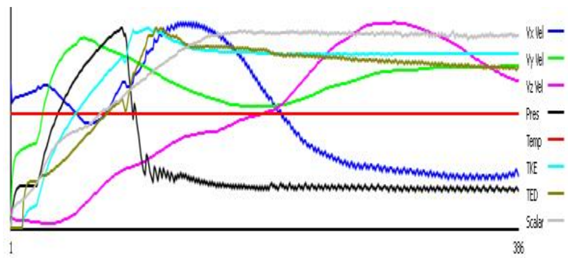

Fig. 1: Convergence plot showing iterations.

Velocity magnitude determines the speed of sewage (carrying sludge and suspended particles) flowing in the tank and their behavior within the unit. Velocity effects the settlement of sludge in clarifier and efficiency of aerations tanks and UASB reactor. Local Mean Age (LMA) determines the duration of time spent (in seconds) by wastewater at different regions of the container/tank of the respective units. This variable is significant in terms of identifying sludge settling zones. The turbulence intensity signifies turbulence in unit with 1% or less considered as low and greater than 10% as high. For this study, the only known variable in the design had been retention time. Model is calibrated and validated using retention time dataset for each unit.

The geometric design of unconventional UASB reactor comprises of a reinforced concreterectangular tank of 30.9m x 16.9m with 300mm of mid partitioning wall. There are eight baffles of 150mm having concrete haunching at bottom ends. At this cell,a pump collects the settled sludge and water is transferred to the next unit (i.e. aeration tank). There are two aeration tanks in the plant of size 45m × 8m × 4.5m, with free boarding of 0.5 meters. There are 500 nozzles, 250 in each tank. Outlet of this unit is inlet to the secondary clarifier.

Secondary clarifier comprises of two units and a disinfection tank. The depth of water of each tank is 3.5m with 0.5m of free boarding. There is 1000mm of haunching and each tank has a scrapper comprising of 11 suction nozzles for collecting settled sludge.

Calibration is the process of aligning the model to replicate the real time performance with its simulation. In this context, the model is calibrated using retention time as to bring simulated retention time close to the design retention time. Table 1 shows the designed vs. simulated retention time.

Table 1: Calibration and validation of CFD model using retention time.

	Sec. Clarifier	Aeration Tank	UASB Reactor
Average simulated Retention time(hrs)	2.93	6.388	2.4
Designed retention time (hrs)	3.00	6.300	2.0

Results and discussion

1. Upward Anaerobic Sludge Blanket (UASB) Reactor

Figure 2 demonstrates CFD simulated flow pattern by wastewater along with the velocity magnitude. It shows that the inlet of UASB reactor has velocity range from 324 m/hr to 400 m/hr as defined on scale. As water travels along the reactor the velocity decreases to a magnitude of 150-250 m/hr. The illustration also depicts that near the inlet there is no regular flow pattern but becomes regular as it travels further.

Figure 3 show the variation in velocity of particles along the flow path within the UASB reactor. Baffle walls tend to decrease the velocity of the incoming fluid where as high velocity zone in developed when fluid enters the second chamber after traveling 35.5m.

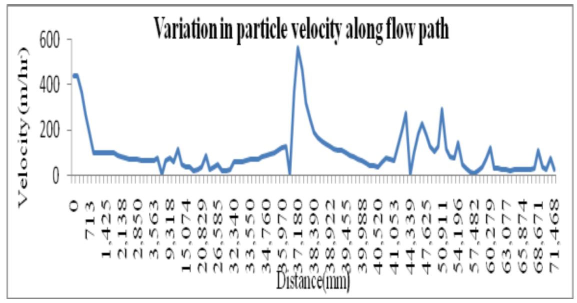

Fig. 2: Variation in velocity of fluid traces along the flow path.

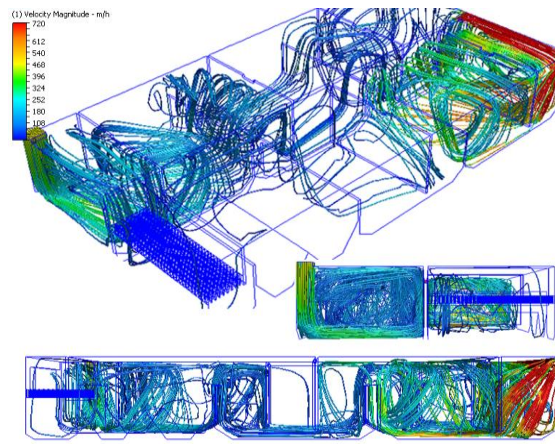

Fig. 3: Flow traces of UASB Reactor.

Figure 4 show the variation in velocity of particles along the flow path. For better visuals, results are shown from the top view as colored filled contours with vectors. The outlook is sectioned into three planes (top, mid plane and bottom of the reactor). Figure 4 elucidates that baffles tend to change the direction of flow illustrating swirl patterns.

Sludge particles are also simulated for UASB reactor in which the diameter of the particles are set as 0.003 mm and the density is taken as 2.63 kg/m^3. Figure 5 shows the movement of sludge particles which is higher at the inlet and slows down as it travels through the reactor. When the particles move to the second chamber of the reactor their velocity increases up to 100 mm/sec. Suspended solids settles in between.

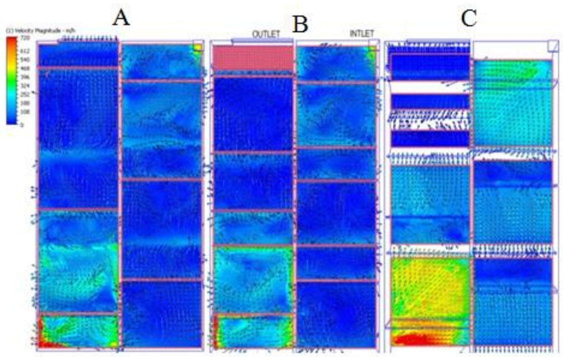

Fig. 4: Velocity magnitude of UASB Reactor. Top (A), Mid (B), Bottom (C).

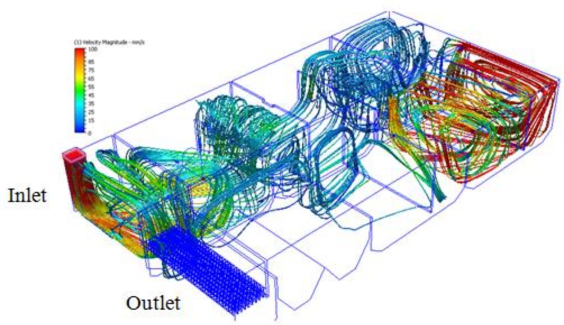

Fig. 5: Pattern of sludge particles for UASB Reactor.

Figure 6 shows results for LMA that water spends very less time (between 10 – 15 seconds) at the inlet section as compared to the time spent at further sections of the reactor. Figure 6 also concludes that water spends less time at the bottom level as compared to water at the surface which spends comparatively more time at one point.

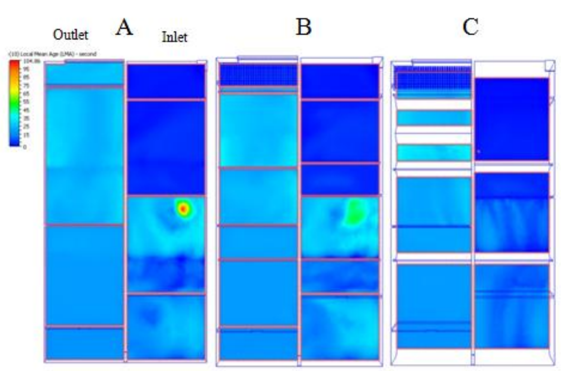

Fig. 6: LMA for UASB Reactor. Surface (A), Middle (B), Bottom (C).

Simulation is carried out for turbulence intensity (Figure 7) where results depict that baffles create turbulence in the flow. Turbulence intensity is higher at the bottom level up to 1.5; whereas, in the middle level, it is comparatively low (~0.9). There is not much variation in turbulence intensity since water flows steady.

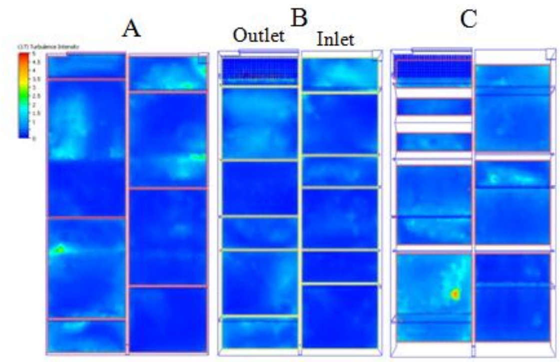

Fig. 7: Turbulence intensity for UASB Reactor. Surface (A), Middle (B), Bottom (C).

2. Aeration Tank

Figure 8 is an orthogonal view of aeration tank where blue dots represent air. The traces show velocity magnitude of water that is high at inlet and decreases as particles are dispersed.

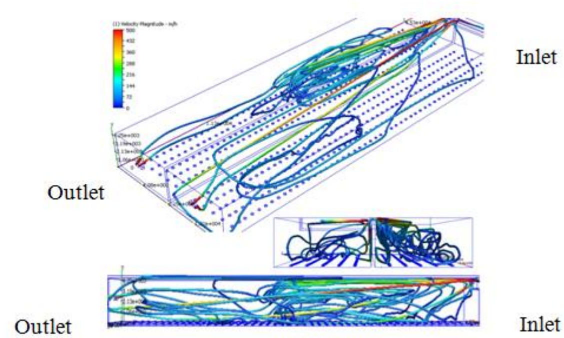

Fig. 8: Flow traces in aeration tank.

In the second tank, velocity ranges from 500 m/hr to 100 m/hr (Figure 9). The middle level of the tanks which comprise of outlets show high velocity at the outlets increasing up to 560 m/hr. At the bottom of the tank the overall velocity did not exceed 250m/hr. It can be concluded that velocity is higher at inlets and outlets of the tanks which supports the applied laws of physics in fluid movement, and the unit is determined to stable.

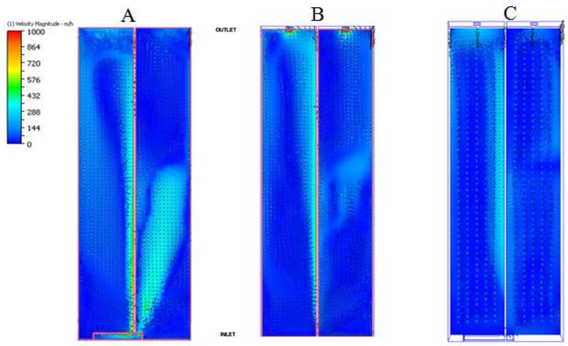

Fig. 9: Velocity magnitude of aeration tank. Surface (A), Middle (B), Bottom (C).

Figure 10 is an illustration of LMA in the aeration tank, defined in seconds. All three planes depict dead zones corners of the tank. However, left corner near the inlet shows even higher value of about 8 seconds. At the bottom, LMA did not exceed beyond 1 second, which is due to the diffusers present at the bottom.

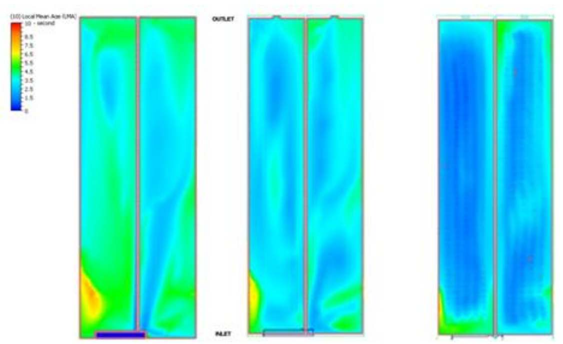

Fig. 10: LMA of aeration tank. Surface (A), Middle (B), Bottom (C).

Another variable for which results are driven is turbulence intensity (Figure 11). The surface level shows that water enters with higher turbulence intensity ranging from 7.5 to 9, and gradually decreases up to 3. The bottom level of the tank depicts red and yellow colors of the scale, which means high instability of water particles. This is justified by the vectors shown in Figure 9 and LMA results, that water does not spend more than one second. This behavior of water at the bottom is due to the disturbance caused by air diffusers.

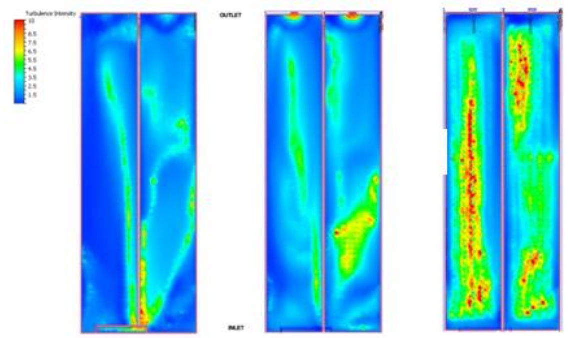

Fig. 11: Turbulence intensity of aeration tank. Surface (A), Middle (B), Bottom (C).

3. Secondary Clarifier

Flow traces are illustrated in Figure 12 that shows particles passing through the scrapper, with velocity decreases from 178 mm/s to 50 mm/s as it acts as a barrier. It also shows maximum velocity of water when it enters the disinfection tank and remains there until it changes direction. Clarifier shows almost 1 mm/s velocity magnitude near the outlet.

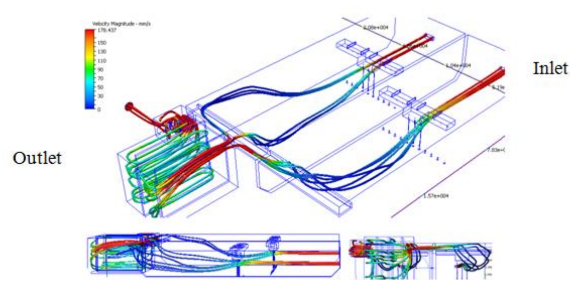

Fig. 12: Flow traces in clarifier.

Figure 13 illustrates LMA of clarifier that shows more LMA at bottom of the tank. The disinfection tank depicts a constant movement of flow where water does not spend more than 8 seconds at any point. Surface of the tank shows that water spends more time at the adjacent corners of the outlet, around 12-15 seconds, which is contrary to middle sections and inlet (6-9 seconds).

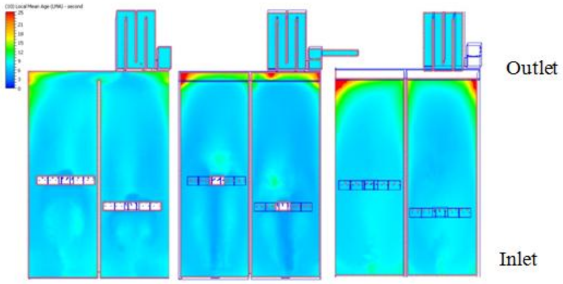

Fig. 13: LMA of clarifier. Surface (A), Middle (B), Bottom (C).

The third variable for which simulation is carried out is turbulence intensity. Figure 14 depicts that towards the end of the tank turbulence is less than 1, concluding that there is less disturbance between particles. The surface of tank near inlet illustrates relatively higher turbulence of about 4 to 4.5 on the scale. Turbulence intensity for the rest of the tank lies between 1 and 2.5 which is suitable for smooth operation. It increases as water enters the disinfection tank. These results justify the vectors discussed in Figure 13, as they represent similar motion.

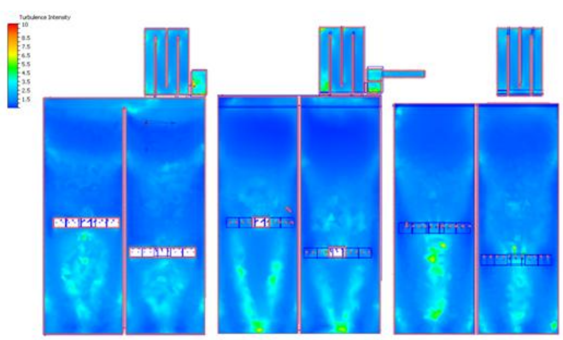

Fig. 14: Turbulence intensity of clarifier. Surface (A), Middle (B), Bottom (C).

Conclusions

The results of the CFD study for unconventional units of wastewater treatment plant to verify the design has the following findings:

- The designed retention time of UASB reactor was 2 hours and the simulated retention time was 2.4 hours which shows that the design of the reactor is adequate enough. From the modeling results, it is demonstrated that velocity is higher at the baffle openings and turbulence is higher at the inlet, outlet, and U-turn between chambers which shows the satisfied behavior of fluid for this unit.
- The designed retention time for aeration tank was 6.3 hours and simulated retention time was 6.338 hrs. It is concluded that diffusers provide adequate air into the tank for mixing. Velocity and turbulence are higher at inlet and outlets due to pressure and backwater effect that support design accuracy.
- Designed retention time for clarifier was 3.00 hours and simulated retention time was 2.93 hours which validated the mode and shows that clarifier is designed adequately. Simulation results demonstrate that velocity is higher at the inlets of clarifier and disinfection tanks, and LMA is higher at the bottom of outlet of clarifier tank that follows basic treatment plant physics and makes the design suitable for execution.

The overall modeling results show that the suggested design is satisfactory and suitable for achieving the objective of the designed plant.

References

1. Amini, E., Mehrnia, M. R., Mousavi, S. M., and Mostoufi, N. 2013. Experimental study and computational fluid dynamics simulation of a full-scale membrane bioreactor for municipal wastewater treatment application. Industrial & Engineering Chemistry Research, 52(29), 9930-9939.
2. Anderson, J. D. and Wendt, J. 1995. Computational fluid dynamics (Vol. 206). New York: McGraw-Hill.
3. ANSYS. 2011. ANSYS Fluent User's Guide. ANSYS, Inc., US.

4. Antonio G. L. Assessed in 2013. Easy CFD. http://www.easycfd.net/index_files/Page551.htm
5. AutoDesk. Assessed in 2013. AutoDesk Simulation CFD. Autodesk product.
6. Bridgeman, J. 2012. Computational fluid dynamics modelling of sewage sludge mixing in an anaerobic digester. Advances in Engineering Software, 44(1), 54-62.
7. CFD Direct. Assessed in 2013. OpenFOAM User's Guide. OpenFOAM Foundation.
8. Chung, T. J. 2010. Computational fluid dynamics. Cambridge university press.
9. Fayolle Y., Cockx, A., Gillot, S., Roustan, M., and Heduit, A. 2007. Oxygen transfer prediction in aeration tanks using CFD, Chemical Engineering Science, 62, 7163 – 7171.
10. Le Moullec, Y., Gentric, C., Potier, O., and Leclerc, J. P. 2010. Comparison of systemic, compartmental and CFD modelling approaches: application to the simulation of a biological reactor of wastewater treatment. Chemical engineering science, 65(1), 343-350.
11. Lee, T. H. 2005. Modeling of the Blanket Clarifier of Water Treatment Plant, Chapter 4, M.S. Thesis, Tamkang University, Taiwan.
12. Wu, R. M., T. H. Lee, and W. J. Yang. 2008. A Study of Flow in a Blanket Clarifier Using Computational Fluid Dynamics. J. Environ. Eng.-ASCE.
13. Wu, R. M., T. H. Lee, and W. J. Yang. 2007. A Study of Water Treatment Clarifier, Tamkang J. Sci. Eng., 10, 335.
14. Xanthos, S., Ramalingam, K., Lipke, S., McKenna, B., & Fillos, J. 2013. Implementation of CFD modeling in the performance assessment and optimization of secondary clarifiers: the PVSC case study. Water Science & Technology, 68(9), 1901-1913.