CFD in Water Systems

Mikhail P. Strongin
Xylem Inc., 8200 Austin Ave, Morton Grove, USA
mikhail.strongin@xyleminc.com

Keywords: CFD; multiphase; UV disinfection; reactions

The main goal of this presentation demonstrates the potential of CFD modeling of the complex systems, which can include the turbulent flow with rotation, mixing, chemical reactions, and radiation. Some examples that modeling are presented in this work.

1. The investigation of mixing and chemical reactions processes of water with other liquid in a pump.
2. The achieving efficiency improvement in the range of 2 – 30% was reached using multi objective optimization approach for pump efficiency improvement.
3. CFD models of bioreactors study was done for gas – liquid mixture including bubble size distribution and chemical reactions.
4. CFD model the ultra violet (UV) disinfection process used in the water industry (as a part of water treatment system).

Results and Conclusions

The systems with gas-liquid mixing can be modeled [3]. In this example (a part of pump system), air is introduced at the bottom of a mixing tank. Air bubbles through the liquid in the tank and exits at the free surface while the paddles are rotating. During this process, oxygen can get absorbed in the liquid. Figure 1(a) shows a CFD setup with sparger at the bottom and two mixing paddles. The high and low velocity areas as well as air volume fractions can be visualized as a function of air injection rate and the paddle speed.

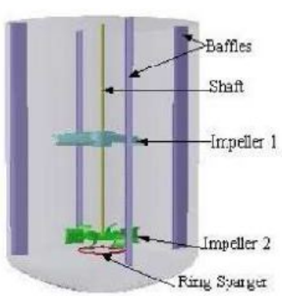
Figure 1(b) below shows volume fraction of air in the tank at different times. In the beginning, while paddles are rotating, air starts coming in through the bottom sparger. As time progresses, more air fills the space and its volume fraction starts to increase I the tank. Air that reaches the top can escape the tank resulting in free surface perturbations at the free surface. The air space above the free surface is also modeled to track the free surface properly.

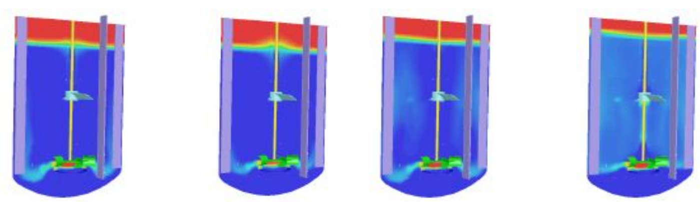
Other bioreactors can also be modeled similarly [3]. Apart from the stirred tank type shown above, airlift types are also popular due to their simple design. Figure 2 shows an example of an airlift reactor modeled using ANSYS FLUENT's Eulerian multiphase model along with population balance model. This allows us to compute bubble size distribution and gas hold up in different parts of the reactor. The breakup and coalescence of bubbles is also computed as this is an important aspect in determining the correct interfacial areas for heat and mass transfer processes. Figure 2(a) shows the air-water interface. Bubble size distribution and oxygen concentration are shown in 2(b) and 2(c) respectively. This type of analysis helps in locating regions with maximum and minimum gas hold up.

CFD can also be used to model the ultra violet (UV) disinfection process used in the water industry (as a part of water treatment system) [3]. Monochromatic or polychromatic lamps are installed in the duct system. The UV light emitted by these lamps is used to neutralize the water-borne pathogens such as E. coli and C-sporidium [3]. A representative example is shown in Figure 3 below where the possible paths of the bacteria are shown through the water duct system

(3(b)). As the pathogens pass near the lamp and go through its radiation field, they get enough dosage that changes their DNA and hence neutralizes them rendering the water pathogen free. The CFD code FLUENT has a radiation module that computes the strength of this UV radiation field around the lamps (shown in 3(a)) while the core fluid flow solver predicts the residence time [4]. The combination of residence time and radiation field strength (depends on the water turbidity or water transmittance) then provides a means to compute the effectiveness of this kill process.
Figures 4(a,b) illustrate the CFD ability to predict characteristics of ozone systems to disinfect water. It is easy to see the dependence of this system performance on the inlet boundary conditions.
Figure 5 shows an influence of chemical reaction on the pump efficiency performance.
Design Explorer module in ANSYS workbench can evaluate various design and optimize the system based on user specified objective functions. This makes the design process quick as less iteration are needed to arrive at the final optimized design [4].

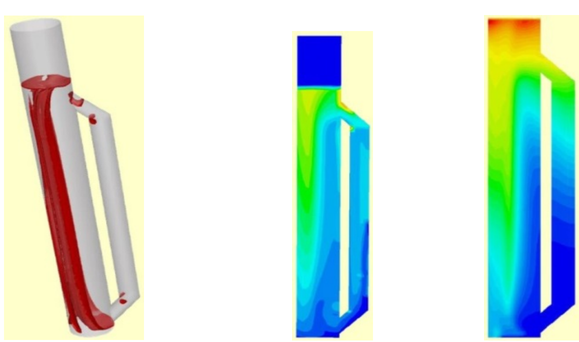
References:
1. S. Dutta; Y. Catano; and etc. (2010), *Computational Fluid Dynamics (CFD) Modeling of Flow into the Aerated Grit Chamber of the MWRD's North Side Water Reclamation Plant, World Environmental and Water Resources Congress 2010.*
2. M. P. Strongin, (2010), *CFD Modeling of Influence of Weak Catalytic Reactions on Pump Performance Parameters, Proceedings of the IMECE2010, IMECE2010-37149*
3. Mikhail P. Strongin and Muhammad Sami (2014) *Computational Simulations in Water Application, Proceedings of the IWA World Water Congress & Exhibition 2014*
4. M. Siddiqi, P. Boppa, M.P. Strongin (2012), *Multi-parameters pump impeller optimization, Proceedings of the ASME 2012 Fluids Engineering Summer Meeting, FEDSM2012-72091.*





1 sec 2 sec 4 sec 10 sec

Figure 1(a). Figure 1(a). Air Volume Fraction in a tank from zero (blue color) to 1 (red color)



(a) (b) (c)

Figure 2. Water-air interface (a), bubble size distribution (b) and Oxygen Concentration in (c)

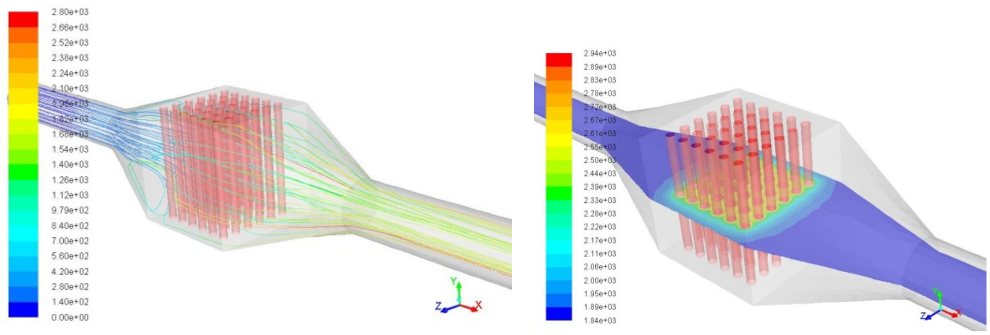
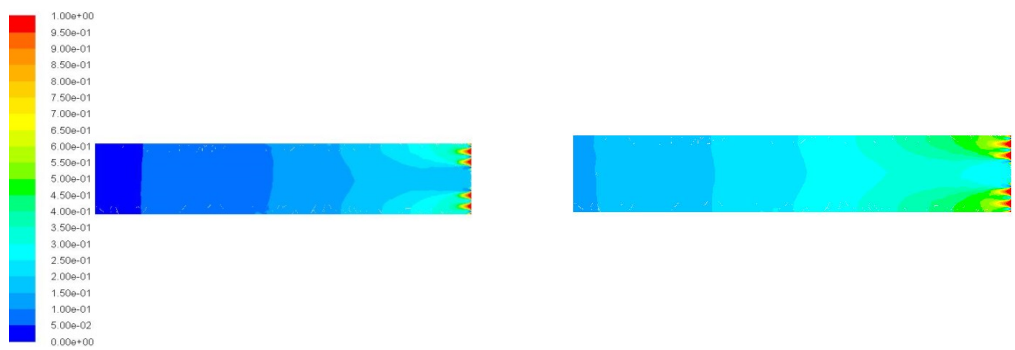


Fig 3(a). UV Radiation Field Strength (W/m2) Fig 3(b). UV Dose on bacteria (sample of tracks)



a **b**

Figure 4. Contours of mass fraction of pollution (**a -** M_0 mass flow rate ; **b** – $2M_0$ mass flow rate) ozone flow rate in both case was the same.

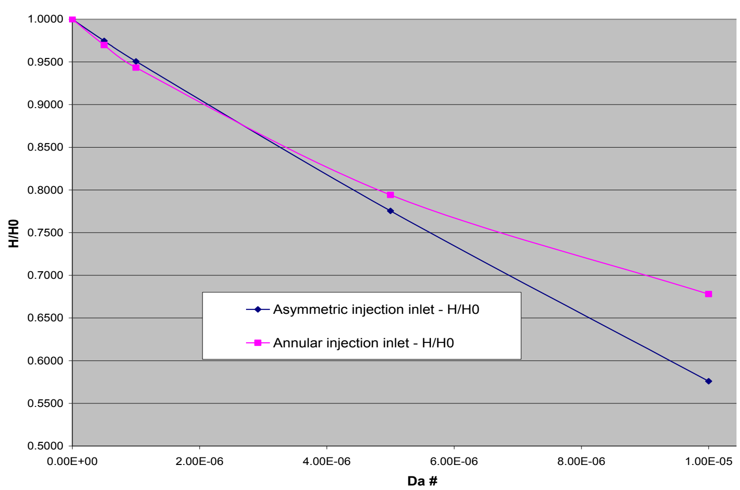


Figure 5. The relative head (H/H0) as function of Da number