9th IWA Symposium on Systems Analysis and Integrated Assessment
14-17 June 2015, Gold Coast, Australia

Watermatex 2015

CFD in Water Systems

Mikhail P. Strongin

*Xylem Inc., 8200 Austin Ave, Morton Grove, IL 60053, USA

mikhail.strongin@xyleminc.com

INTRODUCTION

The goal of using CFD in any industrial application is to reduce and ideally eliminate the prototyping process when designing new, highly efficient products [1,2]. The rapidly growing availability of computational resources and improved physical models turns CFD from facilitating tool to a powerful instrument of innovation including but not limited to mathematical tools, multi-physics models, and parametric optimizers.

CFD may dramatically change the traditional design process to create efficient water and wastewater systems and their components such as pumps, valves, pipe grids, filters, membranes, etc. Depending upon the complexity of the system, physics, and topology, different commercial CFD codes are available for solving practical problems.

In spite of a century of research about cavitation, the detailed mechanism is still not fully understood. Due to that, direct CFD calculation parameters that are dependent on cavitation, such as the net positive suction head (requirement) (NPSH(r) are almost impractical. Simplified cavitation models are needed to effectively use CFD models.

An example of a CFD application is the mixture process in water flow [2]. This can be helpful for developing efficient filter systems connected to the pump because they may be highly sensitive to mixture uniformity. Another example included in this presentation is the CFD analysis for the UV disinfection process as part of a water treatment system.

The main goal of this presentation is to demonstrate a wide range of CFD functionality for modeling the complex systems, which can include the turbulent flow with rotation, mixing, chemical reactions, and radiation.

Some examples of 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).

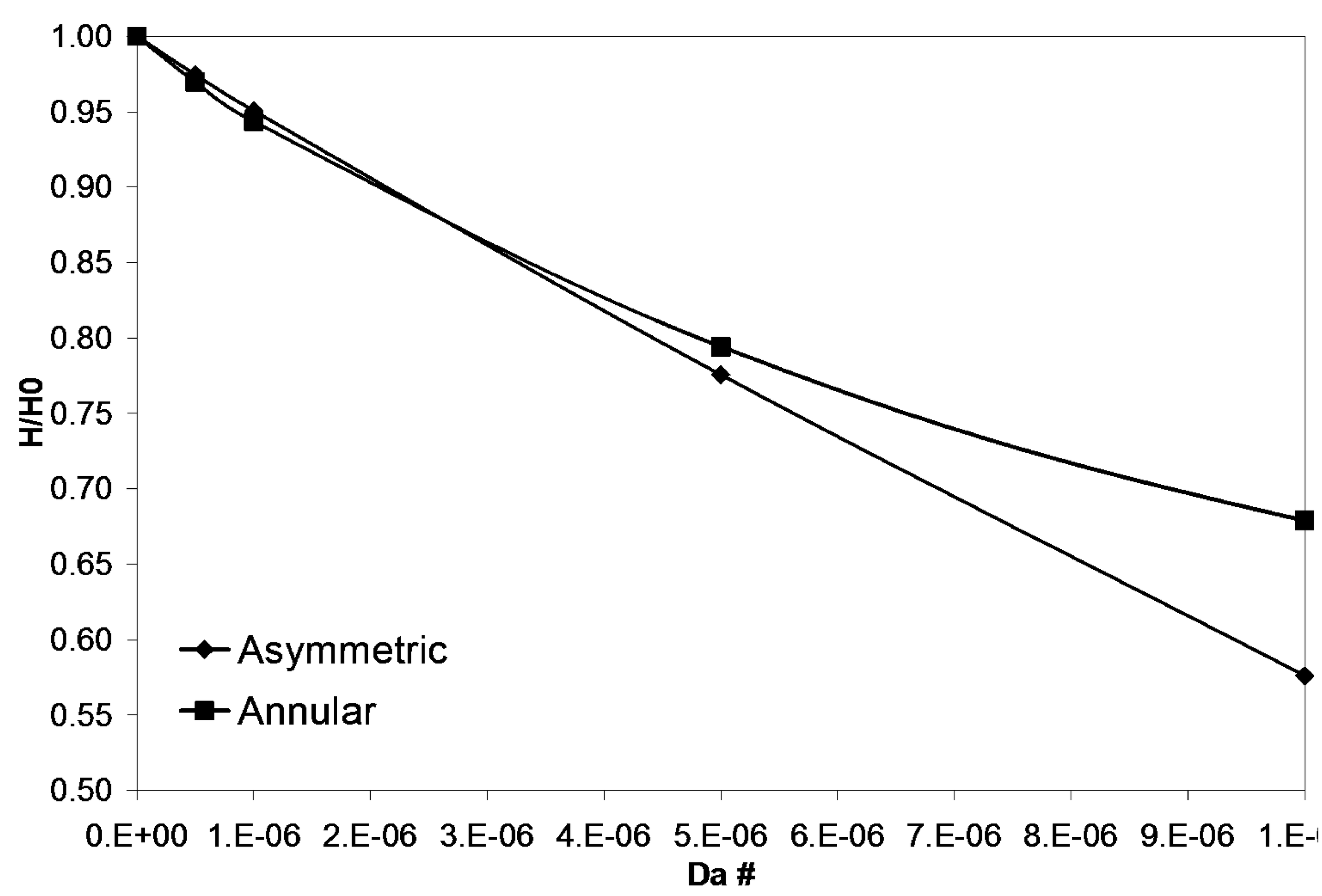

Figure 1. The relative head outlet (H/H0) as function of Da number

RESULTS and DISCUSSION

In a lot of cases, when dealing with water treatment applications, the mixing liquids are non-volumetric reactive. It is possible, however, that part of pump surfaces works as a catalysis, for instance , in case of coating impeller surfaces. For this event some wall reactions can take place.

An influence of weak catalytic reactions (Damkohler number **Da** $< 10^{-5}$) on pump performance parameters such a head and mixture uniformity was investigated in [3].

Figure 1 shows an influence of chemical reaction on the pump efficiency performance.

Results for pump head H of reactive cases relative to the head of non-reactive case H0 for different injection locations of secondary fluid on the pump inlet as a function of **Da** number are shown on the Figure 1.

Figure 2 displays that the test and calculated data have a reasonable agreement for NPSH(r). The difference lies in the 20 percent range, which is acceptable for a simple model [4] and can be used to quickly estimate the NPSH(r).

Figures 3(a,b) illustrates the CFD ability to predict characteristics of ozone systems to disinfect water. The model includes the chemical reactions between ozone and water pollutions. It is easy to see the dependence of this system performance on the inlet boundary conditions.

Other bioreactors can also be modeled similarly [5]. This airlift type is popular due to their simple design. Figure 4 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 4(a) shows the air-water interface. Bubble size distribution and oxygen concentration are shown in 4(b) and 4(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) [5]. 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 5 below where the possible paths of the bacteria are shown through the water duct system (5(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 5(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.

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 [6].

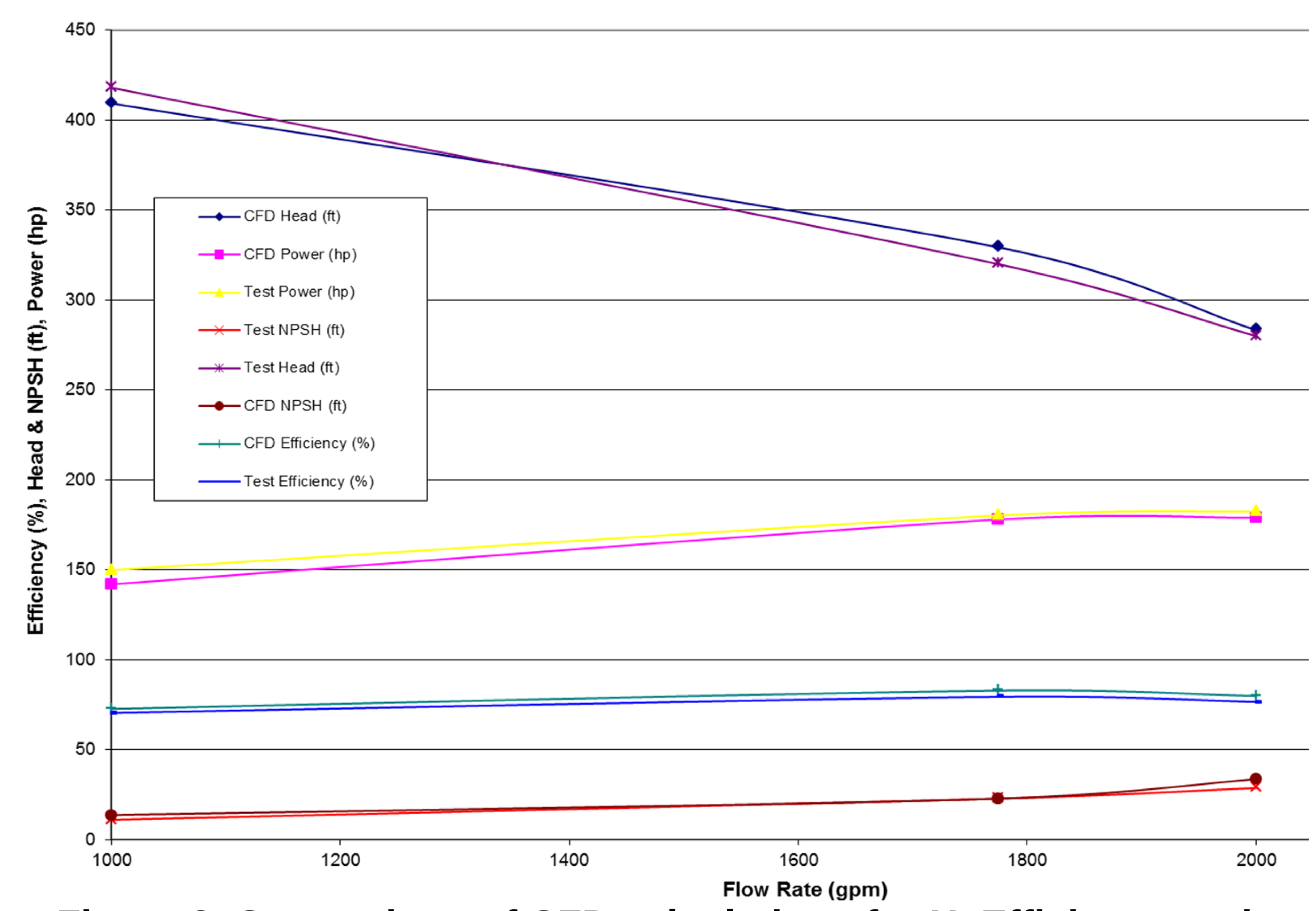

Figure 2. Comparison of CFD calculations for H, Efficiency and NPSHr vs. test data

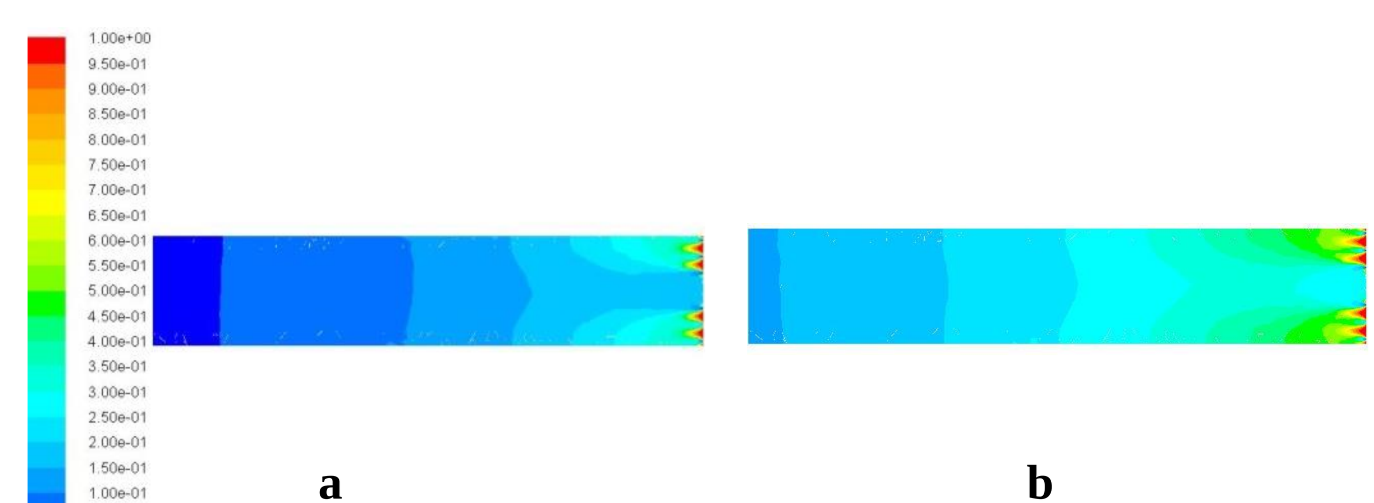

Figure 3. Contours of mass fraction of pollution (a - M_0 mass flow rate ; b – $2M_0$ mass flow rate); ozone flow rate in both case was the same

CONCLUSIONS

Strong influence of very weak wall chemical reactions on the pump performance has been shown. These results may be used for new material chosen for water treatment systems.

Simplified CFD models for the NPSH(r) calculation show acceptable agreement with test data for different types of pumps and may be used as a tool to shorten a design process for new pumps.

The combination of residence time of bacteria and radiation field strength enables one to calculate the effectiveness of the UV disinfection process.

The results of CFD calculations of characteristics of "ozone–polluted water mixture" show the CFD ability in optimization to disinfect water process.

The results presented in this work can help in water transport and water treatment design in a wide variety of applications.

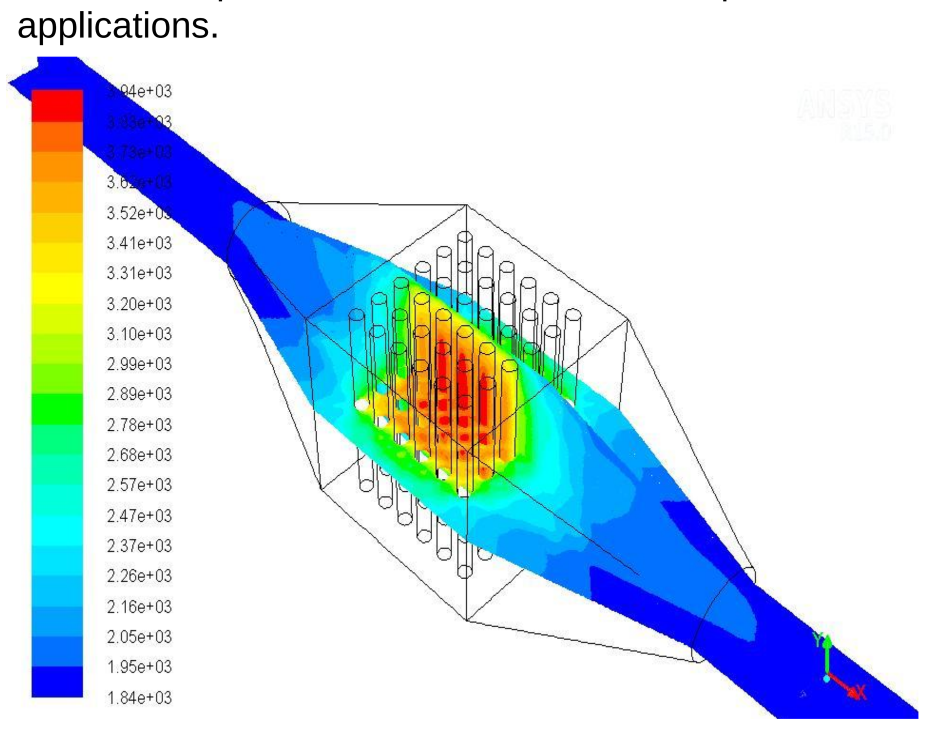

Figure 5(a). UV Radiation Field Strength (W/m2)

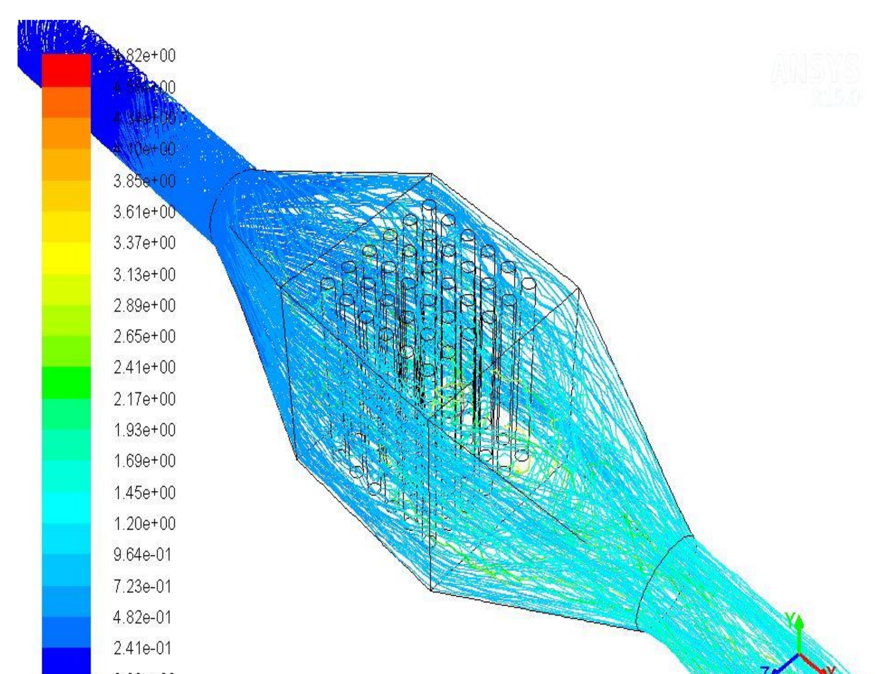

Figure 5(b). Residential time of bacteria (sample of tracks)

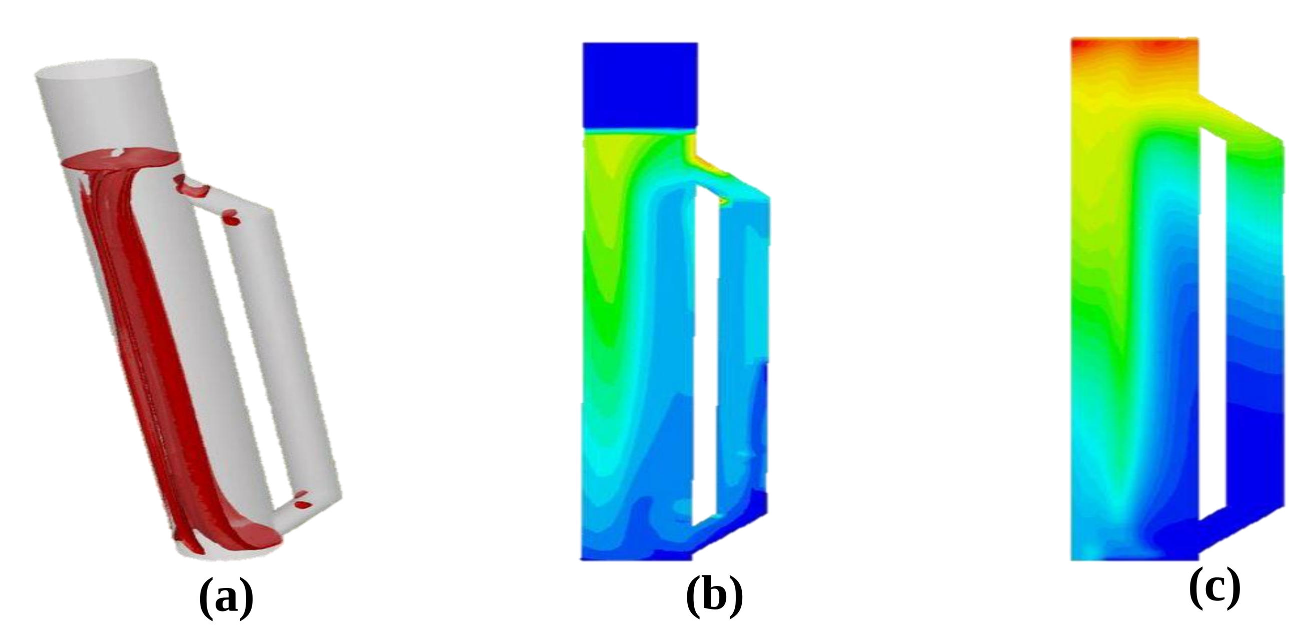

Figure 4. Water-air interface (a), bubble size distribution (b), oxygen concentration in (c)

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. Mikhail.P. Strongin (2009) Pump Mixing Process Calculations, *ASME Fluids Engineering Conference, FEDSM2009-78395*
3. M. P. Strongin, (2010), *CFD Modeling of Influence of Weak Catalytic Reactions on Pump Performance Parameters, Proceedings of the IMECE2010, IMECE2010-37149*
4. Mikhail Strongin (2011), *A Novel Approach to Simplification of NPSH(r) CFD calculation, Proceedings of ASME-JSME-KSME Joint Fluids Engineering Conference 2011 AJK2011-FED, AJK2011-06046*
5. Mikhail P. Strongin, Muhammad Sami, (2014), *Computational Simulations in Water Applications, IWA* World Water Congress*, Lisbon 2014, WWC2014 - 2478909.*
6. M. Siddiqi, P. Boppa, M.P. Strongin (2012), *Multi-parameters pump impeller optimization, Proceedings of the ASME 2012 Fluids Engineering Summer Meeting, FEDSM2012-72091.*