# Spring 2010 CFD Flocculator Simulation Validations

## Validating Computational Fluid Dynamics in ANSYS Fluent

### Introduction

CFD computer program created by ANSYS Fluent is used by AguaClara to attain solutions for water flowing in the flocculator. The solutions from CFD that are important for AgauClara are the pressure lost between the entrance and the exit of the flocculator, the turbulent kinetic energy in the flocculator and between each baffle, and the energy dissipation rate in the flocculator and between each baffle. AguaClara uses a hydraulic water treatment process and has developed design constraints based on dimensional analysis and relationships between equations governing the physics of fluid motion (fluid mechanics). The solutions from Fluent allow the flocculator to fit into rest of the water treatment plant design (Inlet tank and chemical dosing before the flocculator and the sedimentation tank, chlorine dosing, and distribution after the flocculator).

Verification and Validation study from the Fall 2009 CFD research determined that flocculator simulations in ANSYS Fluent are verified, but not validated. Fluent simulations of the flocculator are not validated because they do not accurately represent real fluid flow in a flocculator. The dimensionless pressure coefficient, $C_p = \dfrac{p - p}{\frac{1}{2}\rho v^2}$, is used to relate pressure values extracted from Fluent simulations of the flocculator and measured headloss in real flocculators. The $C_p$ of the Fluent simulation of the flocculator is greater than $C_p$ measured in flocculators, thus the pressure and velocities calculated by Fluent in sections of the flocculator in simulations are incorrect compared to real systems and not validated.

### Goals Spring 2010

Validate the Fluent simulation of the flocculator by having the simulation $C_p$ match $C_p$ from experimental measurements, by Julia Schoen on the PIV measurement project, and from measurements of headloss in the Agalteca flocculator (which has evenly spaced baffles like Fluent flocculator models).

Currently:

| Flocculator | CFD | PIV Experiment | Agalteca |
|---|---|---|---|
| $C_p$ | 3.5 | 2.22 | 1.96 |

### Validation Studies

Study to match Fluent with PIV experimental measurements and make a 3D flocculator

Validation of ANSYS Fluent Turbulence Models for Flocculator Simulations

### Presentations

Spring 2010 Final Presentation