CFD Flocculation Tank 3D Simulation

Computational Fluid Dynamics

Flocculation Tank Simulation

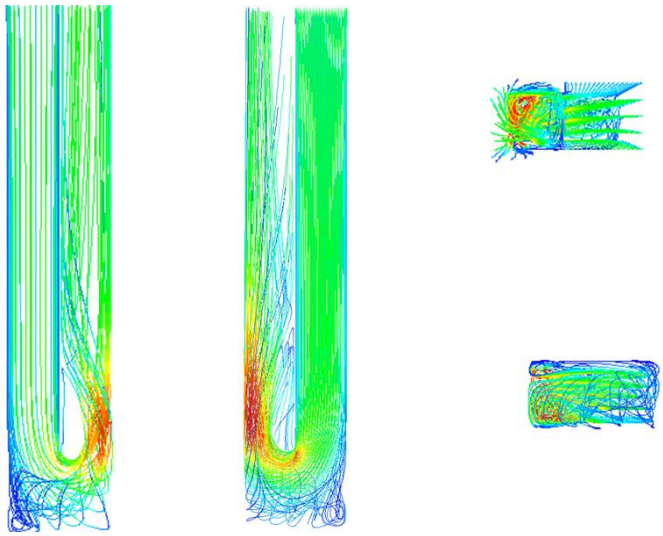
Overview

The flocculation tank simulation team works on building a stable and reliable numerical model to simulate the flow inside the hydraulic flocculation tank, and providing well-studied guidelines for design, construction and operation of the flocculation tank.

Gravity powered hydraulic flocculators are used by AguaClara small-scale water treatment plants due to their low cost, inherent simplicity and robust operation. However, their inflexibility of energy input into the water relative to mechanical flocculators requires well studied design based on the understanding of the flow field and relevant performance parameters.

An appropriate CFD simulation can provide detailed numerical solutions for all the variables in the flow field, and by varying parameters such as tank geometry and flow conditions, we could obtain predictions of each of the flow variables and thus optimize the design towards lower cost and better performance.

Currently, our effort is focused on depicting an accurate energy dissipation map inside the flocculator, describing the size and the shape of the region where most of the energy is dissipated and the formation and collisions of flocs happen, thus providing basis for more efficient and economical design utilizing geometries that dissipate energy as uniformly as possible.



Pathlines of the fluid in a single baffle flocculater, viewed from left, right, top and bottom

Research

Performance parameters analysis in 2D
Characterized the performance of flocculation tanks of different geometry (different ratios of flocculation tank height/baffle spacing), using parameters derived from a semi-empirical flocculation mode, calculated from numerical values of each nodes in the CFD solution.

- **Parameter formulation**: derivation of performance parameters.(word file)
- **Preliminary simulation experiments**: exploratory experiments with encouraging observations and findings about set-up of boundary conditions.
- **Simulation experiments**: characterize performance parameters as a function of flocculation tank height/baffle spacing ratio.
- **Sensitivity analysis**: further investigate the sensitivity of the results to different boundary condition settings at water-air interface and to different convergence(residual) level.

Comparison of 2D model and 3D model with periodic boundary condition using k-epsilon realizable solver
Part of the preliminary effort to extend the 2D models into 3D settings, illustratively demonstrating the underlying simplification -no variation in z-direction - of simulating 3D flows with a 2D model

2D simulations with Reynolds Stress turbulence model
preliminary simulations in 2D settings

3D simulations with k-epsilon realizable turbulence model
Investigating 3D effects with finer mesh and thus better convergence level

Fall 2008

Preliminary 3D CFD simulations of a Hydraulic Flocculation Tank with a Single Baffle

- Fact-finding tests to determine the scope and approach of further research;
- Investigate 3D model's difficulty in converging
- Compare the assumption and validity of 2D and 3D models

Additional Resources

FLUENT quick start