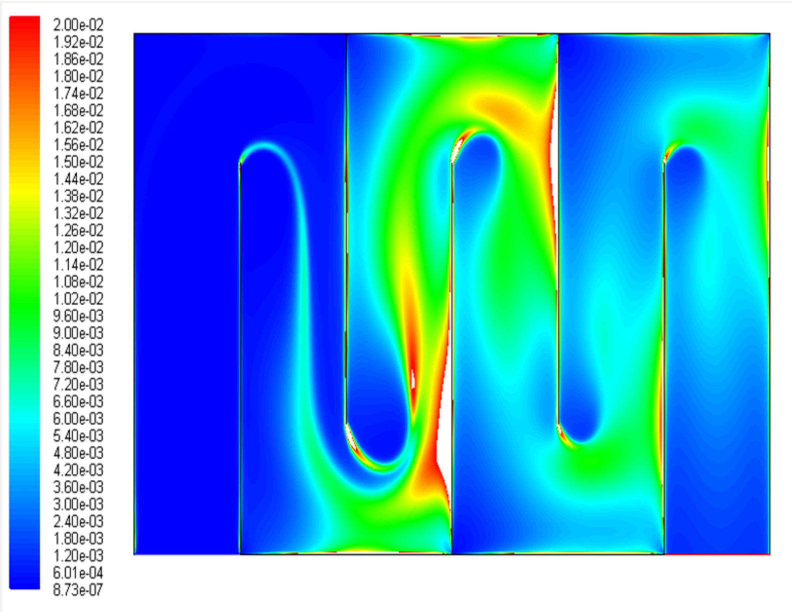
CFD Simulation

Overview

Unknown macro: {float}



Contours of turbulence energy dissipation rate (m^2/s^3)

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve the governing equations of fluid flow problems and obtain a detailed numerical description of the complete flow field of interest.

AguaClara small-scale water treatment plants are composed of gravity powered unit processes. The relative inflexibility of energy input into the water in gravity powered systems requires a better studied design, as opposed to conventional electric-powered, mechanically driven treatment plants. However, few guidelines are available for designing gravity powered systems. The objective of the CFD simulation team is to build a stable and reliable CFD model (using commercial CFD package FLUENT) to simulate and analyze the flow in different parts of AguaClara treatment plants and thus to provide guidelines for the geometry and operation towards better performance and lower costs.

So far, the results of CFD simulations have already facilitated our understanding of hydraulic flocculation tanks and led to some conclusions that can be incorporated in our design.

Stay up-to-date on this project by checking the goals and meeting minutes.

Research

Flocculation Tank Simulation: Spring 2009 research topics on performance parameter analysis and 3D simulation

2D Flocculation Tank Simulation: Fall 2009 research on flocculation tank modeling and performance analysis

Spring 2010 CFD Research: Spring 2010 research to validate Fluent Simulations of the flocculator with experimental flocculator measurements and measurements of headloss from the Agalteca flocculator.

Implications for Design

Flocculator height/width ratio: A clear trend of flocculation performance was observed with various flocculator height/channel width ratios

Flocculator clearance height: There is no need to make the clearance height larger than flocculation channel width, making it possible to handle higher flow rates with the same channel width

Additional Information

Building Meshes in ANSYS Fluent with Workbench

FLUENT tips