

Spring 2009 Mid-Semester Contributions

During the Spring 2009 semester, I explored several different research topics of CFD team. Specifically, the discrepancy between 2D model and 3D model with periodic boundary conditions was resolved with consistent results with the initial hypothesis; I validated the 2D results by comparing with Reynolds stress model and ran 3D simulations with finer mesh in the third dimension; some efforts were put into running simulations on SGI server as the magnitude of our problems scaled up significantly in 3D settings. To address more imminent issues in design, I continued performance parameter analysis which was started in Fall 2008. I checked and corrected the FLUENT set-up, and modified the UDF script and carried out preliminary simulations, simulation experiments and sensitivity analysis. The results were encouraging and consistent with our hypothesis, and implied further interesting issues worth probing into.

In addition, I have been continuously reorganizing the old data files to make it easier to navigate for the next team. And started some documentations that may help with training of new members.

The links to relevant pages can be found from the team main page under Research/Spring 2009.

Fall 2008 Contributions

During the second half of Fall 2008 semester I changed the project from modeling sedimentation tank to 3D simulation of the flocculation tank. A series of preliminary simulations were carried out in order to have an idea of the scope of the problem and how to approach it. The preliminary simulations could be categorized into two topics: investigation of convergence in 3D systems, and comparing 3D results with 2D results to understand the distinction of the key underlying assumptions behind these models and the validity in terms of closeness to the real flow.

Based on the results and reflections from preliminary simulations, I came up with a clear road map about how to do this project to best contribution to AguaClara.

During the first half of Fall 2008 semester I have been working on the CFD team. I'm specifically working on simulating the flow field in the sedimentation tank to optimize the design. I spent the first week understanding the problem I'm going to solve ([Problem Specification](#)) and doing trade-off thinking on whether to adopt computational or experimental approach. I also learned to [document](#) my work with wiki. During the next 5 weeks, I completed a "quick-and-dirty" tutorial of CFD and FLUENT. I learned about the underlying fluid mechanics and numerical methods of CFD techniques, and practiced using Gambit and Fluent with basic simulation problems. And now I'm ready to jump in and create a first mesh. I will start from simulating of 2D cross-section, and will work towards better approximation by iterations of trials.

So far, I can come up with more detailed semester [goals](#) which are both challenging and feasible, and make more sense.

By reasonably reducing the 3D flow field to 2D and further decomposing the 2D cross-section, I now have a starting point to [create the first geometry](#) and the first mesh.

Content created by Anonymous

There are no pages at the moment.
