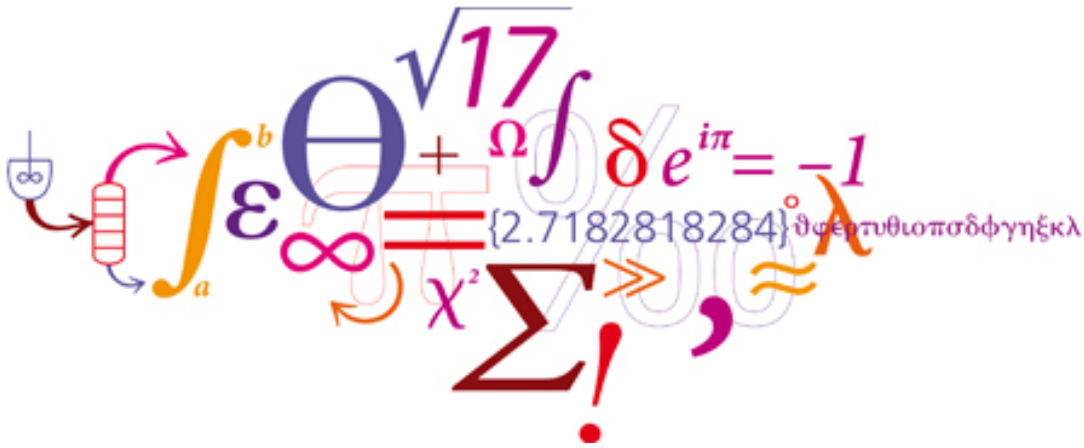
Technical University of Denmark

CFD investigation of a novel self-inducing impeller for gas-liquid dispersion

Master thesis

Tim Bylow Hybschmann - s113172

Jan 30st, 2017



Supervisors

Ulrich Krhüne

Ines Pereira Rosinha Grundtvig

Andreas Spille-Kohof

Tore Svendsen

DTU Chemical Engineering

Department of Chemical and Biochemical Engineering

Contents

6 Conclusion

The topic of this thesis is the investigation of a bioreactor from Bio-Aqua utilizing a self-inducing impeller. This has been performed through CFD simulations. The objective of the work is the investigation of velocity profiles and air distributions inside the bioreactor.
The velocity profiles was first evaluated by doing a single phase simulation on the Bio-Aqua system. It was necessary to increase the timescale in the tank in order to get convergence. The converged solutions showed an upper and a lower recirculation loop which is characterized by a radial-flow impeller. The power consumption was shown to have a linear relation with mass flux in/out.
Multiphase simulations with an interface between the impeller and the reactor domain was performed in order to identify the velocity profiles and air distributions. However, all the simulations in this case crashed.
Furthermore none of the multiphase simulations in this thesis converged regarding RMS.
By removing the interface it was possible to get the multiphase simulations running. However, only the results with a air volume fraction of 2% at the inlet showed air getting sparged into the tank. The air distributions obtained showed air in the pipe, in the impeller and in the reactor above the impeller. The simulation also show that air accumulates behind the impeller blades. For the simulations with a air volume fraction of 100% at the inlet the results showed accumulation of air in the impeller without any of it sparging into the reactor.
It was possible to simulate a self-inducing impeller by using the geometry and setup of Hou-Sheng H. et al and get quantitatively similar results.
It has been observed that the water recirculation in the impeller is a key factor in order to simulate a self-inducing impeller. This could be due to the extra momentum provided and breakup of gas cavities.
All in all the CFD simulations could not predict the air distribution in the bioreactor from Bio-Aqua.

In future work, the obtained mass flux from case study 6 can be used to estimate the volume fraction of air in the outlet of the impeller. This way the pipe domain can be removed and an inlet with a normal speed boundary condition can be implemented in the impeller. The value estimated is then based on a measurement of air inlet from Bio-Aqua and the case study 6.
Mesh studies could be carried out in order to see how mesh dependent the solutions of the different case studies are.
By refining the mesh and lowering the timescale a lot, the volume of fluid method could be used to try and simulate the sparging of the bubbles from the impeller. This could also estimate the bubble diameter. However, this would need a lot of computational resources.