Modeling and numerical simulation of a bioreactor

- Master thesis, research or engineer
- MSCI track
- Supervision: G.-H. Cottet, https://ljk.imag.fr/membres/Georges-Henri.Cottet/
- georges-henri.cottet@univ-grenoble-alpes.fr
- LJK, team EDP, Bâtiment IMAG

A bioreactor (or clarifier) works by sedimentation of the flocs formed by the biomass in a biological reactor, the upper part purifying the water by clarification while the sludge concentrates at the bottom. An interface is formed below which the sludge is compressed.

The models presenting the best compromise between complexity and fidelity of the phenomena are 1-D models based on flow theory. In these models a semi-empirical expression of the sedimentation rate must be defined for each zone, thus diminishing the relevance of the model to represent the physical phenomena. This way of solving the problem leads to different types of equations depending on the area under consideration.

This internship aims to develop a model and a numerical simulation method allowing to determine directly the rate of sedimentation without using empirical expression. The formulation of the model will allow to take into account the dynamical aspects of the system through the equation of momentum, and interaction terms between phases included in these equations (e.g. drag terms) will capture the effects of the concentration on the speed of the phases. There is therefore no arbitrary division of the clarifier in different zones with ad hoc empirical modeling for each zone, the equations being valid over the entire water column.

A first objective of the project will be to propose adapted closure laws in the equations to account for the interactions between phases. A second objective will be to carry out numerical simulations based on these models and on numerical methods developed at LJK, and to compare the results with experimental results.

From a mathematical and numerical point of view, the problem consists of coupling equations describing the fluids (Navier-Stokes equations) with transport equations governing the different phases present in the fluid and the terms of fragmentation-coagulation modeling the flocculation.

The numerical implementation of the problem will be done within the Hysop software library developed at LJK and integrating finite volume, spectral and Semi-Lagrangian solvers for the equations of fluid mechanics.
This internship is part of a joint project with the GIPSA laboratory which combines modeling, physical experimentation and numerical simulation.