3D CFD-ANALYSIS OF SYSTEMS WITH FLUID FLOW

Goals of a 3D CFD-Analysis

A computational fluid dynamic (CFD) analysis of a water-flow through system aims at optimizing the structure of the investigated system in terms of optimal flow conditions. Thus, over and under sizing of the system are prevented. For this purpose, flow conditions have to be represented as realistic as possible (in the model).

the steady-state as well as in the unsteady state. Target figures for improvement are for example:

- Flow velocity
- Retention time
- Pressure conditions
- Water energy state
- Turbulence
- Rotation speed
- Flow capacity
- Mixing conditions

Based upon characteristic target figures for improvement, the optimal structure of the system is determined by means of an iterative approach. On the basis of flexible numerical models, an optimal solution to flow optimization problems may be developed very efficiently. A 3D-CFD-Analysis helps to optimize water-flow through structures in

Scrapers, agitators, pumps, nozzles, air introduction, valves, gas supply, ventilation, suspended sediments, dissolved particles, solid objects, etc. may be considered in the context of single and multiphase flow. The flow optimization increases the reserve capacity of the system and maximizes the benefit from the investment.

Areas where CFD comes into effect?

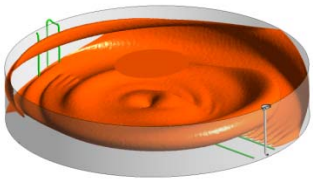
CFD analyses may be utilized in many different disciplines and are multiply deployable. The following application examples show a selection of our credentials in the disciplines

drinking water transmission and distribution, urban drainage, waste water and river engineering.

Water quality & Retention time

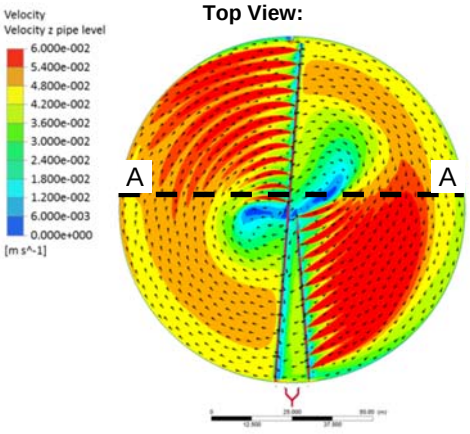
The flow conditions in a water tank are strongly influenced by the system design. The following aspects have to be considered:

1) Prevention of stagnation water in tanks
2) Overall minimization of the retention time of the water in the tank

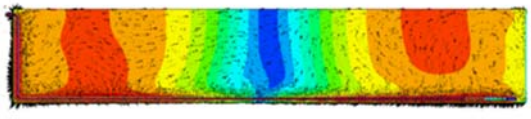


Retention Time
< 12 hours

Top View:

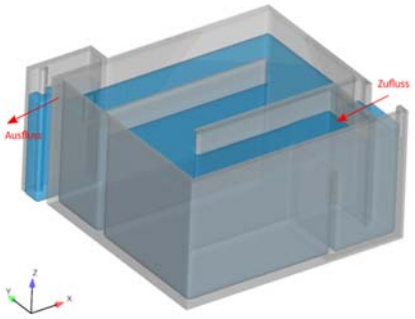


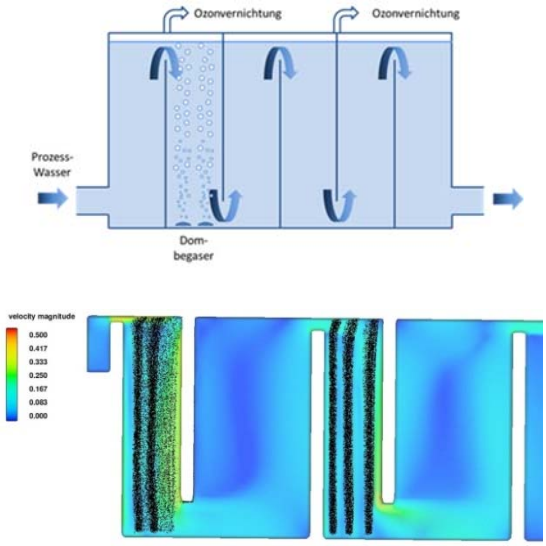
Cut from A - A:



Design of ozone reactors : injection and contact chamber

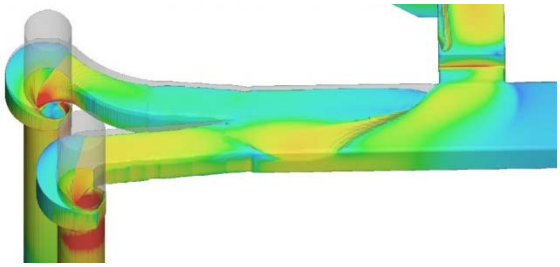
CFD-Analyses are applied for the optimization of ozone reactors. Retention time, contact time, dead zones and mixing in the different chambers may be optimized by means of CFD-modelling.

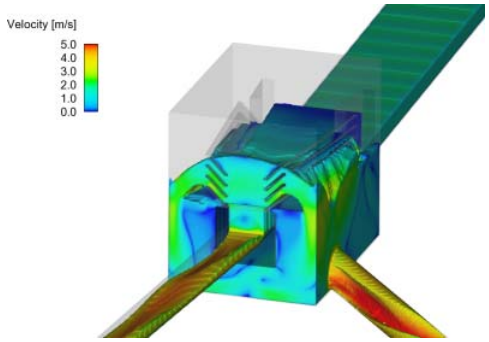


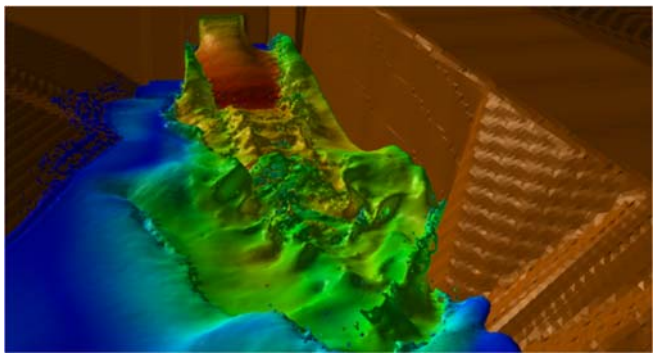


Complex geometries of hydraulic structures:

CFD-Analyses may be applied for the most complex hydraulic geometry structures.







How can IBG help?

IBG has a wide experience in the field of 3D simulation of water systems. The engineering consortium is composed of drinking water, urban drainage, waste water and river engineering specialists. Thanks to the exchange of knowledge with ETH Zurich and many other universities, our methods are continuously adapted to reflect the newest state of research. Thus, IBG is able to investigate complex 3D flow conditions of any hydraulic structure. In consideration of the hydraulic demands, we develop the best possible solution with an optimal cost-benefit-ratio with regard to investment costs and operational costs.