

# Numerical simulation of two-phase flows in bottom outlets

Computational fluid dynamics (CFD) is gaining in importance in the field of water and wastewater engineering and increasing computing power allows for more and more complex models. Despite the growing popularity of 3D CFD-models among practitioners, sophisticated model calibration and validation with experimental data remain scarce (Fig. 1).

This project aims at simulating the two-phase flow in bottom outlets numerically. Bottom outlets are a key safety element of high-head dams. Due to high flow velocities of up to 60 m/s, the flow is highly turbulent and considerable air entrainment occurs. This presents an interesting challenge for numerical simulations. Data for calibration and validation are provided by VAW from a large-scale hydraulic model.

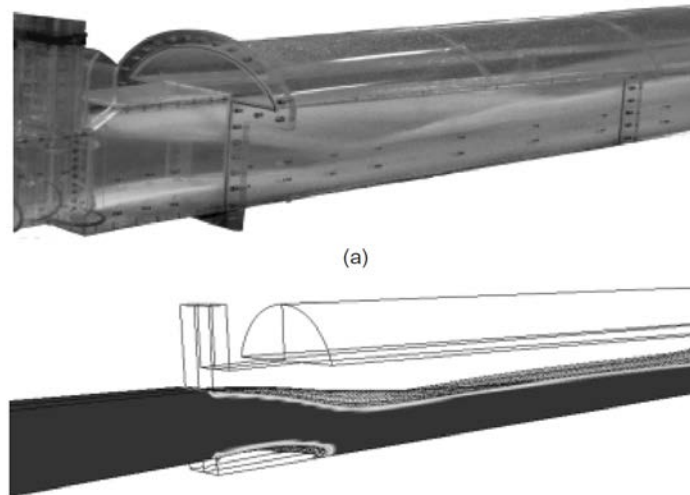


Fig. 1: Comparison of physical and numerical bottom outlet model (Najafi & Zarrati 2010)

The project consists of two parts: Starting from an existing Flow-3D model, the student will thoroughly calibrate and validate the model with experimental data. The model complexity will be increased step-wise until the experimental data can be reproduced satisfactorily. This should lead to the development of a “best practice” standard for the simulation of bottom outlet flows. The second part is more applied: In a parametric study, the student will investigate the effect of different parameters (e.g. tunnel geometry) on the bottom outlet performance. Based on the results from the second part and the physical model experiments, new design criteria for bottom outlets shall be proposed.

**Contact:**

Benjamin Hohermuth  
HIA B13  
044/632 55 41  
[hohermuth@vaw.baug.ethz.ch](mailto:hohermuth@vaw.baug.ethz.ch)

**Additional notes:**

Single MSc-Thesis;  
Use of Flow-3D and AutoCAD  
Communication in German or English