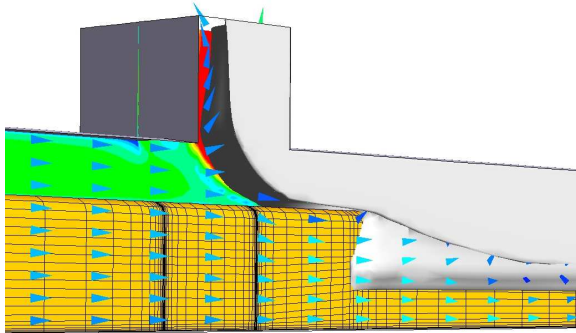
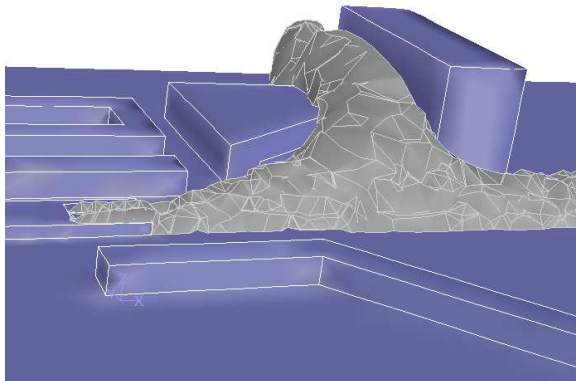


Smoke dispersion in an underground station



Natural smoke exhaust due to stack effect



Pollution concentration in the vicinity of a portal

CFD is a numerical method to calculate flows in systems of arbitrary geometry (Computational Fluid Dynamics). The entire flow field is divided into small elements. The velocities and scalar quantities such as temperature or smoke concentration are calculated in each element.

Our services

- Computational fluid dynamics for arbitrary systems in one-, two- or three dimensional geometry (1D, 2D, 3D)
- Estimation of the dispersion of pollutants from tunnel portals or chimneys into free environment
- Calculation of airflows and smoke dispersion in complex underground installations
- Estimation of air exchange rates and expected temperatures in large buildings and spaces
- Visualisation of flow fields
- Development of CFD software for special purposes
- Calculation of smoke dispersion in tunnel systems and underground rail stations
- Calculation of train- or ventilation-induced airflows, which are required as boundary conditions for calculations

Your benefits

- Through the application of CFD, you may be able to reduce or eliminate your need for physical model tests.
- By visualising the results in a multi-dimensional manner, you are able to grasp a deeper understanding of the main parameters and the functioning of your facility.
- You obtain access to our specific know-how and leading-edge numerical calculations for the simulation of fluid flow.