



Flow Simulation

Gaining Insights through CFD Analysis

Flow analysis using CFD (Computational Fluid Dynamics/ Numerical Flow Simulation) is crucial for evaluating the functionality of fluid-carrying or flow-exposed components.

Our range of service includes a variety of calculation disciplines, from analyses of flow-exposed, flow-through, or rotating components, thermal process calculations and

Our services include:

- Steady-state flow analysis
- Transient flow analysis
- Thermal flow analysis
- Rotating machinery
- Fluid-structure interaction
- Sensitivity and robustness analysis
- Optimization

time-resolved, transient simulations, to the calculation of loads and temperatures for fluid-structure interactions with coupled FEM calculations.

For flow simulation, we use the programs ANSYS CFX and ANSYS Fluent.

Training and Workshops

In addition to conducting flow simulations, we also offer training and workshops. With these workshops, specifically tailored to your needs and projects, we support you in your daily task – online or in-person at your location.



simulations driven by engineering excellence

We accelerate our customers' innovations through virtual simulations and precise calculations of physical and technical requirements. We transform our results into practical solutions that enable our customers to achieve excellence in engineering technology.



Dr. Frank Brehmer,
Managing Director ITB

Our services



**Component
Strength Analysis**



CAD Construction



**Component
Verification**



**Electrothermal
Simulation**



**Flow
Simulation**



**Light
Simulation**

**Are you planning your next project
and want more security from our
expertise? Then get in touch with us:**

**Phone: +49 (0) 231 / 94 53 65-0
Email: info@itb-fem.de**