



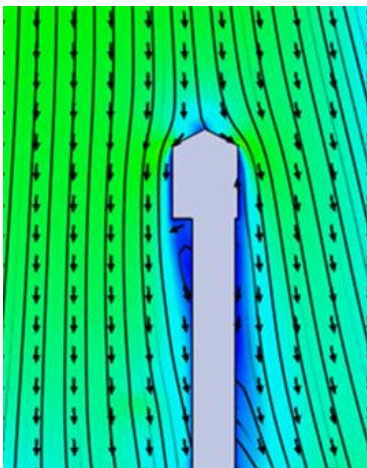
- ✓ Computational fluid dynamics (CFD) simulations of the airflow inside the working chamber

Why CFD simulation?

The CFD simulation tool can be used to achieve significant engineering improvement up to excellence in asset- and process equipment efficiency and reliability. Resulting advancements can directly impact product and process performance. Visualization and simulation are state of the art key-tools to show and predict performance and future potential.

The flow-field visualization of our CFD study allows for a higher spatial and temporal resolution, compared to many measurement techniques.

The study is based on the numerical solution of the steady-state RANS equations for incompressible flows (low Mach-number flows). The required turbulence closure is obtained by employing a two-layer k-epsilon based model.



Qualitative examination of air flow around component

Your Challenges

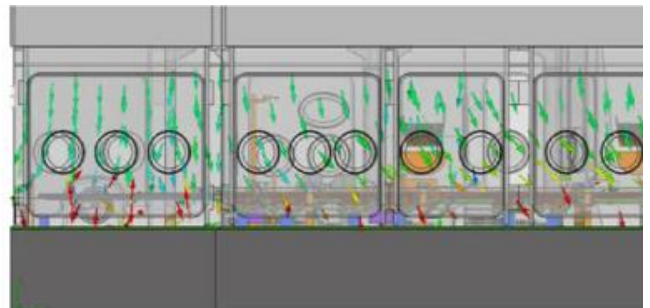
- Design and operation of equipment and processes (i.e. load arrangement and load items) need to be supported by scientific arguments with focus on the mechanistic understanding

Our 5-step Solution

- Geometrical preparation
- Setup boundary-conditions
- Mesh generation
- Numerical solution
- Result processing, decision support and recommendations for action

Your Advantages

- Visualization of air flow characteristics during production mode
- Results support and justify the isolator design, testing and operation
- Cost savings due to minimization of the error probability during the planning phase, followed by early corrective actions



Qualitative investigation of working chamber with regard to unidirectional air flow