

## Characterisation of the AIDA aerosol chamber by means of commercial computational fluid dynamics (CFD) software

S. Benz, H. Bunz, J. Meinen, O. Möhler and T. Leisner

Institute of Meteorology and Climate Research, Forschungszentrum Karlsruhe, Hermann-von-Helmholtz-Platz, 76344 Eggenstein-Leopoldshafen, Germany

Keywords: CFD, aerosol chamber

The aerosol and cloud chamber AIDA of Forschungszentrum Karlsruhe has shown in the past decade to be a powerful tool simulating atmospheric aerosol and cloud processes (e.g. Möhler et al., 2005, Saathoff et al., 2008). We used a computational fluid dynamics (CFD) code to simulate the coupled fields of velocity, temperature, water vapour and particle number concentration inside the chamber during an expansion experiment. The aim of the presented study is to check the degree of homogeneity by numerical means. We present comparisons of numerical simulations with available experimental data sets.

AIDA consists in principle of an evacuable aluminium vessel (volume 84 m<sup>3</sup>, diameter 4m) which is surrounded by a coolable housing. The temperature can be set to any temperature down to -90°C. By controlled pumping the pressure inside the vessel can be reduced inducing a further cooling. This additional cooling – which is adiabatic only in the first stage – is limited by an increasing flux of heat from the chamber wall. The wall rests at almost constant temperature due to its high heat capacity. The gas temperature inside the vessel is monitored by arrays of vertically and horizontally arranged temperature sensors. The inhomogeneity throughout the entire vessel is estimated to +/- 0.3 K. The isothermal wall is internally covered by a thin ice layer which evaporates during the expansion process. This source of water vapour reduces the decrease of the partial pressure of water vapour during reduction of pressure inside the vessel. Thereby the maximum achievable super-saturation is increased. Water vapour is measured in situ using a tuneable diode laser absorption spectroscopy (TDLAS). The sensitivity of the measurement is enhanced by a White-type multiple reflection cell. The measurement delivers a mean value at medium height of the aerosol vessel. Currently no experimental device is available at AIDA to determine the nature of the turbulent flow field inside the vessel.

For our simulations we used the commercial CFD code ANSYS CFX 11.0. The geometry of the AIDA aerosol vessel was highly simplified neglecting details as the above mentioned reflection cell in order to get a rotationally symmetric

geometry. Thereby the computational costs could be reduced extremely as solely a segment of 2° had to be modeled. The mixing fan was implemented by an artificial source of momentum. A hexahedral mesh with 24,300 nodes was generated by means of the software ANSYS ICEM CFD. In the boundary layers the mesh was refined to resolve high gradients near the walls.

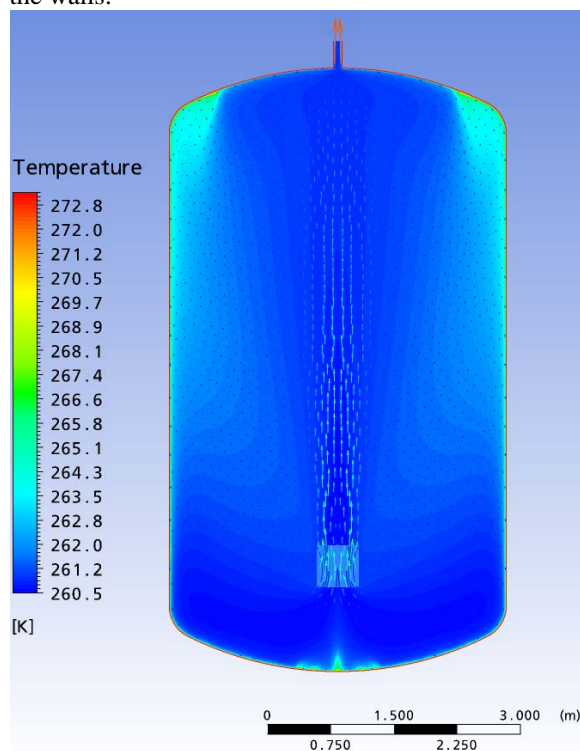


Figure 1. Cross section of AIDA vessel with contour plot of temperature after the pressure was reduced from 1000 hPa to 800 hPa within 250s. The vectors illustrate the flow field. The pressure inside the chamber is reduced via a tube which is mounted at the top of the chamber. The air speed directly above the mixing fan amounts to 3 m/s. The mixing fan is enclosed by a baffle (brightly shaded area at the lower part of the chamber).

Möhler, O. et al. (2005), *J. Geophys. Res.*, 110, 3456-3467.

Saathoff, H. et al. (2008), *Atmos. Chem. Phys. Discuss.*, 8, 15595-15664