Numerical Simulation of Mixed Convection in an Aircraft Cabin Segment

Dr. Oliver M. Webel, oliver.webel@dlr.de **Dr. Markus Rütten**, markus.ruetten@dlr.de **Prof. Dr. Claus Wagner**, claus.wagner@dlr.de	German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology, Fluid Systems Department, Bunsenstr. 10, 37073 Göttingen, Germany

Abstract

During the last decades, the thermal loads in modern passenger airplanes have increased. New equipment such as board electronics, flight entertainment and communication systems have a considerable impact on the temperature distribution inside the passenger cabin. In order to study these effects in an early design phase, aircraft manufacturers are interested in reliable Computational Fluid Dynamics (CFD) methods for predicting the velocity and temperature fields inside the aircraft cabin. In order to improve these methods it is also important to better understand the occurring fundamental flow physics and to validate the CFD-predictions comparing them to cabin flow measurements.

At the conference unsteady Reynolds-Averaged-Navier-Stokes (URANS) simulations of mixed convection in a segment of the AIRBUS A-320 cabin reflecting thirteen seat rows and 78 passenger dummies will be presented. One thirteenth of this segment is presented in Fig. 1. as a volumetric mesh of 20 million cells. The complete mesh will therefore consist of approximately 260 million cells (prisms and tetrahedrons).

The cabin segment model has an approximate width of 3.78m, a length of 1.06 m, and a height of 1.25m. It is equipped with three seats at both sides separated by an aisle and six passenger dummies with fixed surface temperatures (representing thermal loads). Air is supplied by four inlets, two being located at the ceiling and two in a lateral position (see Fig.2). After circulation the air leaves through two air outlets located at the cabin floor.

For computations the buoyantFoam solver family of OpenFOAM 1.6 was employed including the implementation of the P1 radiation model into the transient solver.

The duration of the transient simulations are chosen as 300s (averaged temperature and velocity fields are shown) and initialised by a converged steady-state solution obtained with the steady-state buoyantFoam solver.

Key words: Mixed Convection, Aircraft Cabin Aerodynamics, OpenFOAM, buoyantFOAM

References

[1] J. Bosbach, J.Pennecot, C.Wagner, M.Raffel, T.Lerche and S.Repp: *Experimental and numerical simulations of turbulent ventilation in aircraft cabins*, Energy, **31**:694-705

[2] W.Zhang and Q. Chen: *Large-Eddy Simulation of natural and mixed convection airflow indoors with two simple filtered dynamics subgrid scale models*, Numerical Heat Transfer, Part A: Applications, 37(5), 447-463

[3] S.Yin and T.Zhang: *A new under-aisle displacement air distribution system for wide-body aircraft cabins,* Eleventh International IBPSA Conference, Glasgowl, July 27-30. 2009

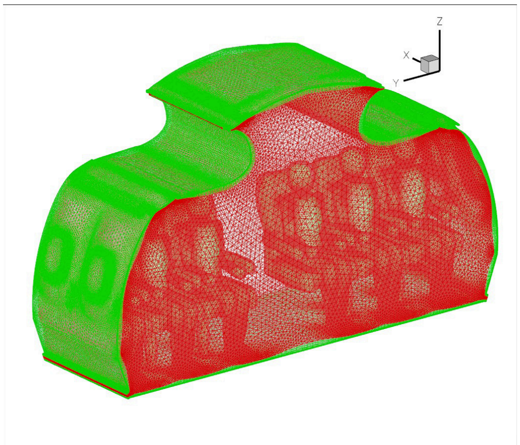

Fig 1.: Volume Mesh of a segment of an A-320 cabin model

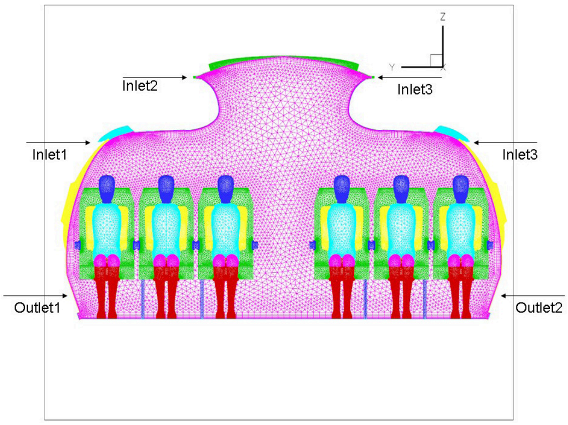

Fig. 2: SurfaceMesh of a A-320 cabin segment model