NUMERICAL SIMULATION OF A MIXED VENTILATION SETUP IN AN AIRCRAFT CABIN

O. Webel, M. Rütten, C. Wagner
Deutsches Zentrum für Luft- und Raumfahrt (DLR)
Institut für Aerodynamik und Strömungstechnik
Bunsenstrasse 10, 37073 Göttingen, Germany

Abstract

Reynolds Averaged Navier-Stokes Simulations (RANS) are conducted for an Airbus A-320 geometry consisting of thirteen seat rows and seventy-eight passenger models with extruded air inlets and outlets. Temperature and Velocity distributions are presented in order to study the comfort feeling of passengers and local convective flow characteristics. Additionally, radiation effects are investigated in a cabin segment with unsteady RANS (URANS) of up to 42 seconds time. The obtained results are compared to those of URANS without radiation modelling.

1. GEOMETRY AND MESH

The considered Airbus A-320 cabin consists of thirteen seat rows and seventy-eight passenger models in total (Fig. 1). One segment of the cabin model has a width of 3.78m, a length of 1.06 m, and a height of 2.25 m. The air inlets and outlets are extruded in order to stabilize the flow in particular when it enters the computational domain. The geometry is divided into thirty-six STL patches (e.g. heads, bodies, ceiling, floor etc.) to be able to employ an improved version of the snappyHexMesh grid generator.

The final mesh (Fig 1, Fig 2) consists of approximately 31 million cells with a base cell size of 64 mm and several refinement regions (e.g. air inlets). 88.4 per cent of all cells are hex-cells, 1.6 per cent prism cells and 9.9 per cent polyhedral cells. We use five surface layers on the passengers and on the side walls. Mesh non-orthogonality and skewness are in the valid range. The mesh was generated in 320 min on eight processors of 8GB memory.

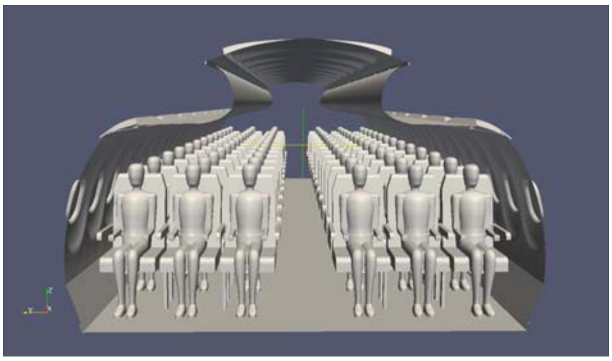

Fig. 1: CAD-geometry of the A-320 cabin

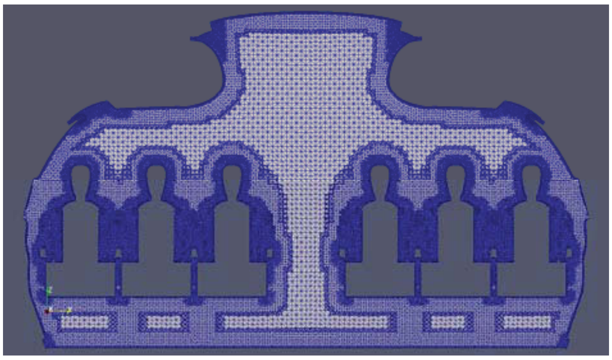

Fig. 2: Hybrid unstructured/structured mesh in a centre cross section of the cabin

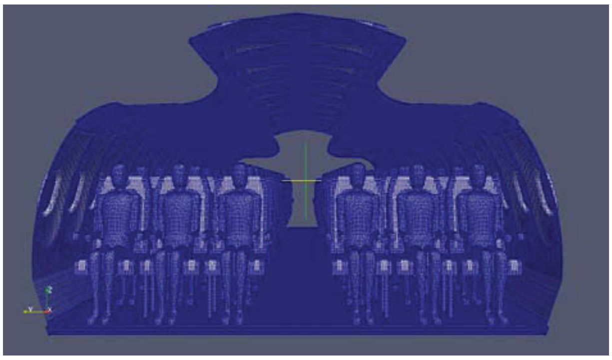

Fig. 3: Surface mesh of the domain

2. SOLVER SETTINGS AND BOUNDARY CONDITIONS

OpenFOAM 1.6 is employed for the presented computations using the steady-state RANS solver buoyantBoussinesqSimpleFoam and the unsteady URANS solver buoyantBoussinesqPisoFoam. The latter is applied together with and without an available radiation model. Further, the k-ω STT turbulence model is used and a non-participating grey discrete ordinates model with sixty rays to be traced every ten solver iterations for radiation modelling. The emissivity of the surfaces is chosen as ε=0.92.

As boundary conditions a fixed velocity vector is used at the inlets and computed from a given volumetric flow rate 0.75 m^3/s from the A-320 air distribution system and for the outlet a fixed pressure condition of 1 bar is applied. For the passenger models a constant heat flux of 62.12 W/m^2 is prescribed at the models surfaces, except for the arms where 68.55 W/m^2 are employed. The front and back walls are impermeable walls and the side walls, floor and ceiling are adiabatic. The air entering the domain at the air inlets has the temperature T=285K.

3. RESULTS

3.1. Steady-State computations of the complete cabin

It can be seen that in the approximate middle of the domain (seventh seat row – Fig. 4 & Fig. 5), there is an antisymmetric flow and a temperature field resulting from the mixing of essentially two components: the local convective flows driven by buoyancy resulting from the temperature differences between the heated passenger models and the surrounding air and the much colder air inlet jets entering the domain. One can observe, that there is a short-circuit between the left air inlet jet and the air outlets. The passenger models located next to the aisle are cooled considerably near the heads and the side of the bodies which are oriented towards the aisle. The local convective flow on the passenger model sitting on the left is almost annihilated by the intensive left air inlet jet. This behaviour is not observed on the right side of the aisle.

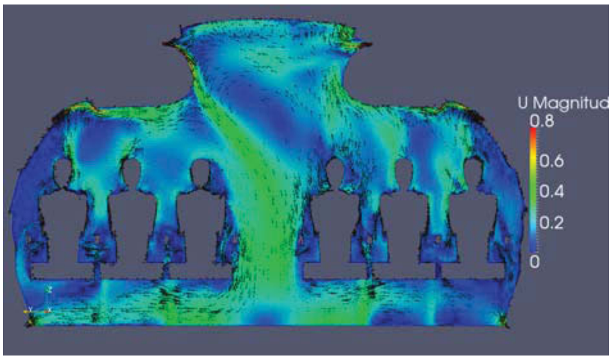

Fig. 4: Velocity magnitude in a plane located in the seventh seat row (middle of the domain)

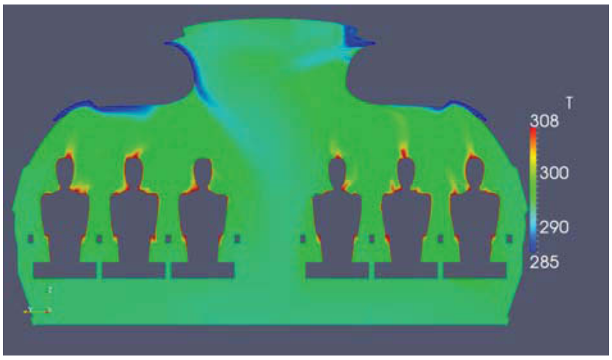

Fig. 5: Temperature distribution in a plane located in the seventh seat row (middle of the domain)

Fig. 6 illustrates the surface temperatures of the passenger models, which indicates that these are - without radiation modelling - much too high. This is true in particular for the head and breast sections of the models, however not for models sitting close to the aisle as well as the feet sections which are cooled considerably.

The mean temperature in the domain is 293.55 K and the maximum temperature is 329.45 K which is located on the passenger models and therefore not acceptable.

Fig. 7 reflects the temperature field resulting from the local convective flows driven by the temperature differences between the heated passenger models and the surrounding air using iso-surfaces of T=299K. In the areas where there is a strong dominance of the air inlet jets (close to the aisle), one can observe that the local convective flows are annihilated. Further on, in the first sections of the cabin, due to a strong symmetry of the flow and related to the close proximity of the front wall, the local convective flows are reduced as well.

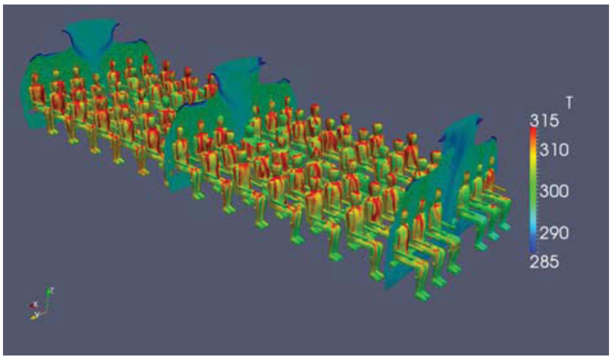

Fig. 6: Surface Temperatures of passenger models in the complete cabin

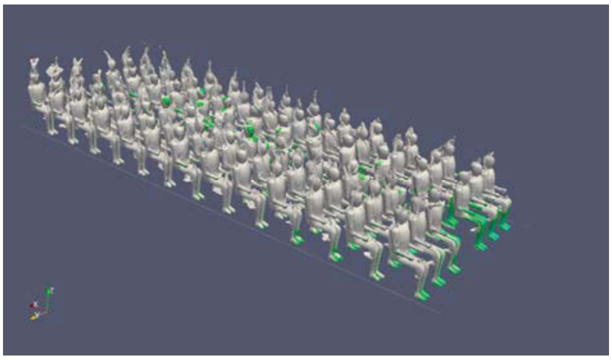

Fig. 7: Iso-surfaces at T=299K of the complete cabin

3.2. Transient computations of one segment – Effect of radiation modelling

In one segment of the aircraft cabin, unsteady RANS computations with and without radiation modelling are conducted in order to illustrate the effect of radiation on the distribution of the temperature field. For the unsteady simulations, heat radiation revealed to have a strong effect on the temperature distribution. Due to the radiated heat the surface temperatures of the passenger models decrease. Therefore the temperature gradient is smaller and the magnitude of local convective flows decreases as well. In Fig. 8 and Fig. 9, velocity and temperature distributions are shown for the transient simulations without radiation. Comparing the temperature distributions in Fig. 9 and Fig. 11, one can observe that the reduced surface temperatures resulting from the heat radiation have a strong effect on the flow field (Fig. 8 & Fig. 10). In Fig. 8, there are local zones of strong convection above the heads of the passenger models while these are significantly reduced in Fig. 10. In both cases however, anti-symmetric flow fields is obtained.

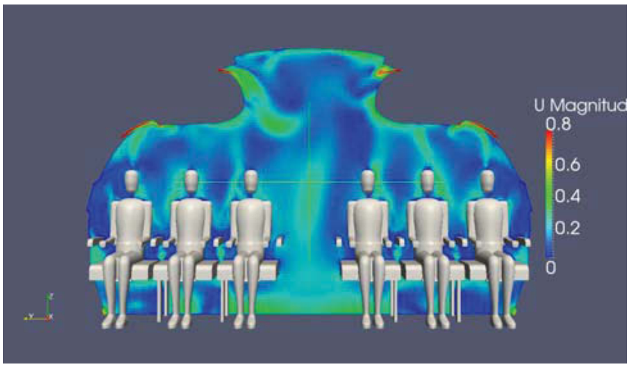

Fig. 8: Instantaneous velocity magnitude distribution obtained in one segment, without radiation modelling at t=42s

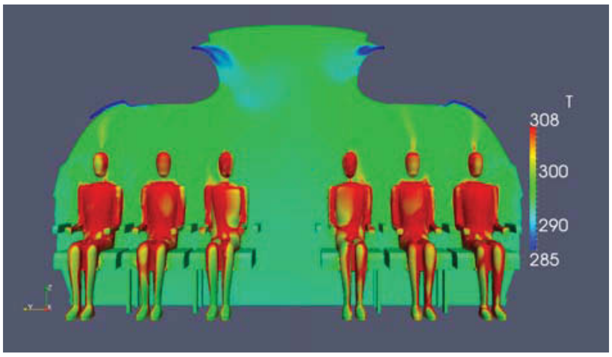

Fig. 9: Instantaneous Temperature distribution obtained in one segment, without radiation modelling at t=42s

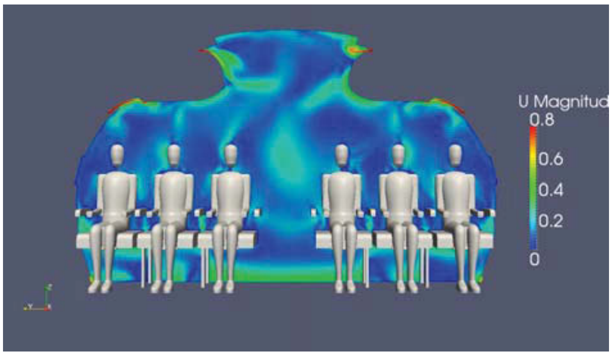

Fig. 10: Instantaneous velocity magnitude distribution obtained in one segment, with radiation modelling at t=42s

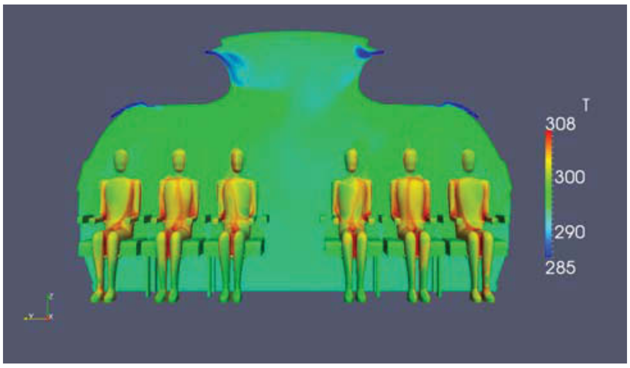

Fig. 11: Instantaneous temperature distribution obtained in one segment with radiation modelling at t=42s

When comparing the surface temperatures of the passenger models directly to each other with and without radiation modelling effects (Fig 12 & Fig 13), one can see that the temperatures with taking radiation effects into account are much more reasonable compared to the computations without radiation modelling. The dominance of the air inlet jets can also be observed since the sides of the bodies located next to the aisle are cooled considerably in both cases. The regions around the legs reflect reduced temperatures due to the proximity to the outlets on the left and right hand side of the domain.

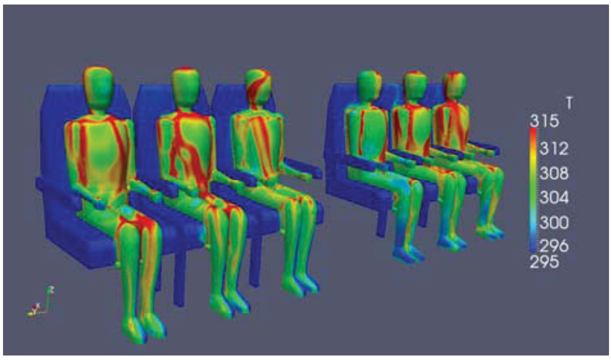

Fig. 12: Surface temperatures distributions on passenger models, without radiation modelling at t=42 s

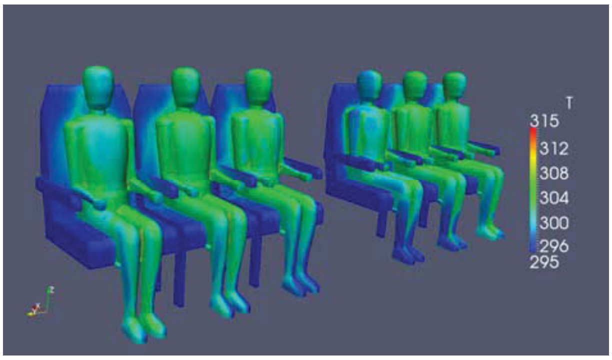

Fig. 13: Surface temperatures on passenger models, with radiation modelling, t=42 s

[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, Glasgow, July 27- 30. 2009