5th International Conference from Scientific Computing to Computational Engineering
5th IC-SCCE
Athens, 4-7 July, 2012

DETAILED SIMULATION STUDY AND SUBJECT TESTING OF INDIVIDUALISED AIRCRAFT CABIN SUITES ENVIRONMENT

Konstantinos Tsikouris[1], Jan Fišer[2], Iris Noeske[3], Michael Trimmel[4]
[1]Icon Technology & Process Consulting Ltd, UK
Berkshire House, Thames Side, Windsor, Berkshire, SL4 1QN, United Kingdom
e-mail: k.tsikouris@iconCFD.com, web page: http://www.iconCFD.com

[2]Brno University of Technology, Czech Republic
[3] Fraunhofer Institute for Building Physics IBP, Germany
[4] Professor, Centre for Public Health, Institute of Environmental Health, Medical University of Vienna, Austria

Keywords: Individualized ventilation, CFD, open source, aircraft cabin, passenger thermal comfort, personal inlets.

Abstract. *The study was carried out as a part of the iSPACE project and its aim was to assist in the scope of fulfilling specific human thermal comfort needs in the frame of an individual aircraft cabin seat.*
A detailed simulation study of individualized ventilation schemes in aircraft cabin seats was performed with the use of CFD methods. The ventilation schemes were selected based on a previously conducted parametric numerical study of personalised ventilation inlets of various sizes and locations in the aircraft suite.
These selected schemes were then applied to business class seats and first class suites, with the boundary conditions based on physical measurements taken during actual subject testing. The study was was performed in an aircraft lining section, comprising of actual human subjects, passive manikins and some empty 'control' seats.

The simulation study was conducted using an open source based CFD process (ICON FOAMpro) and a proprietary software (STAR CCM+). The results were validated against probe measurements from the subject testing and evaluated in order to illustrate the thermal conditions achieved by the individualized ventilation scenarios.

1 IMPLICATIONS

Increasing utilisation of air travel, particularly long distance travel, has made the personal comfort of passengers become increasingly important. Numerical investigation of several individualized ventilation scenarios and comparison with actual subject testing in a mock-up cabin section, has shown that passenger thermal comfort in an aircraft cabin, can be achieved with the use of individual ventilation inlets and personal seat heating, which are manually controlled by the passenger.

2 INTRODUCTION

The personal comfort of aircraft passengers is an important consideration in modern commercial airplanes, with increasing air traffic and long distance travel, where the parameters of the ambient environment (e.g. temperature, pressure, relative humidity etc.) often have to adapt to the external conditions imposed by the constantly changing altitude and climate.

This study summarizes the experimental data collection and the Computational Fluid Dynamics (CFD) modelling work performed for the seventh EU Framework Program iSPACE 234340[4]. The ultimate objective of the program is to investigate the feasibility of introducing personal and individual ventilation devices into aircraft cabins, as well as the design and control of the individual passenger climate by using similar technologies as those used in the automotive industry.

While the Environmental Climate System (ECS) provides overall comfort, the use of personal inlets may prove useful to provide thermal comfort on a scale closer to the individual's needs. These personal ventilation devices can be manipulated by the passenger and different parameters such as adjustable flow-rates may bring satisfactory results for the passenger's personalised nature.

The presented work from the iSPACE project focuses on first class suites and local ventilation. The parametric study briefly introduced here aimed to find the best position for these inlets, in a simplified model of a first class suite, with a computational passive manikin employing Computational Fluid Dynamics (CFD). The detailed simulations investigated more accurately the qualitative effects of individual climate controls and provided a means of validating simulation results against the experimental measurements from an actual full-size populated aircraft cabin at the Fraunhofer Flight Test Facility under realistic in-flight conditions.

The CFD methodology, solution domain, model set-up, solving and post-processing, as well as the experimental testing procedure is briefly illustrated.

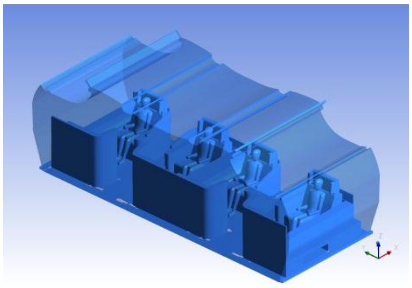
3 THE PARAMETRIC STUDY

3.1 Objective and method of the Parametric Study

The objective of the parametric study was to select step change technologies to be developed for future tests and evaluation. A numerical study of personalised ventilation inlets in an aircraft first class suite was performed using CFD methods. The parametric study was performed on a simplified geometry of a periodic Airbus lining section with 4 rows of seats in the first class compartment as depicted in Figure 1.

Several positions of local personalised inlets as well as seat air suction were tested with different flow rates (according to the possible power and ventilation system demands). The positions are labelled in Figure 2 starting with Case A to Case F respectively. Seat suction is labelled with a red colour showing the position of the air removal through the fabric of the seat base and back-rest.

The effectiveness of each case was evaluated by employing a thermal comfort zones model, age-of-air scalar and draft risk.



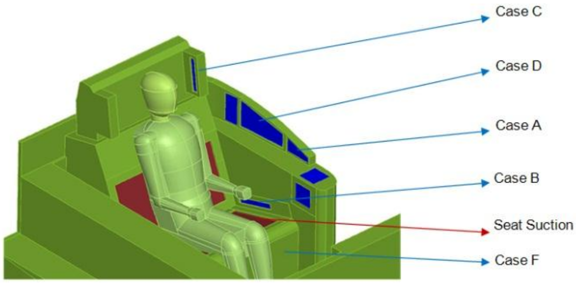


Figure 1. Final volume mesh Figure 2. Locations of the local inlets and tested cases at the seat..

3.2 Choice of software packages

In the study, both in the parametric study as well as the detailed simulation stage, two software packages were tested to evaluate performance between commercial and open source solutions. The CFD solver and meshing tools deployed in ICON FOAMpro [2] were explored and compared with the outcome of the proprietary code STAR-CCM+. ICON FOAMpro is based upon the OpenFOAM® toolbox [3]. Both are based on the finite volume method of discretisation of Navier-Stokes equations. The physical models, turbulence model, solver settings, meshing and solution strategy were selected to predict with reasonable accuracy and speed, the physical phenomena encountered in this type of HVAC comfort problems.

Although some different models were used, for example radiation was computed by the DOM - Discrete Ordinate Method) for the parametric study whereas the P1 model for the detailed simulation study used Icon FOAMpro and the Surface to Surface model was used Star-CCM+. Te results were shown to be comparable in the flow-field, as well as in velocity. There were some differences in temperature profiles but these are of the magnitude of 8-10% and in some cases less than 5% (case C3 of final calculations).

3.3 Conclusions of parametric study and Choice of best individual inlet combination

A total of 12 simulations were conducted with both the proprietary and the open source code. Additionally, the inlet mass flow rates were varied for two of the above cases.

Results of the parametric study showed that a combination of two inlets is needed to overcome the non-uniformity of the flow pattern from the main ventilation system for different suites. The most promising variant is the combination of a side nozzle directed towards the face of the passenger and side displacement nozzles along the armrests. The side nozzle (Case A) brings fresh air to the face of the passenger. However the effect of the jet is overridden by the flow pattern from the main ventilation for the middle seats B and C. This effect is minimized when combining the nozzle with the side displacement inlets along the seat (Case B). A potential problem however could be the risk of discomfort from a draft of fresh air, from the displacement inlets, felt on the arms and hands of the passenger. This could be overcome however by allowing flow-rate control by the passenger, which was investigated in the experimental testing from a mock-up in Fraunhofer Flight Test Facility, and in detailed simulations.

4 DETAILED SIMULATION OF VARIATION OF CASES WITH SELECTED INLET COMBINATION

4.1 Definition of BC to be used in detailed simulation

A preliminary set of boundary conditions was set by Icon based on the parametric study results and previous studies.

The overall air temperature in the cabin was controlled to Tair = 20.0°C // 24.0°C // 28.0°C (293.15 K // 297.15 K // 301.15 K) corresponding to cold, normal and hot operating conditions. Air humidity was introduced as a passive scalar.

Volume flows per cabin frame, concerning the Ceiling & Lateral Inlets, the Personal Air Inlets and the Total Cabin Air Supply, were supplied by Fraunhofer and Airbus. The ratio between Fresh Air and Re-circulated Air was set to 60% / 40 %.

The majority of the seats which were occupied during the tests (Figure 3) were supplied with adjustable personalised inlets: displacement nozzle, ventilation grill inlets and seat heating or special ventilation system products. According to the inlet/outlet humidity, temperature, mass flow or any combination of them could be individually controlled.

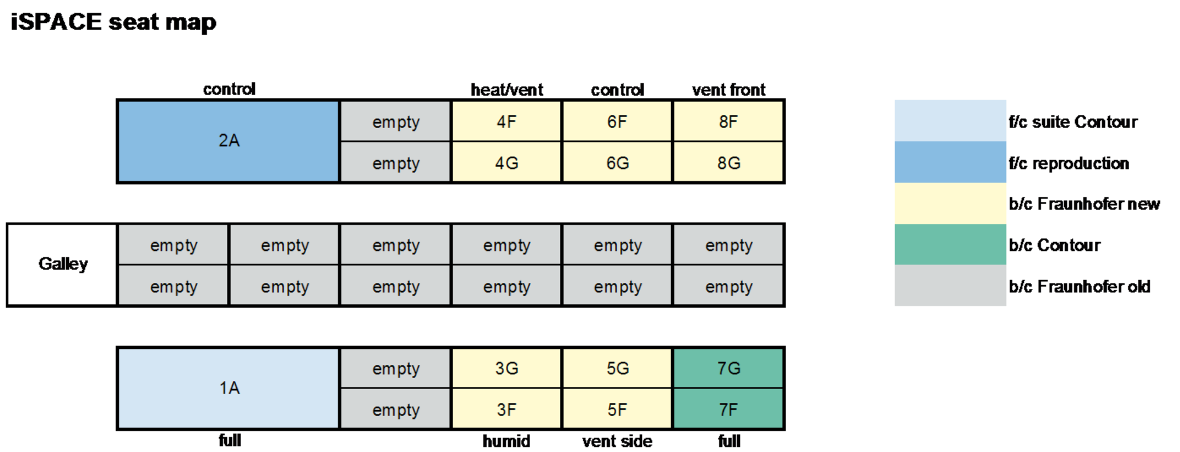


Figure 3 Cabin seating arrangement

Based on the above, velocity at all inlets was calculated and validated/corrected based on additional measurements which were made by BUT at the Fraunhofer Flight Test Facility on the individual inlets which were used to assist finalise their boundary conditions.

4.2 Experimental testing

In the Fraunhofer Test Flight Facility (FTF) cabin a total of 30 Business Class and two First Class suites were integrated(one B/C seat and one F/C suite form Contour) (Figure 4).

Figure 4. Full-size First Class cabin at FTF

Three weeks of testing were carried out, during which subjects occupied part of the seating and manikins were placed on the remaining seats. The subjects participated in an 8h flight under real flight conditions (low pressure, noise, meal brakes, exterior conditions, etc) and had control over their individual ventilation inlets/outlets. Passengers switched positions in a rotation, to achieve a balanced experience by the test subjects. This procedure was supervised by research personnel of Fraunhofer and the Medical University of Vienna who served as flight attendants and were responsible for the collection of the test data.

Each week one subject group of 14 subjects was dedicated to either cold, normal or hot flight conditions. Environmental variables were monitored at a large number of probes (three for each passenger plus multiple probes for the lining, outlets etc – Figure 5)

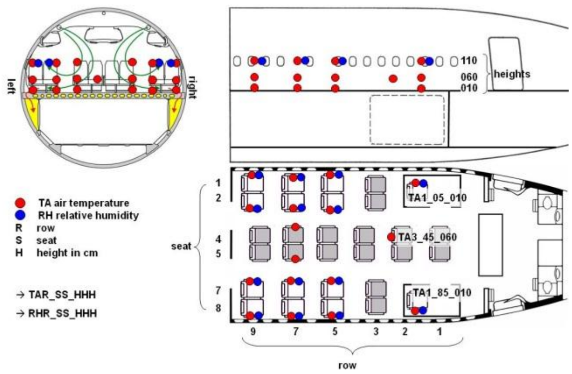


Figure 5. Probe locations in cabin

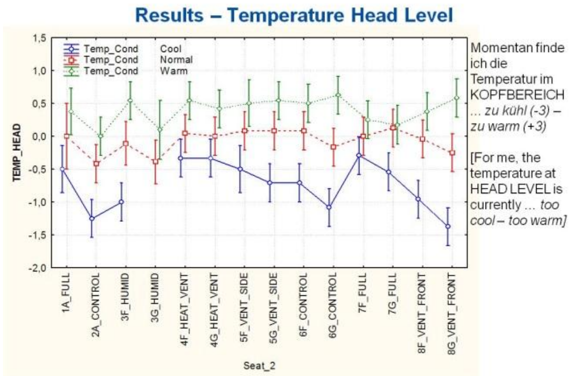


Figure 6. Experimental data analysis

This experiment provided a massive collection of data which had to be organised in groups, dates, operating conditions etc. and finally analyzed by the Medical University of Vienna (MUW). An indicative example of that analysis is illustrated in Figure 6.

Additionally, this data was filtered by BUT to determine and define the boundary conditions for the detailed CFD simulations. Steady conditions were chosen for data extraction to define the boundary conditions. Three operating points (cases) were finalized for each of the cold, normal and hot conditions, resulting in total a matrix of nine cases for the detailed simulations.

These cases were divided between Icon and BUT and then CFD simulations were carried out using the aforementioned software packages (ICON FOAMpro and STAR CCM+).

4.3 Computational Geometry & Mesh

The detailed cabin model (Figure 7) is part of an aeroplane cabin. It shows a total of thirty seats, fourteen of which are occupied by subjects in the testing procedure and the rest were manikins or control seats. The front and rear of the model are symmetry planes. The model has realistic ceiling and lateral air inlets as well as personalised inlets and outlets at the bottom. Heat sources such as human bodies and lamps are included. The model contains first class and business class seats. The inlets for fresh air and/or humid air at the passenger seats have been included. Geometry for the cabin lining and the various seats was provided separately and was assembled into one model, identical to the actual experimental cabin section used at the Fraunhofer Flight Test Facility (FTF).

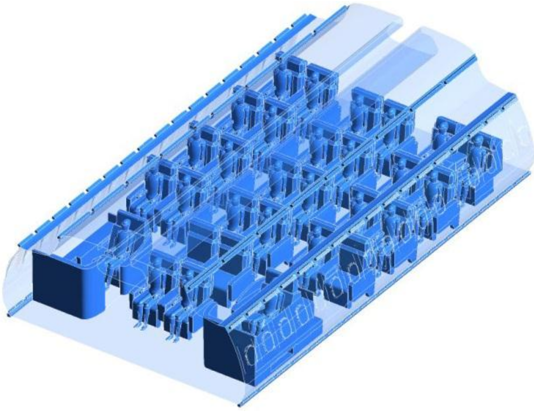


Figure 7 Sketch of the detailed cabin model

Two separate computational meshes were created by Icon and BUT for better compatibility with the two software packages used for the simulations. The two meshes shared the same characteristics, namely a trimmed unstructured hex-dominant mesh with an overall cell size of 10mm and 3 to 8 surface layers of 1cm in height, returning a total mesh size of approximately 13.5 million cells. This resolution was a compromise resulting from the limitation on the number of proprietary software licenses available. It was obviously very important to capture all details on the simulation for both approaches but equally necessary to be able to deliver results on-time and have an acceptable turnaround for both software packages.

In order to have enough accuracy near the inlets and outlets of the model, refinement zones have been included. These are boxes that lie around the inlets and outlets and define a local refinement region. In the detailed cabin model, the volume inside the refinement zones is meshed with a cell size of 2.5mm instead of 10mm. (Figure 8)

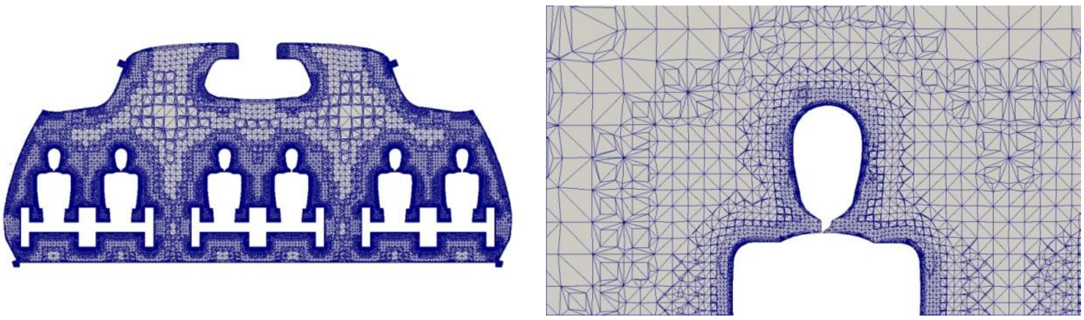


Figure 8. Computational Mesh

4.4 Numerical settings & Flow assumptions

The flow was assumed turbulent in all domains and was modelled by the k-ω SST turbulence model. Heat conduction through the seats and other solid components was ignored, instead the heat loads and temperature values at the walls were provided and used (e.g. for the body of the manikin or certain parts of the cabin lining). Buoyancy and radiation within the cabin were taken into account utilizing the P1 radiation model in ICON FOAMpro and the S2S (Surface-to-surface) radiation model included in STAR-CCM+. Solar radiation effects were ignored. In terms of assumed ambient conditions this compares with a night flight. The molecular viscosity (μ) was estimated using Sutherland's equation and was set to constant value of 1.85508E-5 Pa.s. Humidity was introduced at air inlets as a passive scalar after the simulation had reached a stage where the convergence criteria were considered to be satisfied.

4.5 Results and discussion

The effect of the main ventilation system (main inlets) was presented in detail in previous papers [5] [6]. The current study focuses on the effect of the individual ventilation and the comparison of the CFD results to actual experimental measurements.

Figures 9 to 11 illustrate the effect of the individual inlets to the velocity, temperature and relative humidity in the vicinity of the passengers who have control over them. It is also shown that drafts which might reduce thermal comfort are avoided through the use of a combination of several personal inlets.

Finally it can be seen in Figure 12 that the age of air around the seats with individual inlets is reduced, providing an overall better microclimate to the passenger.

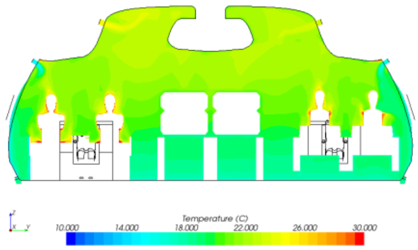


Figure 9. Temperature (C)

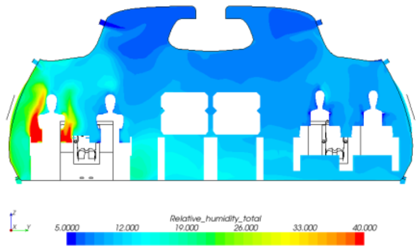


Figure 10. Relative Humidity (%)

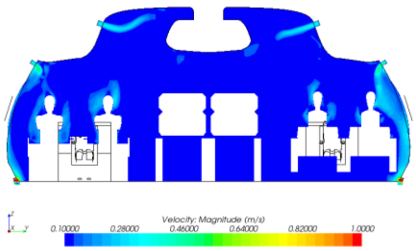


Figure 11. Velocity Magnitude (m/s)

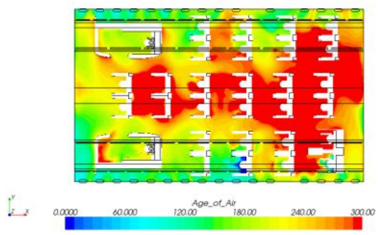


Figure 12. Age of Air

The following graphs illustrate the relative error of the CFD results in terms of temperature (Figure 13) and relative humidity (Figure 14) at probe locations, when compared to the measured values at the same locations during the subject testing at FTF.

The temperature values from CFD simulations are, in most cases, in close agreement with the testing monitored data, showing a very small deviation of approximately 5%-10%. The relative humidity, however, shows big discrepancies, reaching a deviation of around 20-30% in many locations and in some cases more than 40%.

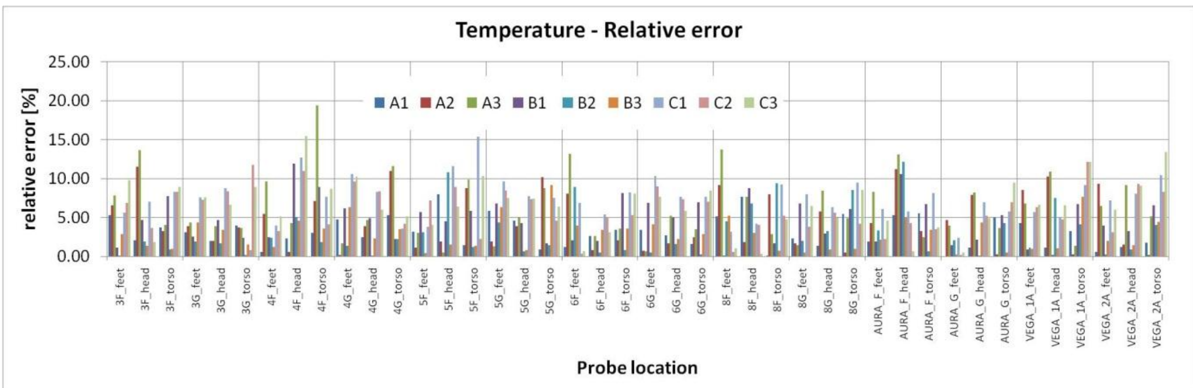


Figure 13. Temperature relative error between CFD results and experimental data

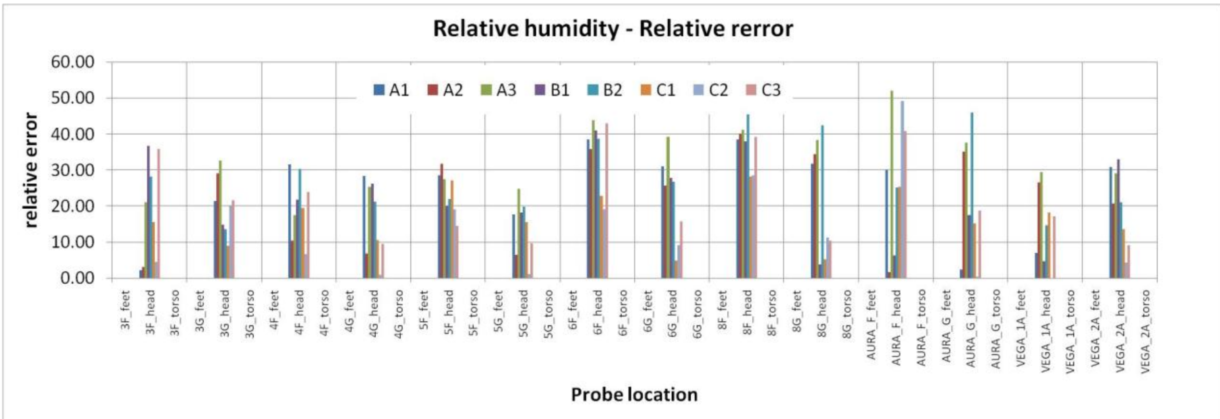


Figure 14. RH relative error between CFD results and experimental data

A closer look at the CFD numerical settings and results when compared to the monitored data indicates a number of reasons for thes deviations.

Although the flow in the cabin is time dependent and oscillates with a certain frequency, the detailed simulation study was performed in the steady state regime. It is evident from the results that certain transient effects may exist in the flow field. This brings a degree of error which appears to be relatively large for the humidity profiles in the cabin. Although all other residuals (U, T, p) gradually reduced down to at least 10e-03 and in most cases 10e-06, humidity would not converge easily, which is considered as an indication of transient effects.

Additionally, humidity was solved as a passive scalar. Usage of a coupled solver might produce better results, especially since buoyancy was calculated. Finally, ambient air before introduction of humidity passive scalar was assumed to be dry, i.e. with RH 0%. Since in most cases RH was under-predicted by CFD, initialisation with actual ambient RH measurements might prove to balance overall humidity field.

One case (A1 – cold conditions) was run transient for a very short amount of time, in order to provide a rough indication of the average humidity values compared with monitored values. As shown in the table below, running the case as transient produced more accurate results in comparison to the steady state calculations.

Relative Humidity (%)					
		CFD	Meas.	ABS error	Rel. error
Case A1 Steady State	AVG whole cabin	11.88	15.39	3.51	22.80%
	Left side	14.15	16.37	2.22	13.55%
	Right side	9.62	14.42	4.80	33.30%
Case A1 Transient	AVG whole cabin	14.23	15.39	1.16	**7.55%**
	Left side	16.60	16.37	-0.23	**-1.41%**
	Right side	11.86	14.42	2.56	**17.73%**

Table 1. Comparison of steady state to transient (Relative humidity)

5 CONCLUSIONS

This study investigated the feasibility of introducing personal and individual ventilation into aircraft cabins. The numerical investigation of personalised ventilation inlets in the aircraft first class cabin using CFD methods was described and compared to actual experimental testing carried out in a 9 m long part of an aircraft cabin. The study showed that a combination of two inlets is able to overcome the non-uniformity of the flow pattern from the main ventilation system for different suites, although in some cases even one individual inlet has a significant impact. A proprietary software code and an open source CFD code were used to conduct the CFD simulations in a steady state regime. It was found that the flowfield and temperature predicted are close to the values provided by actual physical testing. However relative humidity was shown to be under-predicted in some cases, indicating that a transient simulation approach may produce results with higher accuracy.

Time scale for flight cycle or even the time to reach steady conditions during flight is orders of magnitudes larger than the local time step needed to resolve local fluid dynamics. A trade-off between reasonable resource investment and accuracy had to be balanced to get results delivered. This limitation could be solved by the application of new local time step solvers present in ICON FOAMpro, which can provide pseudo-transient results for these simulations in affordable time. The application of these solvers is seen as a further improvement step of the methodology and will be presented in future studies.

6 ACKNOWLEDGEMENTS

Authors gratefully acknowledge the support of European Commission in the frame of project 7th Seventh Framework Programme iSPACE 234340 (innovative Systems for Personalised Aircraft Cabin Environment).

7 REFERENCES

[1] Kok J. C. et al. 2005, *Enhancement of Aircraft Cabin Comfort Studies by Coupling of Human Thermoregulation With Radiation and Turbulent Convection*, EUROMECH Colloquium 471, Göttingen.

[2] Icon, 2010, http://www.iconcfd.com/es/services/foampro

[3] OpenFOAM® and OpenCFD® are registered trade marks of Silicon Graphics International Corp. The work described in this paper are not approved or endorsed by OpenCFD Limited."

[4] http://www.ispace-project.eu

[5] Bouwman R., Tsikouris K., Forman M., Fišer J., (2011) "Open Source CFD: A Parametric Study of Individualised Cabin Environment", *Proceedings of the NAFEMS World Congress, A World of Engineering Simulation, Boston, 23-26 May 2011*.

[6] Forman, M.; Fišer, J.; Bowman, R.; Jícha, M., (2011) "Numerical Study of Personalised Ventilation Inlets In The Aircraft Suite", *Proceedings of the 12th International Conference on Indoor Air Quality and Climate, Indoor Air 2011, Austin, Texas, 5-10 June 2011*.

[7] STAR CCM+® is a registered trade mark of CD-adapco.