CHEMICAL ENGINEERING TRANSACTIONS

VOL. 43, 2015

A publication of AIDIC

The Italian Association of Chemical Engineering
Online at www.aidic.it/cet

Chief Editors: Sauro Pierucci, Jiří J. Klemeš

ISBN 978-88-95608-34-1; **ISSN** 2283-9216

DOI: 10.3303/CET1543413

Experimental and Numerical Study of an Air Lock Purging System

Marco Pontiggia[a]*, Marco Derudi[b], Paolo Paci[a], Valentina Busini[b], Renato Rota[b], Giovanni Uguccioni[a]

[a]D'Appolonia S.p.A., San Donato Milanese, Italy
[b]Politecnico di Milano, Dip. Chimica, Materiali e Ingegneria Chimica "G.Natta", Milano, Italy
marco.pontiggia@dappolonia.it

High concentrations of H_2S in offshore wells represent a major concern for personnel safety: if a significant external H_2S contamination occurs, depending on wells and process conditions, it may prove impossible an effective evacuation, and thus Temporary Refuge (TR) provisions must be set-up to provide prompt availability of safe and reliable protection to personnel. Air Locks (ALs) to enter the TR may be necessary to ensure isolation of the safe internal environment when entering into the TR. ALs modelling is essential to verify that sufficient time for entering the TR is available to all personnel in case of accident. Nevertheless, due to the extreme conditions (high toxicity, short characteristic times, high purging air velocity etc.), experimental modeling of the AL can prove difficult and very expensive. Given the importance of ALs efficiency in a real emergency situation, simulation of its performances in realistic condition and optimization of the design of air purges to ensure the required efficiency is a factor of extreme importance for the overall safety of the installation. In this work, the purging efficiency of a typical AL has been analyzed through a combined approach of experimental tests and CFD simulations, to prove the capability of CFD modeling to analyse real AL conditions. A scaled model have been realized and analyzed using CO_2 as tracing gas to determine the concentration field; even if the realization constraints above mentioned do not allow for a full scaling of all the involved variables, fluid-dynamic conditions have been set to reproduce real AL purging capabilities as close as possible. Experimental results have been used for a fine tuning and validation of the CFD tool in an operative range close to a real configuration, through the comparison of the obtained flow and concentration fields with those predicted by the CFD simulations of the experimental set-up. Subsequently, the tuned CFD approach has been used to simulate a typical AL and to check its performances.

1. Introduction

Temporary Refuges (TR) are essential for offshore/onshore facilities characterized by scenarios of large toxic gas release (for instance offshore wells with high concentration of H_2S) when evacuation time is not compatible with safe personnel escape. TR entrances must be designed in order to grant a fast and reliable access to the safe area; this task is usually accomplished through the use of Air Locks (ALs) optimized in order to maximize the washing efficiency and to reduce the quantity of fresh air (Lu et al, 2005). Test chambers aimed at fully reproduce and determine the flow, temperature and concentration field inside the AL can prove rather expensive due to severe constraints arisen from gas toxicity, material resistance, and instrumentation sampling time, leading to the need of very low scaling-factor (1:1 or 1:2) in order to properly represent the AL (Bauer et al., 2014). On the other hand, the use of Computational Fluid Dynamic (CFD) codes is rapidly spreading as an advanced modelling technique in a large variety of safety applications (Aresta et al., 2013; Busini et al., 2011; Derudi et al., 2008).
In this paper a combined approach has been used for the optimization of a typical AL representing those that can be applied in real conditions: a medium-scale (1:10) Test Chamber (TC) has been realized in order to reproduce the AL features as close as possible considering the above mentioned constraints; CO_2 has been

Please cite this article as: Pontiggia M., Derudi M., Paci P., Busini V., Rota R., Uguccioni G., 2015, Experimental and numerical study of an air lock purging system, Chemical Engineering Transactions, 43, 2473-2478 DOI: 10.3303/CET1543413

used as tracing gas to avoid toxicity issues. Results of the TC has provided only a qualitative estimation of the real AL purging efficiency but they have granted a solid validation case for the CFD simulation in a situation close to the real purging conditions. The validated CFD approach has been used to simulate the AL performances. The combined use of experimental runs and computational simulations can prove very cost-effective as discussed by (Zitek et al., 2010) for airliner cabin micro environmental control, or (Wang et al., 2007) for cleanrooms for particles and microbiological contamination control; few works (Poussou et al., 2010) address also the problem of moving body in the control room. The experimental set-up can be simplified in order to avoid the major issues (mechanic, safety, construction) thus strongly reducing the required resources; on the other hand, the reliability of the results provided with a general purpose code is validated in a situation very close to the real one.

2. Material And Methods

2.1 Air Lock geometry

The realistic AL considered (Figure 1) in the analysis is assumed to be designed for 28 people: outlined squares represent the personnel position during the purging cycle and a nozzle is centred on each square to provide individual air-washing. Rectangles on the right side represent aspiration grids, placed close to the external door (door connecting the external environment with the AL). Estimated time for people entering in the AL is 14 s, while the purging system is designed to accomplish an air exchange in about 40 s.

2.2 Small-scale Test Chamber (TC)

In order to maintain the same fluid dynamic regime of the flow inside the AL, Reynolds number ($\mathrm{Re} = \rho v D / \mu$) of a real AL must be conserved in the TC. Since air will be used as purging gas inside the TC, density (ρ) and viscosity (μ) are fixed; therefore the air velocity (v) in the TC is given by:

$$\frac{\rho v_{AL} D_{AL}}{\mu} = \frac{\rho v_{PM} D_{PM}}{\mu} \quad \Rightarrow \quad v_{PM} = v_{AL} \frac{D_{AL}}{D_{PM}} = S_F \cdot v_{AL} \tag{1}$$

where the AL subscripts indicate the values in the real Air Lock and TC subscripts the values in the Test Chamber and S_F is the scale factor. Time required for and air exchange can be therefore estimated dividing the volume (V) by the flow rate ($Q = vA$, where is the inlet surface):

$$\tau_{PM} = \frac{V_{PM}}{v_{PM} A_{PM}} = \frac{V_{PM} / S_F^3}{v_{AL} A_{AL} / S_F} = \frac{\tau_{AL}}{S_F^2} \tag{2}$$

Since the instrument sampling time is about 5 s and τ_{AL} is 40 s, to achieve at least one instrument measurement per air exchange a maximum scale factor equal to 2.8 is required. Therefore, using the TC as the exclusive modelling technique for AL optimization, a nearly real-scale model is needed in order to assure a proper number of measurements, thus involving very large costs and resources.

The combined use of TC and CFD simulations has permitted to use a scale factor of 10. Air inlet velocity has been chosen in order to consistently reproduce in the TC the real dynamic of air jets and the main features of the flow field; a lower Reynolds number has been set, due to the above mentioned constraints. As a consequence, the air purging system of the TC results less efficient than the real one in the gas concentration reduction, still maintaining its representativeness of the real flow field. Results of the TC have been used to validate the capabilities of the CFD tool used for the TM (Theoretical Model) development to reproduce the concentration profiles for an application similar to the real case.

Two experimental tests (with and without personnel shapes, Figure 2) with CO_2 as tracing gas have been performed. Tracing gas has been fed through a 1/8” diameter pipe, placed in correspondence of the inlet door in the AL, with a total flow rate of 3.3 Nl/min. Air purging flow rate through 28 air nozzle 4mm diameter, has been set to 70 Nl/min.

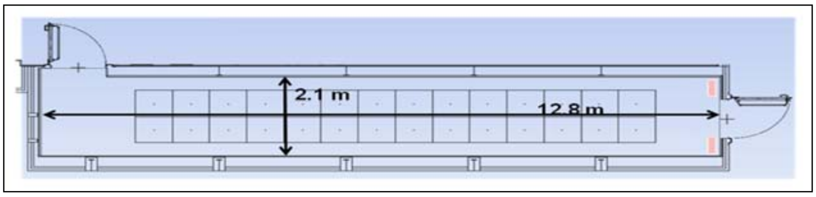

Figure 1. Realistic Air Lock geometry

Figure 2. Empty TC and TC with 28 personnel shapes

The small diameter of the CO_2 orifice, grants a gas velocity of about 23 m/s, that assures a good penetration of the tracing gas within the model, along with a relevant concentration of CO_2 throughout the whole chamber. CO_2 has been fed until the reaching of steady-state concentration profiles, as for preliminary tests. CO_2 concentrations have been measured sampling small quantities of gas and analysing them with an HORIBA PG-250 gas analyser, provided with IR detector for CO and CO_2 detection within the gas phase.

2.3 Theoretical Model (TM)

The TM has been developed using Ansys Fluent software (Fluent, 2009), a Computational Fluid-Dynamics code that solves the Navier-Stokes equations along with model-specific transport equations, discretized on a computational grid. For the AL simulation, along with Navier-Stokes equations for momentum transport, the species transport and the turbulence models have been activated. Input data for momentum evaluation are velocity or mass flow at inlet boundaries and pressure at open boundaries; close boundaries (such as walls) has been treated as no-slip surfaces (velocity has been set equal to zero).

For the realization of the AL simulation the standard k-ε model (Jones and Launder, 1972) has been used, due to its very good arrangement between solution accuracy and simulation stability. Turbulence input data for inlet and open boundaries has been specified as turbulence intensity and hydraulic diameter (or turbulence length scale); input data have been calculated accordingly to Fluent manual advices. Wall-type boundaries have been treated as smooth surfaces (turbulence generation due to wall roughness has been neglected).

The AL geometry has been discretized using an unstructured grid, in order to model the complex geometry of the AL (grids, nozzles, people in the AL) refining the mesh size only on small details; the tetrahedral cells have been successively clustered in polyhedral elements, in order to enhance cells quality and simulation stability (Figure 3).

The results independency on meshing criteria has been checked comparing the result of a sample grid simulating the empty AL with an initial uniform H_2S concentration of 25000 ppm to the results of a coarser and a finer grid. Characteristic time needed to halve the initial gas concentration (t_2) and to reduce the concentration of one order of magnitude (t_{10}) have been calculated and compared. Details of the analysis have been resumed in Table 1.

3. Results

3.1 TM validation

TM set-up geometry for the CFD code validation and tuning is shown in Figure 4, for both the empty chamber and the chamber with personnel configurations.

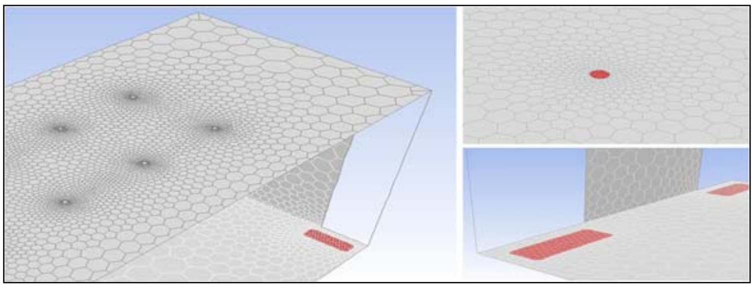

Figure 3. Theoretical Model, grid details

Table 1. Grid sensitivity analysis

	Coarse grid	Sample grid	Fine grid
Grid spacing on Nozzle [mm]	8	4	2
Grid spacing on Aspiration Grid [mm]	60	30	15
Growth rate	1.2	1.2	1.2
Grid Max. Size [mm]	1500	1000	750
# of Nodes	600k	865k	1220k
t_2 [s]	42	39	38
t_{10} [s]	138	128	127

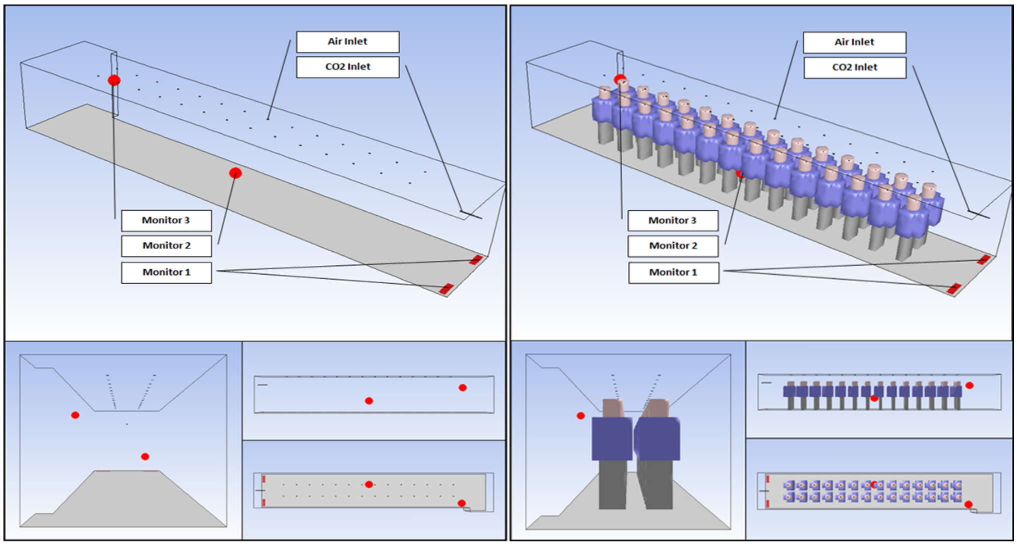

Figure 4. Theoretical Model, geometry configuration used for model validation

Comparison between experimental data and CFD simulations have been performed on the basis of CO_2 concentration on three monitor points (M1, M2 and M3), shown in Figure 4 as circles.
The comparison between experimental data and model predictions for the three monitor points for both the configurations is shown in Figure 5. For the empty chamber configuration the comparison show a very good agreement in terms of steady-state gas concentration, meaning that the CFD code is able to reproduce the flow field and the concentration field inside the chamber. Slightly faster transient is observed in M1, meaning that the CFD tends to overestimate the turbulence, and therefore the mixing, in the AL; this is a well-known tendency of the RANS (Reynolds Averaged Navier-Stokes) turbulence closure models, such as the k-ε model used for these simulations; nevertheless the discrepancy is negligible, compared to the level of uncertainty of the provided boundary conditions (surface roughness, analytical uncertainties etc). A better agreement in transient times is observed for M2 and M3, where the curves are substantially equivalent. The scattered conformation of the M3 profile is probably due to the large grid element size and relevant numerical instability. The profile could benefit from a finer calculation grid, but this procedure involves a larger calculation load, without providing any significant physical results improvement (only a mitigation of numerical fluctuations). Also in the model with 28 personnel shapes, M1 and M3 show a very good agreement between experimental data and simulation calculations, both in terms of maximum CO_2 concentration and transient durations. Moreover grid element dimension near M3 has been reduced, due to the proximity with personnel shapes: numerical fluctuations are avoided and a flat profile has been obtained. On the other hand, some differences are observed in M2: gas concentration is slightly overestimated and purging time is underestimated (purging in CFD simulation is faster).

3.2 AL simulation

Since the TC reproduces the scaled real geometry, for the flow field simulation inside the AL, the same virtual geometry along with the same meshing criteria has been retained, removing the scale factor. In order to account for H_2S entering the AL, two different contributions have been modelled: a convective flux (a flux associated with a fluid velocity) has been imposed at the door connecting the AL with the external, contaminated environment, and a diffusive flux (a flux associated with a concentration gradient) has been imposed at people surface.

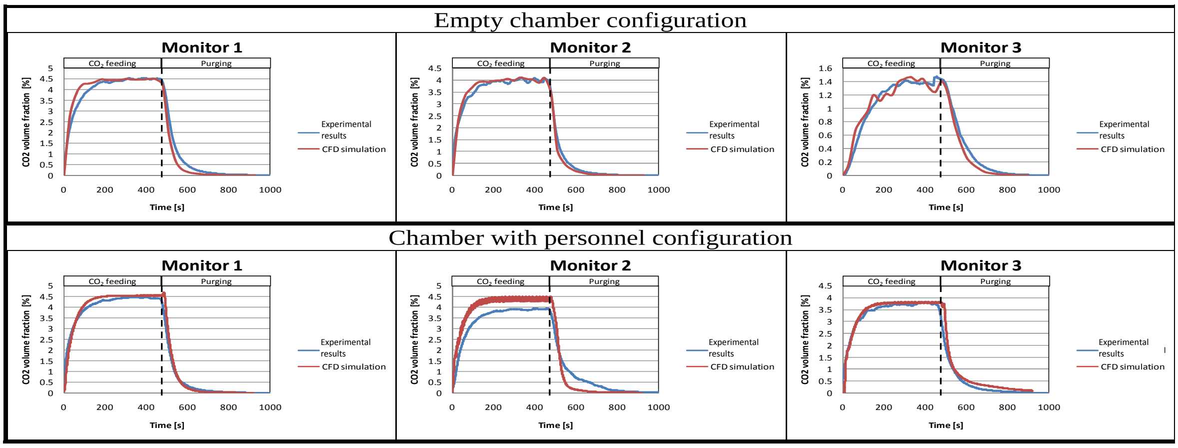

Figure 5. Comparison between CO_2 volume fraction evaluated fromTM and TC

The convective flux has been intended to mimic the H_2S entrained by the personnel entering the AL: a person-shaped inlet boundary has been created on the inlet door surface, and a uniform flux with a velocity of 1.25 m/s and a concentration of 25000 ppm (representing the maximum concentration that could be expected in case of severe toxic gas leakages) has been conservatively imposed for the 14 s representing the personnel entering phase. The surrounding free-space of the door has been modelled as a pressure outlet, in order to account for H_2S purged by pressurization flow, during the entering procedure. After the personnel entering phase, both the door and H_2S inlet boundaries are set to wall-type boundaries to represent the closure of the door. The diffusive flux, on the other hand, has been intended to mimic the H_2S entrapped in personnel clothing; a constant thickness layer has been assumed over the whole personnel clothing surface, thus creating a shell around each body. Gas diffusion has been kept until the gas in the shell has been released. A specific routine has been developed and implemented in the core solver using the User-Defined Function (UDF) approach provided by Fluent: the amount of H_2S released through the diffusive flux (calculated by the code as a function of concentration gradient and effective diffusivity) is monitored for each simulation time step, and the UDF regulates the surface concentration in order to locally stop the H_2S flux where the designed threshold has been reached. Therefore, the UDF allows to set-up a total amount of released H_2S (expressed as the H_2S amount hold by a constant thickness layer enclosing the personnel shapes), and to evaluate the AL clothes-washing performance: surfaces directly exposed to fresh air jets (such as the head) experience larger diffusivity values and larger concentration gradients, thus leading to a faster H_2S release; cells belonging to these surfaces have a shorter emission time. Hidden surfaces (such as the bottom of the backpack), on the other hand, are characterized by slower velocities, H_2S recirculation, thus involving small diffusivity values and small concentration gradients along with longer emission time. The analyzed AL procedure includes personnel entering phase (14 s) and two purging cycles (the first cycle including two air changes for a total of 80 s and the second cycle including one air change, for a total of 40 s), up to a total of 134 s simulation. During the personnel entering phase a pressurization air flow has been used; air flow has been switched to purging air flow after the door closure. H_2S layer thickness is a key parameter: assuming, for instance, a uniform layer of 2.5 cm, the volume of H_2S trapped in the personnel clothes is equal to 30% of the total volume occupied by the personnel and it is therefore considered strongly conservative; nevertheless, there are no experimental or theoretical evidences for the univocal evaluation of this contribution.

A simulation without the diffusive flux has been performed and results have been shown in Figure 6A. Convective flux contribution curve falls below the 10 ppm threshold at about 35 s: due to the small personnel velocity in the entering phase, convective flux has a minor penetration and its effect is circumscribed to the initial zone of the AL, thus allowing a very fast H_2S depletion. The critical contribution is represented by diffusive flux because it allows H_2S to spread in the whole chamber length.

Results highlighted that at early stages a 25000 ppm H_2S concentration is observed close to the inlet door, due to the contaminated air entrained by the entering personnel. Due to the relatively low inlet velocity, H_2S penetration is limited to the initial empty space between the door and the first personnel row. Gas diffusion from personnel clothes is also observed, thus dispersing H_2S around people in the whole chamber. At later stages, a large accumulation zone at 300 ppm is observed in the final section of the AL (close to the door connecting the AL and the TR): due to the absence of aspiration grids and fresh air jets, concentration in this area cannot be efficiently depleted. Results highlighted that at early stages a 25000 ppm H_2S concentration is observed close to the inlet door, due to the contaminated air entrained by the entering personnel.

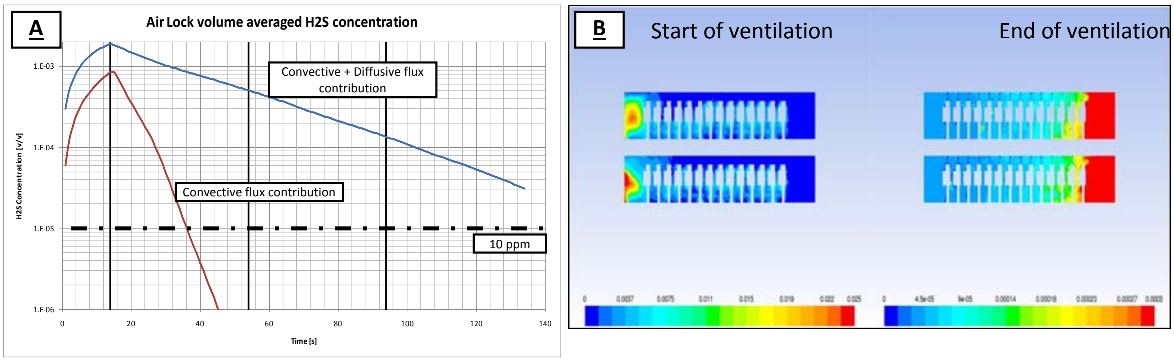

Figure 6. Simulation results: (A) Convective and Diffusive flux contribution analysis, (B) Concentration plot

Due to the relatively low inlet velocity, H_2S penetration is limited to the initial empty space between the door and the first personnel row. Gas diffusion from personnel clothes is also observed, thus dispersing H_2S around people in the whole chamber. At later stages, a large accumulation zone at 300 ppm is observed in the final section of the AL (close to the door connecting the AL and the TR): due to the absence of aspiration grids and fresh air jets, concentration in this area cannot be efficiently depleted.

4. Conclusions

A combined approach has been applied to the analysis and optimization of a typical AL purging system; a full scale reproduction of the AL, due to the large scale required and the toxicity issues can prove very expensive. Nevertheless results obtained with a small-scale model with inert tracing gas have provided both a qualitative behaviour of the internal flow field and a consistent validation and tuning data-set for the CFD model; CFD validation, achieved through the comparison of tracer gas concentration in three monitor points, has shown a good agreement with experimental data (Figure 4).

Tuned CFD approach has been subsequently used to optimize the AL geometry and parameters. H_2S inlet has been modelled with two different contributions: a convective flux from the door during the personnel entering phase (mimicking the entrained contaminated air) and a diffusive flux from the personnel surfaces (mimicking the H_2S entrapped in the clothes). The simulation of the starting configuration (Figure 6B) has highlighted a recirculation area with a significant H_2S accumulation. Three air exchanges have been used for purging. This approach has proven to be applicable in real cases of Air Locks.

References

Aresta P., Pilato S.D., Derudi M., Nano G., 2013, CFD - Assisted safety design in a flue gas treatment plant retrofit, Chemical Engineering Transactions, 31, 865-870.

Bauer E.R., Matty T.J., Thimons E.D., 2014, Investigation of Purging and Airlock Contamination of Mobile Refuge Alternatives, NIOSH, Publication 2014–116.

Busini V., Pontiggia M., Derudi M., Landucci G., Cozzani V., Rota R., Safety of LPG rail transportation, 2011, Chemical Engineering Transactions, 24, 1321-1326.

Derudi M., Pontiggia M., Nano G., Rota R., 2008, A test chamber for the assessment of human exposure to fine particles, Chemical Engineering Transactions, 16, 403-410.

FLUENT Inc., 2009. Fluent 12 User's Guide. Lebanon

Jones W. P., Launder B. E., 1972, The prediction of laminarization with a two-equation model of turbulence. Journal of Heat Mass transfer, 15, 301-314.

Lu W.Z., Leung A.Y.T., Yan S.H., So A.T.P., 2005, A preliminary parametric study on performance of SARS virus cleaner using CFD simulation, Int. J. Numer. Meth. Fluids, 47, 1137-1146.

Poussou S. B., Mazumdar S., Plesniak M. W.,, Sojka P. E., Chen Q., 2010, Flow and contaminant transport in an airliner cabin induced by a moving body: Model experiments and CFD predictions. Atmospheric Environment, 44, 2830-2839.

Wang F. J., Zheng Y. R., Lai C. M., Chiang C. M., 2007, Thermal comfort and contamination control assessment of a cleanroom. Proceedings of IAQVEC2007, 6th International Conference on Indoor Air Quality, Ventilation & Energy Conservation in Buildings.

Zitek P., Vyhlidal T., Simeunovic G., Novakova L., Cizek J., 2010, Novel personalized and humidified air supply for airliner passengers. Building and Environment, 45, 2345-2353.