Topic B6: Prediction & measurement

# UNSTEADY CFD SIMULATIONS FOR PREDICTION OF AIRFLOW CLOSE TO A SUPPLY DEVICE FOR DISPLACEMENT VENTILATION

Mathias CEHLIN, Taghi KARIMIPANH*, and Ulf LARSSON

Faculty of Engineering and Sustainable Development, Department of Building, Energy and Environmental Engineering, University of Gävle, Sweden

**Corresponding email: tkh@hig.se*

Keywords: Unsteady CFD, URANS, Displacement ventilation

## SUMMARY

Modern diffusers applied in the field of ventilation of rooms are often complex in terms of geometry, including perforated plates, dampers, guide rails, curved surfaces and other components inside the diffuser, with the intention to create satisfying thermal comfort for the occupants. Also connecting ducts can be different for the same diffuser in different situations, affecting the supply velocity profile. It is obvious that simulation of airflow and air temperature particularly in rooms with displacement ventilation is very troublesome, particularly if the near-zone of the diffuser is of interest.

Experiments commonly indicate very high turbulence intensities in the near-zone of displacement ventilation supply devices, especially close to the floor where high mean flow gradient occurs. This indicates that the air flow from inlet devices designed for displacement ventilation might be very unstable; the position of the stream leaving the diffuser and entering the room is changing with time, hence diffusion of momentum and temperature are increased. This effect is not captured in RANS simulations, since it is performed with the assumption of time-independent conditions.

In this paper URANS simulations were performed for prediction of velocity and temperature distribution close to a complex air supply device in a room with displacement ventilation.
The presented study show that unsteady simulations with the realizable turbulence $k$-$\varepsilon$ model generates too high eddy viscosity and therefore damps out the unsteadiness of the flow especially inside the diffuser.

## INTRODUCTION

Because of transient behavior of almost all existed flows in nature, the real life flow modeling and simulations are not easy task. By increasing the computational capacities transient flow modeling has been used in many applications e.g. water distribution techniques (Wood et al 2005), pulsing flow ventilation, displacement ventilation, residence time of a pollutant in a room and some turbo-machinery flows etc. The flow issued from a displacement supply device is a complex flow and one cannot be sure how developed it is. In addition, the perforated plate at the exit, i.e. forced changing in geometry, causes more unsteadiness, see figure 1.

In displacement ventilation systems, the air is supplied at floor level in the zone of occupation. The air is supplied with a lower temperature than the ambient air and the ventilation air is therefore both a source of momentum and buoyancy. Close to the diffuser, air is accelerating in the vertical direction and the air undergoes a free fall. Rooms with displacement ventilation are often associated with complaints about drafts and dissatisfaction due to relatively high velocities and low temperature at the ankle level (Wyon and Sandberg 1989). In addition, the turbulence intensity and the dominant frequency of the flow fluctuation affect the draft (Fanger et al 1988, Fanger and Pedersen 1977). Environmental parameters are not only influenced by the type of ventilation system but also diffuser characteristics. It is obvious that the limiting factor for displacement ventilation, in particular schools and offices, seems to be thermal comfort.

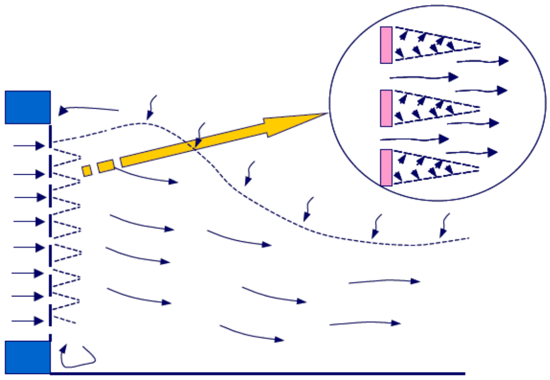


Figure 1. Air flow pattern with perforated panel diffusers, see Bulter (2002)

Prediction of room airflows is a challenging task because of the complex nature of turbulence. Simulation of airflow in rooms with displacement ventilation are particular troublesome, because air is supplied under-tempered at low velocities. In ventilation strategies, the commonly used approach is time averaging and steady state calculations. In case of displacement ventilation supply devices these approximations may cause some uncertainties close to the supply outlets. This may be suspected case in near-zone regions, i.e. close to the supply, where we know that the flow situation so called “gravity current” occurs. This transient behavior causes also draught problem in the region and the occupant may not be able to seat there and that is to say practically one meet a useless space in workplace. Therefore we may solve this situation by a time dependent (unsteady) airflow solution techniques.

Solving the Navier-Stokes equations with no approximations, Direct Numerical simulation (DNS), is extremely memory intensive. It is more or less infeasible to performed DNS for room airflows with the capacity and speed of present computers. Therefore turbulence is often modeled, and these models can be divided into two major groups: Large Eddy Simulation (LES) and Unsteady Transient Reynolds Average Navier Stokes (URANS) simulation. In LES large vortices are resolved by the computational grid and the smaller turbulent scales are completely modelled. LES has rarely been applied to modeling indoor air flows because of restrictions in computer power due to the requirement of a very fine (Peng and Davidson 2000, Chen et al 2007). Here because of time cost and relevance to ventilation practice we used URANS method.

The purpose of the paper was to study the use of URANS simulations to achieve satisfying prediction of velocity and temperature distribution close to a complex air supply device in a room with displacement ventilation. The predictions were compared with experimental data.

## METHODOLOGIES

### Physical model and experimental setup

The room under consideration had the following dimensions, length by width by height, 3.4m×4.1m×2.7m, see Figure 2. The air was supplied through a flat low-velocity diffuser with dimensions, width by height, 0.49m×0.40 m, located 0.09 m above the floor level at one of the walls. The diffuser had a perforated plate with 49×24 round openings with diameters of 6.8 mm. The first row of openings was located 0.1 m above floor level. The total free area of the diffuser plate was 427.09 $cm^2$.

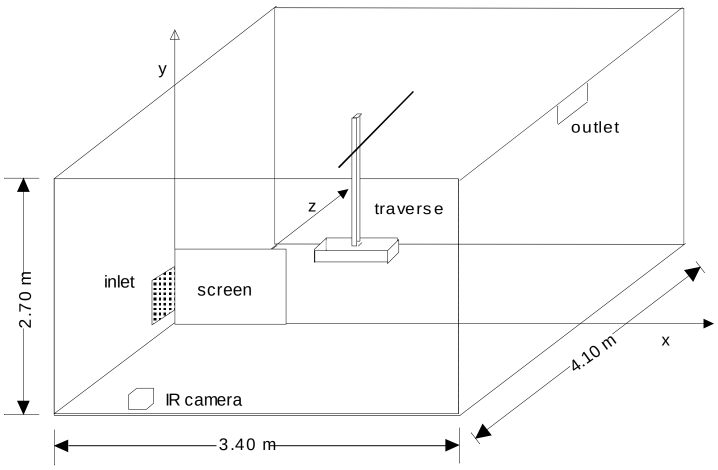



Figure 2. Layout of the physical model.

Experiments were performed with hot-wire anemometer, thermocouples and infrared thermography, see Cehlin and Moshfegh (2010) for detailed information.

A CFD model was built in a way geometrically similar to the room and the supply diffuser, see Figure 3. Only half of the diffuser was included in the CFD model due to symmetry. All 25×24 openings were for simplicity modeled as squares of 6.0×6.0 mm, except for the openings along the symmetry plane which was modeled by 3.0×6.0 mm. This resulted in a total free area of 211.68 cm2, which was 0.9% lower than in the experiments. The velocity profile of the air entering the diffuser from the duct was defined at the top of the diffuser.

The discretization of the governing equations was based on the finite volume method, and the computations were carried out using Ansys Fluent 14.5. The SIMPLE algorithm was used for coupling velocities and pressure. The grid dependence was tested with different grid densities. The simulations were performed on a computer with two parallel 2.4GHz processors each having 6 cores.

A non-conformal mesh setup was applied between the inlet openings and the room. This interface makes it possible to have separate cell-sizes for the inlet openings and the rest of the room. The ratio between the cell-sizes at the interface was 1/5. The final 3-D model consisted of 5,384,880 hexahedral cells (majority of the cells located inside the diffuser) where each opening for the perforated plate consisted of 25 elements. The mesh has a higher density near the walls, especially at the perforated plate, in order to capture in detail the airflow pattern leaving the diffuser with higher density near the inlet diffuser. The enhanced wall function

was used and in the final mesh y+ values were below 1 inside in diffuser (invoking two-layer zonal model) and below 5 in the room.
The simulation was declared converged when the parameters and the residuals were no longer changing with successive iterations and the residuals for all variables were less than $10^{-8}$. The time-step was set to $10^{-4}$ s with a maximum of 20 iterations per time-step.

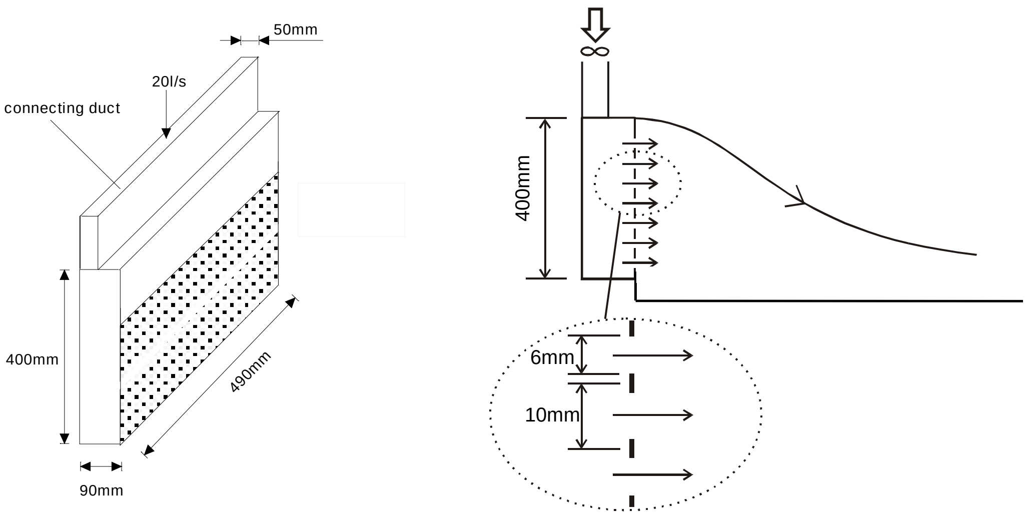
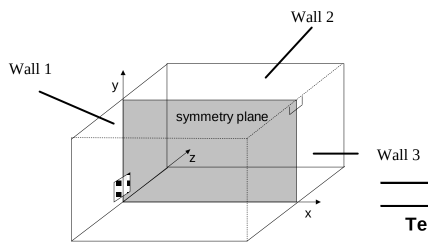



Figure 3. Full 3-D model of the diffuser.

The boundary conditions of the simulations duplicated measurements in the test-room, see Figure 4. The buoyancy effect was modeled based on the incompressible ideal gas law.




| | wall 1 | wall 2 | wall 3 | floor | ceiling | inlet |
|---|---|---|---|---|---|---|
| **Temp. (°C)** | 21.7 | 22.0 | 22.2 | 21.5 | 23.0 | 16.9 |

Figure 4. Temperature boundary conditions for the walls and the inlet.

For unsteady, three dimensional, non-isothermal, incompressible turbulent flow and using the ideal gas law, the continuity, Navier-Stokes and energy equation are given by:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + \frac{\partial (U_j U_i)}{\partial x_j} = -\frac{\partial P}{\rho \partial x_i} + \frac{\partial}{\partial x_j}\left( \nu \frac{\partial U_i}{\partial x_j} - \overline{u_i' u_j'} \right) + \frac{(\rho - \rho_0)}{\rho} g_i \tag{2}$$

$$\frac{\partial T}{\partial t} + \frac{\partial (U_j T)}{\partial x_j} = \alpha \frac{\partial}{\partial x_j}\left( \frac{\partial T}{\partial x_j} - \overline{u_i' \theta} \right) \tag{3}$$

where $\alpha$ is thermal diffusivity and $\overline{u_i'u_j'}$ , $\overline{u_i'\theta}$ constitute the second-moments statistical correlations or so-called Reynolds stresses and turbulent heat fluxes. These unknowns must be modelled in order to close the system of equations. The turbulence model used in this study is the Realizable $k$-$\varepsilon$ model, see Ansys Fluent Manual 2013.

## RESULTS AND DISCUSSION

Figure 5 shows the mean velocity contours and the mean temperature field at the symmetry plane close to the supply diffuser. Three strong recirculation bubbles can be observed in the figure, where two are located inside the diffuser affecting flow development and one just beneath the supply.

The resolved and modeled turbulent fluctuations at z = 0 m are presented in figure 6. As one can clearly see the resolved fluctuations are much lower than the modeled one especially inside the diffuser indication too much modeled dissipation (too large turbulent viscosity which dampens resolved fluctuations). Too much eddy viscosity damps out the unsteadiness of the flow and showing the weakness of the turbulent model used in this study. It is worth mentioning that more sophisticated like the v2f model, see Parneix and Durbin (1997), and non-linear eddy-viscosity models like the realizable Reynolds stress algebraic model (Shih et al 1993) could overcome this shortcoming.

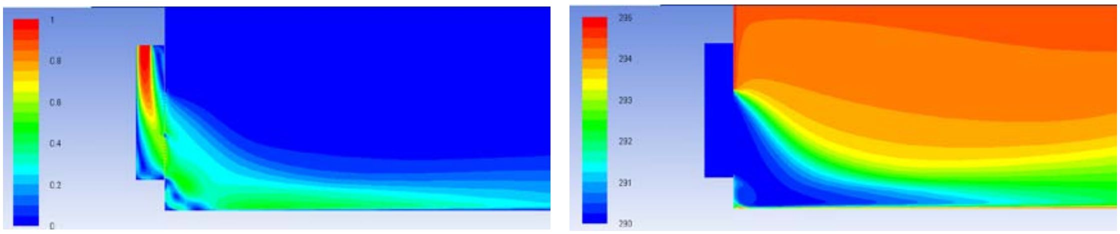


Figure 5. Mean velocity distribution (left) and mean temperature distribution (right) at z = 0m.

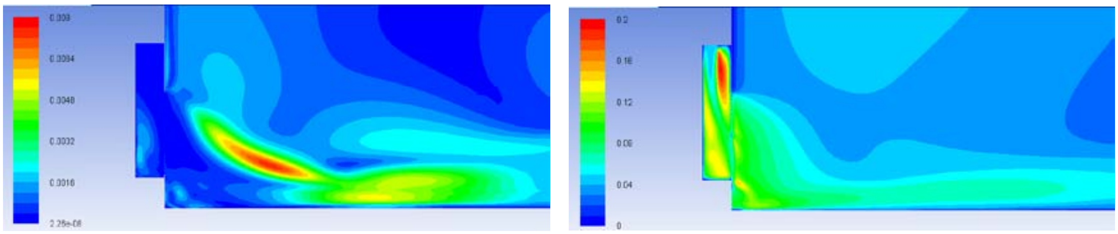
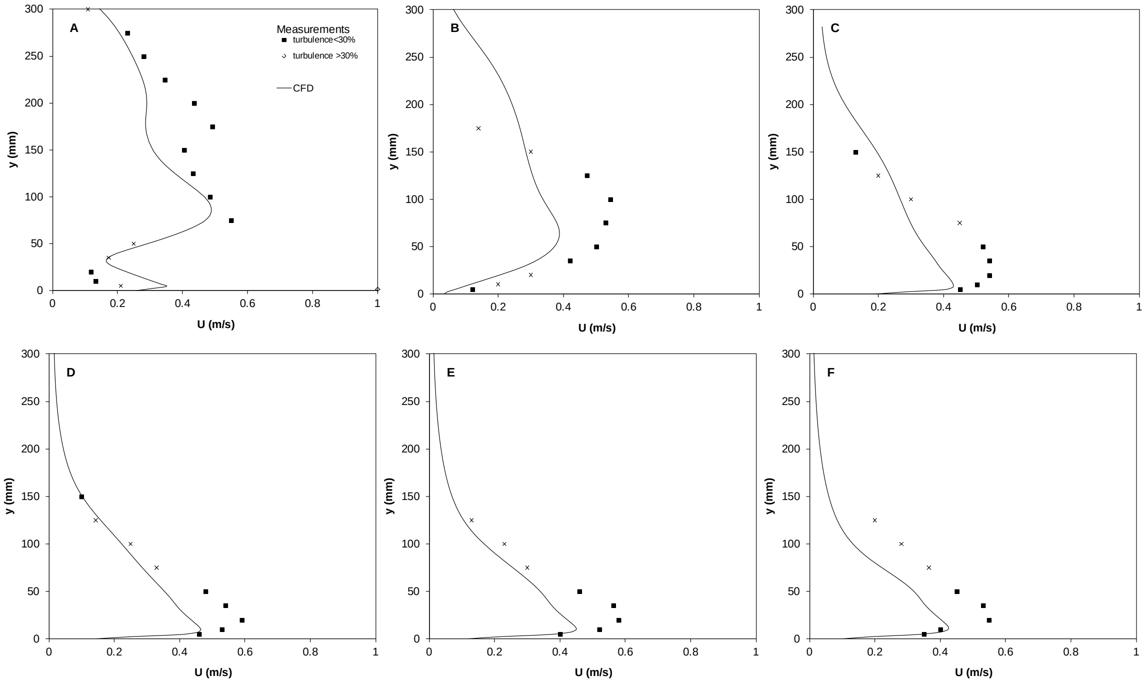


Figure 6. Resolved turbulent fluctuations, $\overline{v'}$ at symmetry plane (left). Modeled turbulent fluctuations, $v''$ at symmetry plane (right).

Figure 7 compares the predicted mean velocity profiles with measured values at corresponding distances from the diffuser for the symmetry plane. As mentioned before, because of the dissipative characteristic of the turbulence model used the predicted mean velocities are slightly under-predicted. The trend is however captured pretty well but not the levels as is expected due to the restriction of the used turbulence model. The realizable $k$-$\varepsilon$ turbulence model used here assumes isotropic turbulence which is not really true in this case due to impingement, curvature effects and separation at the upstream region.




× = measured mean velocities with turbulence intensity > 30%
■ = measured mean velocities with turbulence intensity > 30%

Figure 7. Mean velocity profiles at z = 0m at different distances from the supply device. **A**: x=0.05m; **B**: x=0.1m; **C**: x=0.2m; **D**: x=0.3m; **E**: x=0.4m; **F**: x=0.5m.

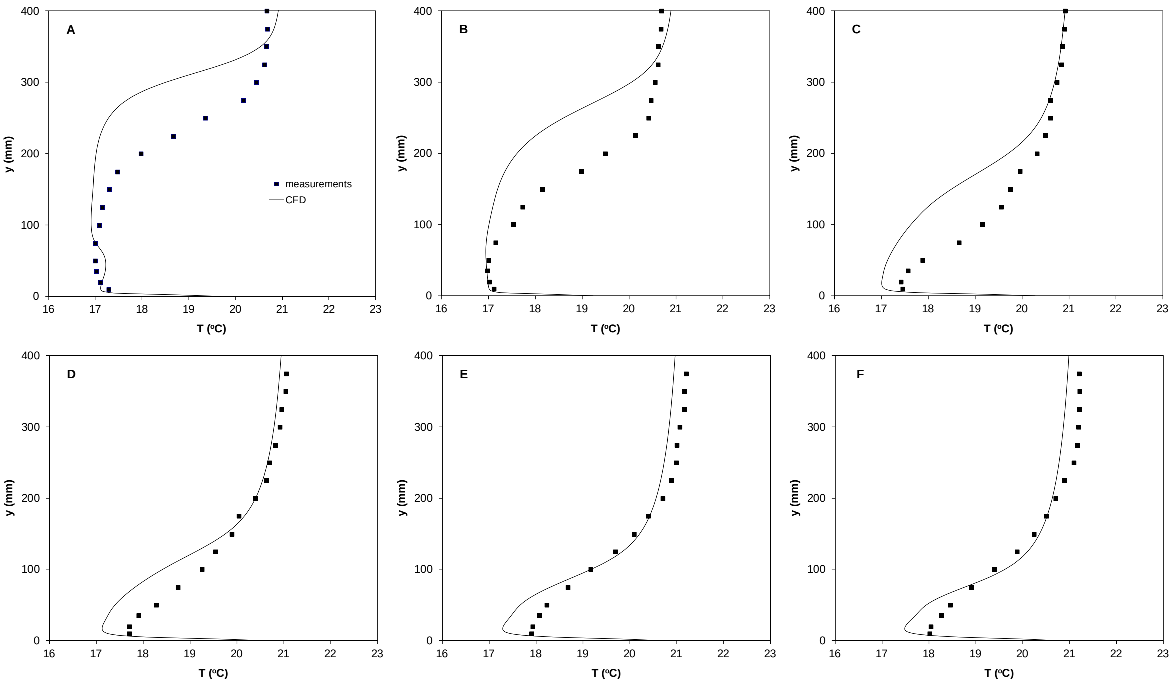



Figure 8. Mean temperature profiles at z=0m at different distances from the supply device. **A**: x=0.05m; **B**: x=0.1m; **C**: x=0.2m; **D**: x=0.3m; **E**: x=0.4m; **F**: x=0.5m.

Figure 8 compares the predicted mean temperature profiles with measured values at corresponding distances from the diffuser for the symmetry plane. The trend is captured well and also the levels are predicted fairly well after 0.3 m downstream the diffuser, i.e. when the jet leaves the near-zone region because of the known gravity current phenomena in displacement ventilation systems. The predicted temperature profiles very close to the supply device have some discrepancies compare to measurements because of failure of the turbulence model to fully predict the flow complexity inside the diffuser. As can be observed form figure 9, taken by an infrared camera, the flow pattern close to the supply outlet cause an instable region so called for “the gravity current”. In our ongoing project, we are going to verify sophisticated non-linear models with this type of imaging. For more detailed image analysis and successes in using infrared camera imaging for CFD validation see Karimipanah et al (2014).

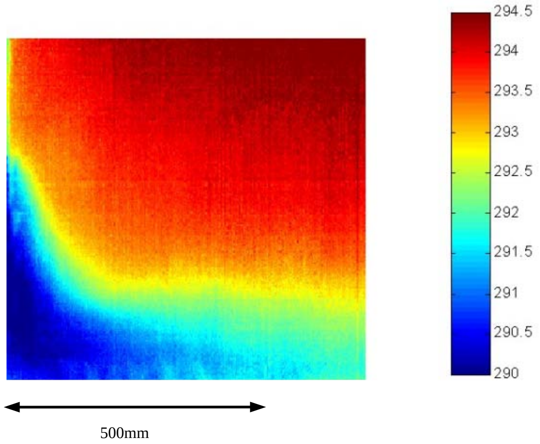



Figure 9. Temperature field imaged by infrared camera for near-zone region (close to the supply terminal).

## CONCLUSIONS

- This effort shows that the simulation of airflow in rooms with displacement ventilation is still troublesome.
- It is also pointed out that for displacement ventilation supplies using the under-tempered air at low velocities the flow will be influenced by both momentum and buoyancy forces.
- The small details of a supply air diffuser have an obvious influence on the environmental parameters and the thermal comfort in particular displacement type of ventilation.
- The air diffusion in a room is also at least partly dominated by the supply air parameters, such as inlet velocity, airflow rate, turbulence intensity and inlet temperature.
- The presented applications show that unsteady simulations with the realizable turbulence k-ε model generates too high eddy viscosity and therefore damps out the unsteadiness of the flow especially inside the diffuser.
- Physical effects can be predicted accurately (almost qualitatively accurate) by URANS.
- Weak prediction of flow unsteadiness caused by anisotropy of turbulence inside the supply device is noticed in connection with influence of perforated plate.

- The resolved fluctuations are observed to be lower than the modeled fluctuations indicating too high turbulent viscosity which causes damping effect on the unsteadiness of the flow.
- It was suggested that in future investigations one should use more sophisticated non-linear turbulence models.

## ACKNOWLEDGEMENT

The authors are thankful for the financial support from the University of Gävle. The authors gratefully acknowledge the support received by personnel at the laboratory of Built Environment in University of Gävle.

## REFERENCES

Butler, D. (2002) Air conditioning using displacement ventilation to maximise free cooling. In: CIBSE Conference 2002.

Cehlin M. and Moshfegh B. (2010) Numerical modeling of a complex diffuser in a room with displacement ventilation. *Building and Environment* 45.10 (2010): 2240-2252.

Fanger P.O., Melikov, A.K., Hanzawa, H., and Ring, J. (1988). Air turbulence and Sensation of Draught. *Energy and Buildnings*, vol. 12, pp. 21-39.

Fanger, P.O. and Pedersen, C.J.K. (1977). Discomfort Due to Air Velocities in Spaces. Proceedings of the Meeting of Commission B1, B2, E1 of the International Institute of Refrigeration, vol.4, pp.289-296.

Karimipanah T., Larsson U. and Cehlin M. (2014) Investigation of flow pattern for a confluent-jets ventilation system on a workbench of an industrial space, will be presented in IndoorAir 2014, Hongkong.

Parneix, S. and Durbin, P. (1997) Numerical simulation of 3D turbulent boundary layers using the V2F model. *Annual Research Briefs*, 1997, 135-148.

Peng, S. and Davidson, L. (2000). The Potential of Large Eddy Simulation Techniques for Modeling Indoor Air Flows. Proceedings RoomVent 2000, vol. 1, pp. 295-300.

Shih T.H., Zhu J., and Lumley J. L. (1993) A realizable Reynolds stress algebraic equation model. Technical memo 105993, NASA.

Wood et al (2005) Numerical Methods for Modeling Transient Flow in Distribution Systems, *Journal of American Water Works Association* (AWWA) 97:7.

Wyon, D and Sandberg, M. (1989). Thermal Manikin Prediction of Discomfort due to Displacement Ventilation. *ASHRAE Transactions*, vol. 95, part 1.

Chen, Q., Zhang, Z., Zuo, W. (2007). COMPUTATIONAL FLUID DYNAMICS FOR INDOOR ENVIRONMENT MODELING: PAST, PRESENT, AND FUTURE. The 6th International Conference on Indoor Air Quality, Ventilation & Energy Conservation in Buildings IAQVEC 2007, Oct. 28 - 31 2007, Sendai, Japan.