On the performance of the Stravent ventilation system in an office space

Numerical and experimental investigations

S. Janbakhsh[1, 2] and .B. Moshfegh[1, 2]

[1] Division of Energy and Mechanical Engineering, University of Gävle
S -801 76 Gävle, Sweden

[2] Division of Energy Systems, Linköping University
S -581 83 Linköping, Sweden

Corresponding email: bahram.moshfegh@.liu.se

ABSTRACT

The objective of this study is to predict the air-flow pattern, temperature distribution from a newly developed supply diffuser in an office room. The proposed supply air diffuser can be described as a number of free circular jets issue in a staggered configuration from different apertures at the inlet of the supply device. The RNG k-ε turbulence model with a two-layer wall treatment has been used. The numerical predictions are validated against detailed experimental measurements, using hot-wire and omni-directional thermistor anemometers type CTA. The results of the investigation explore the detail air-flow distribution close to the supply diffuser, along the wetted wall as well as on the floor. Both isothermal and non-isothermal cases have been investigated. It is concluded that the predicted air velocity from the supply diffuser are in good agreement with the experimental values.

INTRODUCTION

During the past two decades, the demand of primary energy for the world energy has been doubled and during the same time the demand for the electrical energy has been tripled. In Sweden, the energy demand in the built environment is a growing issue and the building sector accounts not only for nearly 36% of total energy use but also for 15% of the total CO_2 discharge. In Sweden, the Environmental Advisory Council has stated that the demand for purchased energy in the building sector should decrease at least by 30% in 2025 compared to 2000. As a result, the need for choosing the right energy conservation measures and reduction of electrical energy usage in the built environment is very crucial. Ventilation systems, thermal comfort and air quality within built environment are important issues as they are related to both energy conservation and the health of the occupants. Poor indoor environment conditions, e.g. in offices and classrooms, cost large amounts of money in health-care, users' sustained cognitive functioning, administration and lost productivity, see e.g. *Wargocki and Seppänen* [1]. This stresses the importance of well functioning Heating, Ventilation and Air-Conditioning (HVAC) systems in built environment.

Although the traditional mixing systems have low ventilation effectiveness and air-exchange efficiency, see *Etheridge and Sandberg* [2], and therefore are less energy efficient, they still occupy a large portion of the market. When displacement ventilation was first introduced almost three decades ago, it seemed at the time to be a promising ventilation concept due to its high air-exchange efficiency and ventilation effectiveness. But displacement ventilation is limited to be used only when there is a need of cooling because it is a low momentum supply and therefore warm goes straight upwards. The stratification will be broken down if the mechanical energy imparted to the room exceeds a certain magnitude. Another weakness of the displacement ventilation will be feasible in a multiple buoyancy source environment, where the weaker sources are locked in the occupied zone. To overcome these problems a new ventilation system, i.e. the wall confluent jets ventilation, has been proposed.

The characteristic of confluent jets can be described as a number of free circular jets issue from different apertures of a supply diffuser in the same plane and flow in a parallel direction, then at a certain distance downstream they coalesce and move as a single jet. The behaviour of the wall confluent jets ventilation is such that it produces a slow diffusion due to lower velocity decay, therefore the momentum of the confluent jet can be more conserved, see *Chao et al.* [3].

The aim of the present paper is to investigate the air-flow pattern created from the newly proposed air supply system, based on confluent jets principle, along the wetted wall towards the floor using numerical simulation supported by experimental investigation. Both isothermal and non-isothermal cases have been investigated. The simulation model presented in this study will be used to predict indexes such as PMV, PPD and DR for different summer and winter cases in the near future.

COMPUTATIONAL SET-UP AND NUMERICAL SCHEME

Geometrical set-up and boundary conditions

The computational domain is a well insulated test room with the dimensions 4.2×3.6×2.5 m, with one window as a mock-up of an office room, see Figure 1. The window has dimensions 1.5×1.35 m. The room simulates a realistic office environment, which furnished with a PC, a mannequin and lighting. Both supply and exhaust diffusers are mounted on the wall opposite to the external wall. The differences between ventilation cooling load, internal heat load is compensated by the cooling power provided by a chilled ceiling beam. The chilled beam covers the whole ceiling.

The following boundary conditions are used: the internal and external walls are adiabatic with no-slip conditions. The inlet velocity profile, v, has been provided by a curve fit from measurement carried out at 0.09 m below the supply diffuser, i.e. the box method, see *Nielsen* [4], with an inlet temperature of 16 °C for non-isothermal cases. This temperature is approximately equal to 14 °C at the outlet of the supply diffuser. Pressure outlet is used for the exhaust. The inlet k and ε are estimated by assuming that the turbulence intensity is 10% and the turbulent viscosity ratio is 10.

Governing equations

A steady state three-dimensional model is considered for analyzing the flow and heat transfer in the whole room. The flow is assumed to be turbulent. The thermo-physical properties of the air are assumed to remain constant except for the buoyancy term of the y-momentum equation, i.e. the Boussinesq approximation. The radiation heat transfer is included. The radiative surfaces are assumed to be grey-diffuse with constant emissivity and the participating fluid, i.e. is air, does not interact with the radiative process directly. The radiation model used in the present study is P1 radiation model, which takes into account the heat transfer exchanges between different objects by solving a so-called a radiative transfer equation. The RNG k–ε model is used for the turbulence modeling. Details about the RNG model can be found in *Moshfegh and Nyredy* [5] or *Fluent 6.3.26* [6].

Near-Wall Treatments

In this study the enhanced wall function (EWF) treatment has been used. The EWF subdivides the near wall region into a viscous sub-layer and a fully turbulent flow. In the viscous sub-layer region, only the k equation is solved, while in the fully turbulent flow the RNG model is used.

Numerical details

The finite-volume code Fluent 6.3.26 was used to numerically solve the governing equations with a segregated scheme and the SIMPLE algorithm solved the pressure-velocity coupling. The governing equations are discretized spatially with second-order upwind scheme for non-linear terms and second-order central scheme for viscous terms.

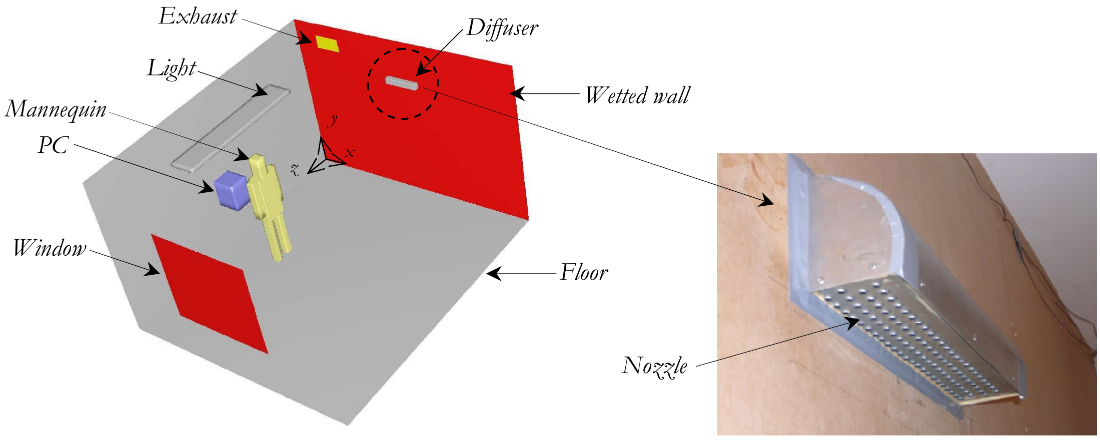

Figure 1. Test room layout (left) and the newly design supply diffuser (right).

The mesh consists of 441 708 and 1 956 206 structured hexahedral cells for isothermal and non-isothermal cases, respectively. There are 30 × 10 cells for the inlet supply. The mesh is refined enough near the solid walls ($y^+ < 1$) to solve the all boundary layer with the two-layer model. A grid independency study was also performed.

EXPERIMENTAL SET-UP

The measurements were carried out in a well insulated test room with the same dimensions and furnished as simulation model, located at the laboratory of ventilation and air quality, University of Gävle. In the measurement part of the study the ventilation of the room was performed by the proposed supply diffuser with an air-flow rate 0.015 m^3/s. The air-flow behavior was analyzed in detail for an isothermal case i.e. the difference between the inlet air temperature and room mean air temperature was kept less than 1 °C.

Experimental procedure

The measurements were carried out under steady-state condition. The smoke visualization was used to explore the flow pattern below the new design supply diffuser, see Figure 2. Detailed air velocity measurements were logged close to supply devices and at different heights and different distances from the wall with one dimension hot-wire anemometer. Hot-wire was placed on a 2–D traversing system close to the diffuser, where the probe can capture the velocity profile with minimum disturbance on the air-flow pattern. The hot-wire measured low air velocity with a sampling rate 100 Hz with 12 000 numbers of sample with accuracy ± 0.03 m/s. Different sampling times were used to analyze the effect of the sampling rate on the arithmetic mean values. The calibration was done between 0.2 - 18.0 m/s with two wind tunnels.

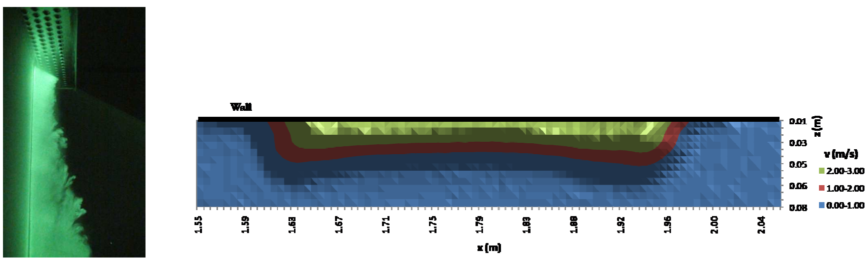

Figure 2. Flow pattern visualization by smoke (left) and the velocity profile 0.09 m below the diffuser used as inlet boundary conditions in the numerical simulation (right).

Both hot-wire and omni-directional thermistor anemometers type CTA, see *Lundström et al.* [7], were used to measure the velocity at different distances from the floor and along the length of the room. The velocity measurement with CTA carried out at two seconds interval and the total number of samples of 150. The CTA anemometers used in this study can perform the low velocity measurement with accuracy ±5% or 0.05 m/s. Figure 2 shows also the velocity profile 0.09 m below the diffuser which is used as inlet boundary conditions in the numerical simulation.

RESULT

In this section, experimental results will be compared with numerical predictions for validation of the simulation model. Then, the numerical predictions will focus on the effect of the supply air flow rate on the spreading behavior of the jet and finally three non-isothermal cases with different supply air flow rates will be presented. Case 2 is the validation case for this study, where the experimental results have been carried out. Table 1 summarizes the cases investigated in this study.

Table 1. Set-up boundary conditions for the numerical model.

Cases	q_{total} (W/m²)	$Q_{mannequin, PC, lighting}$ (W)	Q_{floor} (W)	Q_{window} (W)	$\dot{V}$ (m³/s)	T_{in} (°C)	$Q_{chilled\ beam}$ (W)
1	---	---	---	---	0.010	---	---
2	---	---	---	---	0.015	---	---
3	---	---	---	---	0.020	---	---
4	23	95, 115, 144	---	---	0.010	16	293
5	23	95, 115, 144	---	---	0.015	16	262
6	23	95, 115, 144	---	---	0.020	16	232

Figure 3 shows the stream-wise velocity, *v*, profiles at *x*=1.8 m and three different heights from the floor for Case 1-3. The experimental results with the same operating conditions as Case 2 are also included in Figure 3. As it is shown, the maximum velocity of the wall jet decreases by decreasing the height and the jet becomes wider due to mixing with the room air. The predicted velocity profiles are in good agreement with the measurement especially for the height close to supply. Higher mixing rate with the room air has been observed in the measurement compared to predicted values at the other heights.

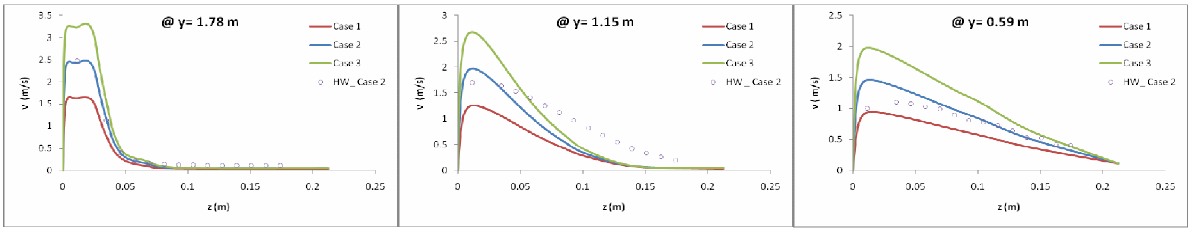

Figure 3. Velocity, *v*, at different heights from the floor.

Figure 4 shows the air velocity in *z* direction, *w*, profiles at *x*=1.8 m and three different downstream locations in the room. As it is shown rather good agreement has been observed between the experimental measurement performed by the hot-wire anemometry and the predicted results. It is also interesting to mention that after 3 m the maximum velocity is still close to 0.2 m/s for Case 2. Thus, it reveals that the new supply device has low velocity decay due to quite slow diffusion and the momentum of the jet has been more conserved.

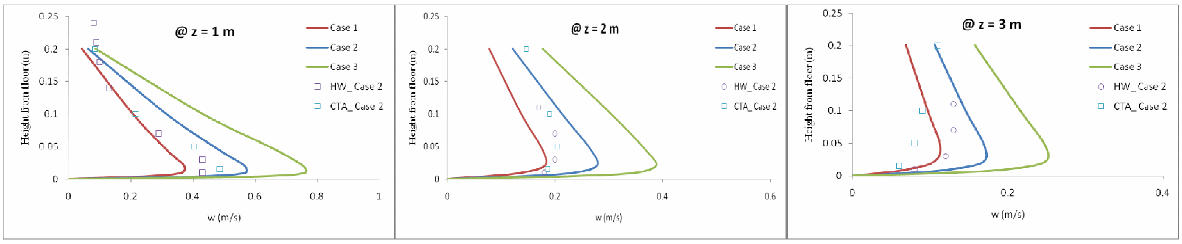

Figure 4. Velocity, *w*, at different downstream locations in the room.

Figure 5 shows the air velocity in *z* direction, *w*, and the velocity magnitude, *U*, profiles at *x*=1.8 m and three different heights above the floor. Very good agreement has been observed between the experimental measurement performed by the hot-wire and CTA anemometers and the numerical results. It is worth to mention that the generated *y*+ in the numerical mesh capture quite well the wall effect. The high velocity observed at *y* = 0.2 m is closed to the wetted wall.

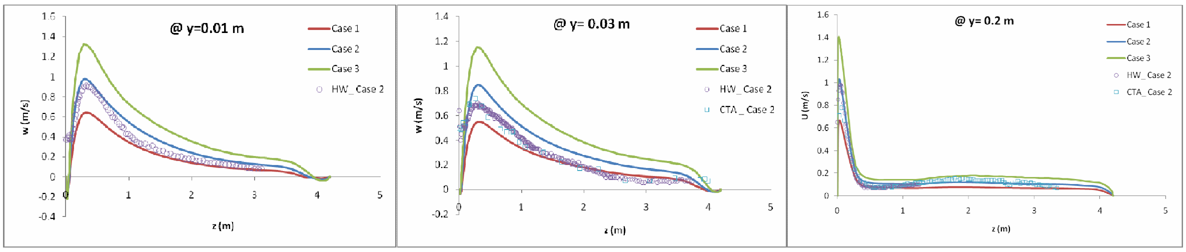

Figure 5. Velocity, *w and U*, at different heights above the floor.

Figure 6 shows different iso-velocities for Case 2. Due to rather high momentum conservation, the jet spreads over the floor, covers nearly the whole floor and reaches to opposite wall. Figure 6 shows also the near zone for the proposed supply diffuser, which is reasonably small for a stratified ventilation systems.

Iso-vel = 0.1 m/s, Iso-vel = 0.2 m/s, Iso-vel = 0.4 m/s, Iso-vel = 1 m/s,

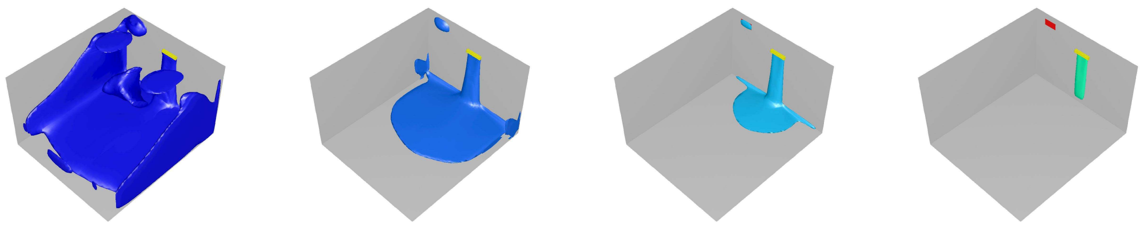

Figure 6. The iso-velocity for Case 2.

Figure 7 shows the velocity and temperature profiles in the middle of the room for Case 4-6. The stratification effect of the new supply diffuser is clearly presented in the Figure 7. Supply air with lower temperature than the room air and with a low velocity is supplied directly into the occupied zone. The supply air rises as its temperature increases along the room's height, thus creating temperature and density stratification. As it is shown, the air velocity is rather low in the occupied zone due to the fact the air does not meet any heat sources in this part of the room.

Figure 8 shows the air velocity in *z* direction, *w*, profiles at *x*=1.8 m and three different downstream locations in the room for Case 2 and 5. As it is shown, the predicted velocity profiles are quite similar for the isothermal and non-isothermal cases and are in relatively good agreement with experimental results. The same characteristics have been also observed along the wetted wall. One can conclude that the internal heat loads have negligible effect on the spreading of the jet due to quite slow diffusion and high momentum conservation.

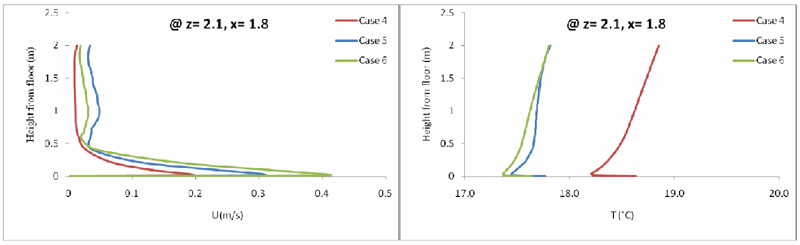

Figure 7. The velocity and temperature profiles in the middle of the room for Case 4-6.

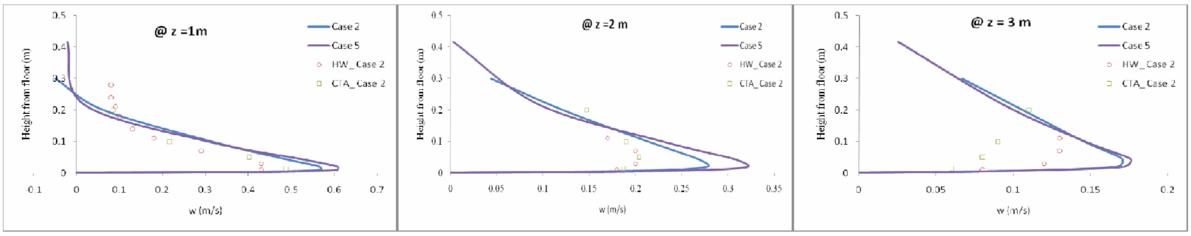

Figure 8. Comparison of the velocity profile, *w*, at different downstream locations in the room between Case 2 and 5.

CONCLUSION

The results of the study have shown that the predicted air velocity from the supply diffuser along the wetted wall and the floor are in good agreement with the experimental values. The employed $y+$ captured the wall effect properly. The results also confirmed that the new supply device has low velocity decay due to quite slow diffusion and the momentum of the jet has been more conserved. The stratification effect of the new supply diffuser is clearly observed in the room. The predicted velocity profiles from the supply diffuser along the wetted wall and the floor are quite similar for the isothermal and non-isothermal cases.

ACKNOWLEDGMENT

The authors would like to acknowledge the financial support from University of Gävle (HiG) and the Stravent AB and the technical support during experimental work by Hans Lundström at HiG.

REFERENCES

1. Wargocki P. and Seppänen O., Indoor climate and productivity in offices-how to integrate productivity in life – cycle cost analysis of building services. REHVA Handbook No: 6 (2007).
2. Etheridge, D. and Sandberg, M., Building Ventilation: Theory and Measurement. *John Wiley & Son Ltd*, ISBN 0 471 96087 X (1996).
3. Cho Y-J., Awbi H. B. and Karimipanah T., Theoretical and experimental investigation of wall confluent jets ventilation and comparison with wall displacement ventilation. *Journal of Building and Environment.*
4. Nielsen P. V., The box method – a practical procedure for introduction of an air supply devices in CFD calculation. Institute for Bygningsteknik Aalborg University, 1997.
5. Moshfegh B. and Nyiredy R., Comparing RANS models for flow and thermal analysis of pin fin heat sink, In Proc. *of 15th Australasian Fluid Mechanics Conference,* Sydney, Australia (2004).
6. Fluent 6.3.26, 2006, Fluent Manuals, Fluent Inc.
7. Lundström H., Blomqvist C., Jonsson P. and Pettersson I. A microprocessor- based anemometer for low air velocities, the National Swedish Institute for Building Research Gavle, Sweden (1990).