Modelling of Active Displacement Air Distribution in Heating and Cooling Conditions – A Case Study

Koskela H,
Finnish Institute of Occupational Health, Turku, Finland

Introduction

The performance of supply air distribution is highly dependent on operating conditions, such as room geometry, temperature differences and other flow elements including plumes from heat sources and flows along the wall surfaces. Therefore, predicting the flow patterns or temperature and contaminant distribution in the space is a demanding task. CFD-modelling provides a means of predicting the flow pattern and of understanding the interaction between the supply air and other flow elements. However, detailed modelling of actual premises often requires too much computer resources. Therefore, simplifications of the room geometry and, especially, the air distribution devices are necessary in practical CFD-simulations.

In this study, CFD-modelling was applied to analysing the performance of Active displacement air distribution in the IKEA furniture store in different operating conditions. The air distribution of the store was designed basically for cooling conditions, but reasonable performance was expected also in heating conditions during cold mornings. The main aim of the study was to analyse the air flow pattern and temperature stratification in heating conditions. Active displacement is a relatively new low-impulse method, which is typically used in large spaces (1). The supply air is distributed with nozzle ducts, which are placed above the occupied zone (Figure 1). Air velocity in the nozzles is high, but it is quickly reduced due to circulation between the jets. The supply air mixes with the room air in the near zone of the duct. The air flow pattern and the momentum supplied by the nozzle duct depend on the location of the nozzles in the duct. The device in this study was Activent nozzle duct with a nozzle sector of 240° upwards. This type of nozzle duct device is typically used for cooling applications. It creates a flow pattern that is a combination of mixing and displacement. The unit takes secondary air from below and the supplied momentum is high enough to prevent the direct downfall of the mixed subtempered air.

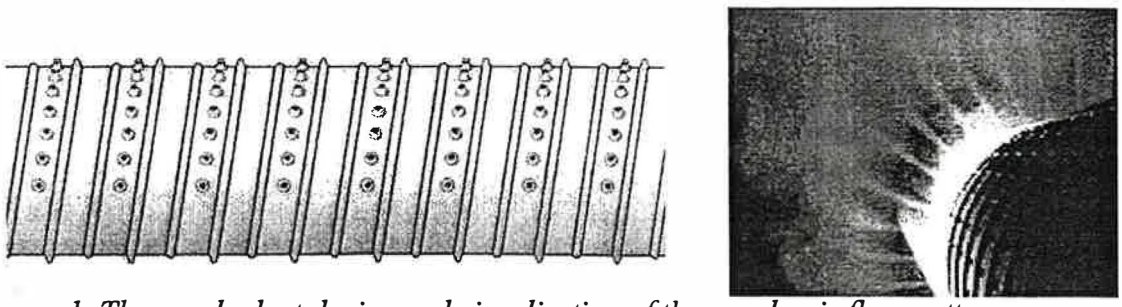

Figure 1. The nozzle duct device and visualisation of the supply air flow pattern.

Methods

A simplified model was built consisting of the hall module served by one nozzle duct unit (Figure 2). The air flow rate of one unit is 0.165 m^3/s, which gives a ventilation rate of 5.2 $l/s \cdot m^2$ or 4.6 air changes per hour. The nozzle duct (diameter 0.25 m, length 5 m) is located close to the ceiling at the height of 3.25 m. The outlet is at the same height and in the model it is placed on the outer wall. The modelled volume is located in the middle of the hall and symmetry boundary conditions are therefore used in three directions. The hall is situated on the upper floor and therefore most of the thermal leakage takes place through the ceiling (U-value 0.3 $W/m^2 \cdot K$) and a smaller part through the walls and floor. The main internal heat source is the lighting (12 W/m^2). 75 % of the heat power was placed to the ceiling and 25 % to the floor. The thermal load of people was estimated to be relatively small, 2–4 W/m^2.

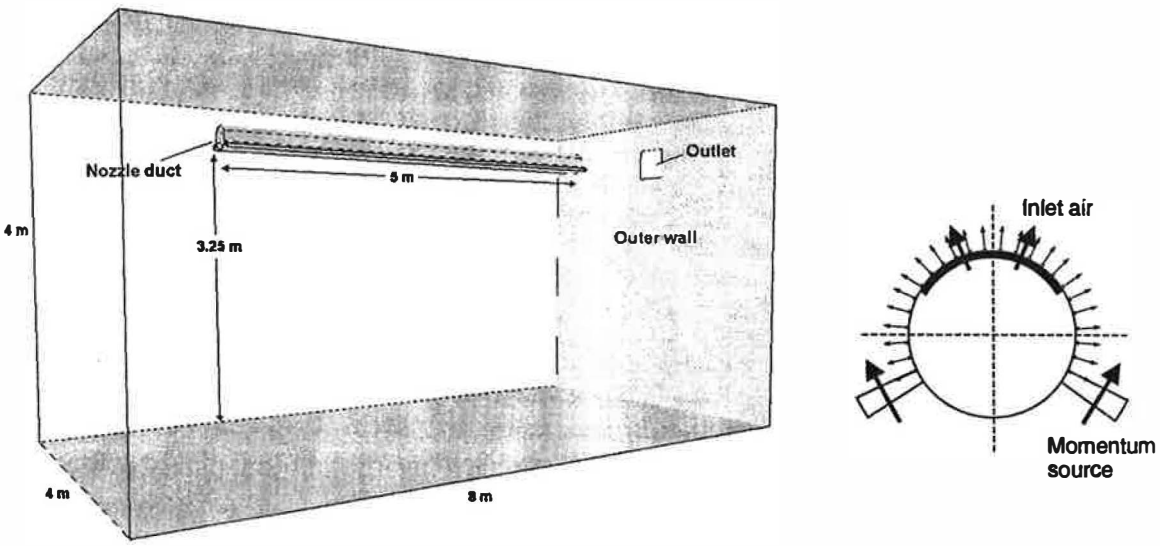

Figure 2. Simplified models for the room module and the nozzle duct.

The CFD-simulations were carried out with an unstructured code, CFX 5.3, by using k-ε turbulence model and Boussinesq approximation for buoyancy. The density of the computational grid was 0.2 m, which was refined near the nozzle duct to 0.05 m. The simplified model for the nozzle duct (2) is shown in Figure 2. The inlet air is blown in the model with low momentum from the upper surface of the unit. The momentum flow of the device is modelled by two momentum sources on the sides of the duct. The total magnitude of the momentum sources was set as 0.26 N.

Results

The CFD-simulation was carried out in three operating conditions as shown in Table 1.

Table 1. Boundary conditions of the simulated cases.

	Heating or cooling (W/m^2)	Ventilation	Heat sources Lighs (W)	Heat sources People (W)	Thermal leakage Ceiling (W)	Thermal leakage Wall (W)	Thermal leakage Floor (W)	Outdoor temp. in steady state (°C)	in
Heating	21	100 %			-380	-100	-190	-20	
Normal	-8	50 %	380	65	-145	-75		+5	
Cooling	-21	100 %	380	130	+160			+25, sun	

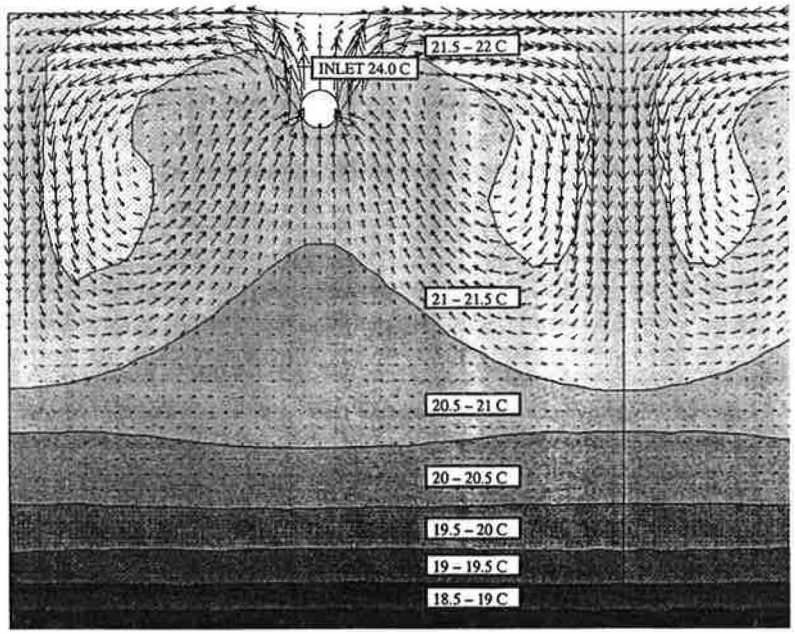

Figure 3. Flow pattern and temperature distribution in heating conditions.

The flow pattern and temperature distribution in heating conditions in a vertical plane in the middle of the module is shown in Figure 3. The warm inlet air is blown along the cool ceiling. In the symmetry plane, the inlet flows from two devices meet and the flow is directed downwards. The eddies formed near the symmetry plane cause mixing of the room air and prevent the formation of strong temperature stratification in the room. The simulated flow pattern was confirmed by smoke tests and measurements, which were carried out in the actual furniture hall in heating conditions. The measured distributions of air temperature and speed are compared with the simulation results in Figure 4.

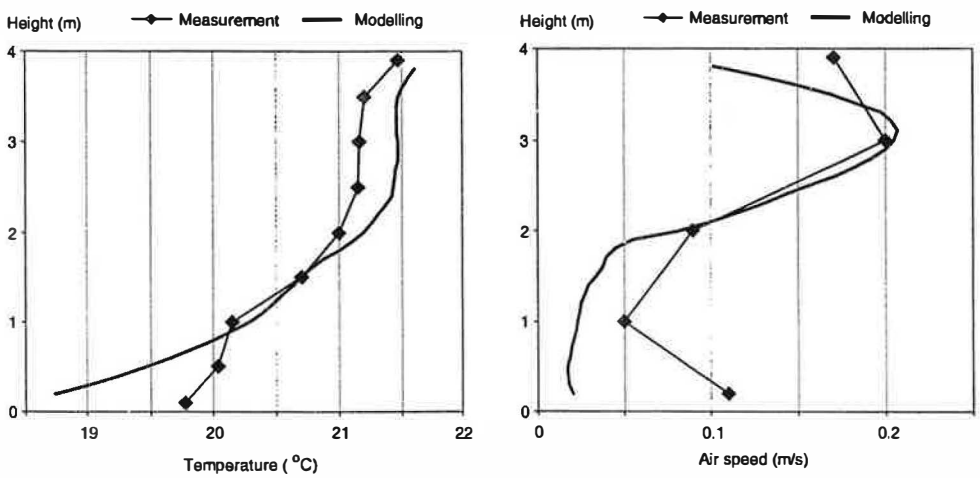

Figure 4. Vertical distribution of air temperature and speed in heating conditions.

The temperature stratification and air velocity in the simulated cases are shown in Table 2. Air speed was calculated from air velocity and turbulence intensity results (3).

Table 2. Calculated mean values of temperature gradient and air speed.

	Temperature gradient (0.5m - 3.5m) (°C)	Air speed at 1.8 m mean (m/s)	max (m/s)
Heating	2.0	0.06	0.13
Normal	0.1	0.08	0.19
Cooling	0.1	0.10	0.28

Discussion

According to the CFD-simulations, the performance of the Active displacement air distribution was found to be adequate also in heating conditions in this case. The positioning of the inlet devices near the ceiling, where also the main part of thermal leakage took place, led to sufficient mixing of the room air. This result can, however, not be applied to e.g. higher enclosures, because the flow pattern is strongly dependent on the geometry and other conditions. The modelled room module was greatly simplified and had no obstacles on the floor. Differences can therefore be expected between the simulated and real air flow pattern and air speed values.

References

1. Sandberg E, Koskela H and Hautalampi T. Convective Flows and Vertical Temperature Gradient with the Active Displacement Air Distribution, Proceedings of ROOMVENT '98. Stockholm, Sweden; Vol 1:61-68.
2. Koskela H. CFD-Modelling of Active Displacement Air Distribution. Submitted to Proceedings of ROOMVENT 2000. Reading, UK.
3. Koskela H, Heikkinen J, Niemelä R and Hautalampi T. Turbulence Correction for Thermal Comfort Calculation. To be published in Building and Environment.