

# INFLUENCE OF A COOLED CEILING ON INDOOR AIR QUALITY IN A DISPLACEMENT VENTILATED ROOM EXAMINED BY MEANS OF COMPUTATIONAL FLUID DYNAMICS

Henrik Brohus

Dept. of Building Technology and Structural Engineering, Aalborg University, Denmark  
Fax +45 9814 8243 E-mail hb@civil.auc.dk

## ABSTRACT

The influence of a cooled ceiling on the air quality in a displacement ventilated room is examined by means of CFD. The objective of the study is to examine how the flow field in a displacement ventilated room is influenced when a cooled ceiling removes a major part of the total heat load, and in particular to examine the effect on the contaminant distribution and the indoor air quality.

The simulations show that the inclusion of a cooled ceiling has a significant impact on the flow field but only a minor influence on the personal exposure in this study.

## KEYWORDS

Displacement Ventilation, Cooled Ceiling, Air Quality, CFD, Breathing Zone.

## INTRODUCTION

During the last two decades the displacement principle has gained increasing popularity for ventilation of non-industrial buildings like offices, assembly halls, and educational facilities, especially in Germany and the Scandinavian countries.

The main reason for applying the displacement principle is the possibility of removing excess heat in an energy efficient way and at the same time obtaining a high ventilation effectiveness (Mundt, 1996).

When the heat load is high, i.e. above 40 - 50 W/m<sup>2</sup>, the vertical temperature gradient will usually exceed the limit of thermal comfort and the adjacent zone close

to the inlet device may occupy a significant part of the floor area. In this case displacement ventilation can be combined with a cooled ceiling which may extend the range of application up to a heat load of approximately 100 W/m<sup>2</sup> (Fitzner, 1996).

When the cooled ceiling is applied the room air in close contact with the ceiling will be cooled and an unstable layer of cold air in the upper part of the room is created. At the same time radiation heat transfer will cool the surrounding surfaces and thus cause a descending air flow along the walls. Both effects of the cooled ceiling cause an increased mixing of the room air.

The purpose of the work presented in this paper is to perform a numerical case study of the influence of a cooled ceiling on the indoor air quality in a displacement ventilated room.

## METHODS

The geometry of the displacement ventilated room used in the numerical case study is shown in Figure 1. Two different cases are defined in Table 1.

Table 1. Definition of test cases.

CSP = Computer Simulated Person

| Parameter                          | Case I | Case II |
|------------------------------------|--------|---------|
| Air change rate (h <sup>-1</sup> ) | 0.8    | 2.1     |
| Person simulator (W)               | 50     | 50      |
| Point heat source (W)              | 225    | 675     |
| CSP (W)                            | 20.25  | 20.25   |
| Total convective load              | 295.25 | 745.25  |
| Load per area (W/m <sup>2</sup> )  | 12     | 31      |

height of 1.7 m and a surface area of 1.62 m<sup>2</sup>. The heat transfer boundary condition is a convective heat flux of 25 W/m<sup>2</sup> which corresponds to an activity level of a person standing relaxed. The personal exposure of the CSP is simulated by means of the contaminant concentration in the nearest cell along the body in a height of 1.5 m (Brohus and Nielsen, 1996).

Two different contaminant sources are applied. First, a warm point contaminant source simulated by a small transparent volume source located above the point heat source in a height of 2 m above the floor. Secondly, a passive planar source in shape of constant emission from the floor is applied.

The steady-state, three-dimensional, non-isothermal flow field is simulated by means of a numerical solution of the continuity equation, the Navier-Stokes equations and the energy equation.

turbulence model. Standard wall functions are applied on all surfaces (Brohus, 1997). For discretisation of the flow domain the Finite Volume Method and the SIMPLE algorithm are applied (Patankar, 1980).

### Simulation of a cooled ceiling

A cooled ceiling may be implemented in the CFD simulations in different ways, e.g. by means of: 1. Full scale measurements (prescribed heat flux or surface temperatures in CFD), 2. Conjugate heat transfer including radiation modelling, and 3. CFD combined with building dynamic simulation.

The three suggested ways of simulating a cooled ceiling are relatively demanding and cumbersome. In this study another and more simple approach is used in order to get an overview of the influence of a cooled ceiling on the flow field and the indoor air quality.

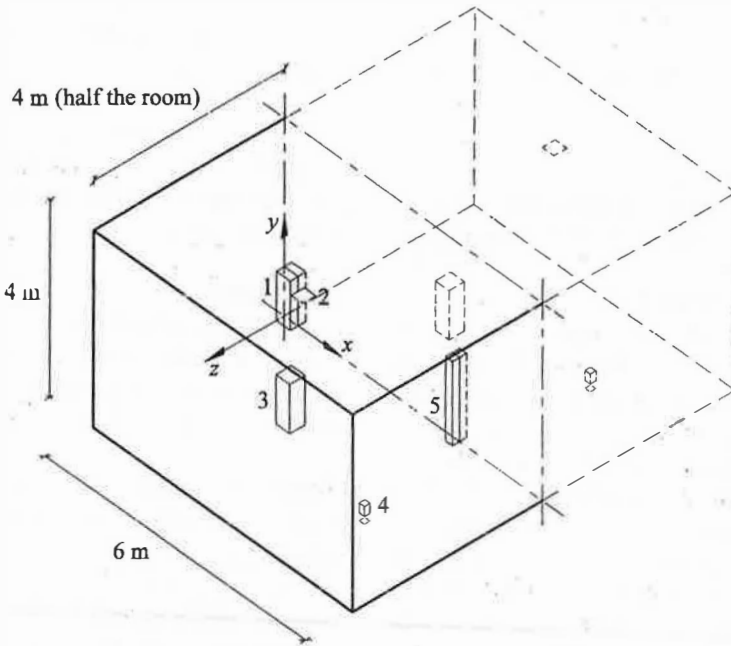


Figure 1. Geometry of the CFD simulated displacement ventilated room. Only one half of the symmetric room is simulated. The subcooled air is supplied through the inlet device (1) and exhausted through openings in the ceiling (2). The heat load is generated by person simulators (3), a point heat source (4), and the Computer Simulated Person (5).

A conceptual model for the overall influence of a cooled ceiling is illustrated in Figure 2.

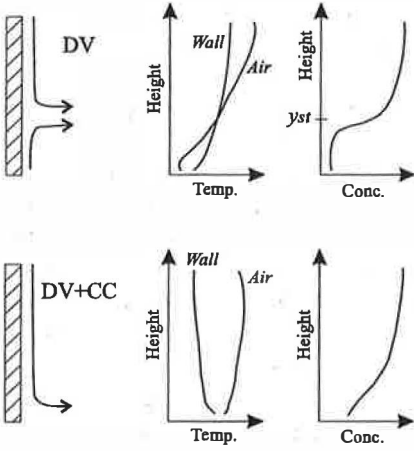


Figure 2. Conceptual model for the overall influence of a cooled ceiling (CC) on the flow field in a displacement ventilated room (DV).  $y_{st}$  is the stratification height.

Usually, in a displacement ventilated room the stratified flow field in the room is separated in a lower, cleaner part and an upper, more contaminated part. A vertical temperature gradient prevails on the surfaces as well as in the room air where the slope of the air temperature exceeds the slope of the surface temperature. The two temperature profiles cross around the stratification height,  $y_{st}$ , and cause an ascending air current below  $y_{st}$  and a descending air current above  $y_{st}$ . When a cooled ceiling is applied the radiation heat transfer will cool the surrounding surfaces and a new vertical temperature distribution is created, see Figure 2. Here, the surface temperature is below the room air temperature for the entire height.

In order to cover the effects of the cooled ceiling in a simple way in the CFD simulations it is assumed that the convective part of the heat removed by the ceiling can be prescribed as an outward heat flux at the ceiling. The radiant heat removal is distributed to the surrounding surfaces

according to the radiation shape factors and then prescribed as an outward heat flux at the respective surfaces.

If the relative cooling capacity of the cooled ceiling is termed  $\omega_c$ , we get

$$\Phi_{Heat} = \Phi_{Vent} + \Phi_{CC} \quad (1)$$

$$\omega_c = \frac{\Phi_{CC}}{\Phi_{Heat}} \quad (2)$$

where

$\Phi_{Heat}$  = Total convective heat load (W)

$\Phi_{Vent}$  = Heat load removed by the ventilation system (W)

$\Phi_{CC}$  = Heat load removed by the cooled ceiling (W)

$\omega_c$  = Relative cooling capacity of the cooled ceiling (n.d.)

The heat load removed by the ceiling is separated in a convective part,  $\Phi_{CC,Con}$ , and a radiative part,  $\Phi_{CC,Rad}$ . If it is assumed that each part contribute with 50%, we have

$$\Phi_{CC,Con} = \Phi_{CC,Rad} = 0.5 \cdot \Phi_{CC} \quad (3)$$

As mentioned previously the radiative part is distributed to the surfaces according to the shape factor between the cooled ceiling and the actual surface, i.e. for a surface  $i$

$$\Phi_i = F_{CC-i} \Phi_{CC,Rad} \quad (4)$$

$$\sum F_{CC-i} = 1 \quad \text{and} \quad \sum \Phi_i = \Phi_{CC,Rad} \quad (5)$$

where

$\Phi_i$  = Outward heat flux prescribed at surface  $i$  (W)

$\Phi_{CC,Rad}$  = Heat load removed by CC by means of radiation (W)

$F_{CC-i}$  = Radiation shape factor between CC and surface  $i$  (n.d.)

reads

$$\Phi_{Heat} = (1 - \omega_c) \Phi_{vent} + \quad (6)$$

$$\omega_c \cdot 0.5 \cdot \Phi_{CC} + \omega_c \cdot 0.5 \cdot \sum F_{CC-i} \Phi_{CC}$$

where the second term on the right side is prescribed as an outward heat flux at the ceiling and the last term is prescribed as outward heat fluxes at the respective surrounding surfaces. The heat load removed by the ventilation system is easily found when the temperature difference between the supply air and the exhaust air is known together with the air flow rate.

The radiation boundary condition in the simulations is prescribed as mentioned above when the cooled ceiling is applied, while the case where  $\omega_c = 0$  is based on full-scale measurement (Brohus, 1997). Here, the heat flux is prescribed at the walls (one value below  $y_{st}$  and another value above  $y_{st}$ ) and surface temperatures are prescribed for the floor and the ceiling.

For the simulations including the cooled ceiling the supply air temperature is chosen in order to obtain a return air temperature of 25°C.

## RESULTS

In Figure 3 the flow field in Case I is presented for the relative cooling capacity of the cooled ceiling  $\omega_c = 0$  and  $\omega_c = 1$ , respectively.

Figures 4 and 5 show the dimensionless contaminant concentration distribution in the symmetry plane for the flow fields shown in Figure 3.

The personal exposure of the CSP is summarised in Figure 6 for two different orientations relative to the inlet device. Here, the exposure versus relative cooling capacity of the cooled ceiling,  $\omega_c$ , is shown for Case I and Case II.

The computational grid comprises 82,764 control volumes for the results

independence a finer grid with 261,000 control volumes has been applied and no significant differences due to grid size was found.

## DISCUSSION

The flow field in Figure 3 for  $\omega_c = 0$  shows a typical displacement ventilation case where the subcooled air enters the room and spreads out radially along the floor due to buoyancy. The main flow direction is horizontal due to the stratification, only close to sources of momentum like the CSP and the walls significant vertical flow is found.

In case of a cooled ceiling with  $\omega_c = 1$  we see a different flow pattern. Now, the supply air is almost isothermal and the local flow looks like a typical wall jet assisted by the descending convection currents created along the walls. Due to the removal of heat at the surfaces an increased mixing of the room air takes place and the dominant flow direction is not horizontal anymore. The general velocity level is significantly increased which corresponds well with the measurements reported by Fitzner (1996).

If the contaminant distribution in Figures 4 and 5 are observed significant influence of the cooled ceiling on the contaminant transport in the displacement ventilated room is found. The supply air flow, ranging from a strong buoyant flow to almost isothermal conditions, is also found to influence the air quality experienced by the CSP in the present work.

In Figure 5 the floor is the source of contamination. Here, the results show how the unpolluted air from the inlet device is gradually more and more polluted during the flow throughout the floor. When  $\omega_c = 1$  the contaminant transport is highly influenced by the descending currents along the walls.

If the concentration distribution close to the breathing zone is examined a significant influence of the orientation of the person is found. This fact is caused by the combined influence of the entrainment and

transport of air along the CSP and the effect of the CSP acting as an obstacle to the flow field.

The personal exposure of the CSP is presented in Figure 6. It is found that the dimensionless exposure ranges between 0.6 - 0.8 in case of the warm point contaminant source, while in case of the passive planar source the exposure is close to 1.

The effect of  $\omega_c$  in this numerical case study is relatively weak. Depending on the CSP orientation  $\omega_c$  may both cause an improved as well as a deteriorated indoor air quality.

Measurements by Fitzner and Krühne (1995) show a significant effect at  $\omega_c \sim 0.6$  where the stratified displacement ventilation flow changes into mixing flow. One reason for the discrepancy between the measurements and the present numerical case study may be that the CFD simulation encounters some difficulties in the creation of the significant concentration step-profile in case of  $\omega_c = 0$ , see Figure 2. Another reason for differences may be due to local flow phenomena in the numerical simulations. For instance, the flow from the inlet device which is found to affect the local concentration distribution around the CSP highly.

The influence of the cooled ceiling is modelled in a simple and relatively coarse way in this study. However, the simplified approach represent an easy way to perform a preliminary examination of the overall influence of a cooled ceiling in an early phase of the design procedure.

## REFERENCES

Brohus, H. (1997) *Personal Exposure to Contaminant Sources in Ventilated Rooms*, Ph.D.-thesis, ISSN 0902-7953 R9741, Aalborg University, Dept. of Building Technology and Structural Engineering, Aalborg, Denmark.

Brohus, H. and Nielsen, P. V. (1996) CFD Models of Persons Evaluated by Full-Scale Wind Channel Experiments, *Proceedings of Roomvent '96*, Vol.2, pp.137 - 144, Yokohama, Japan,.

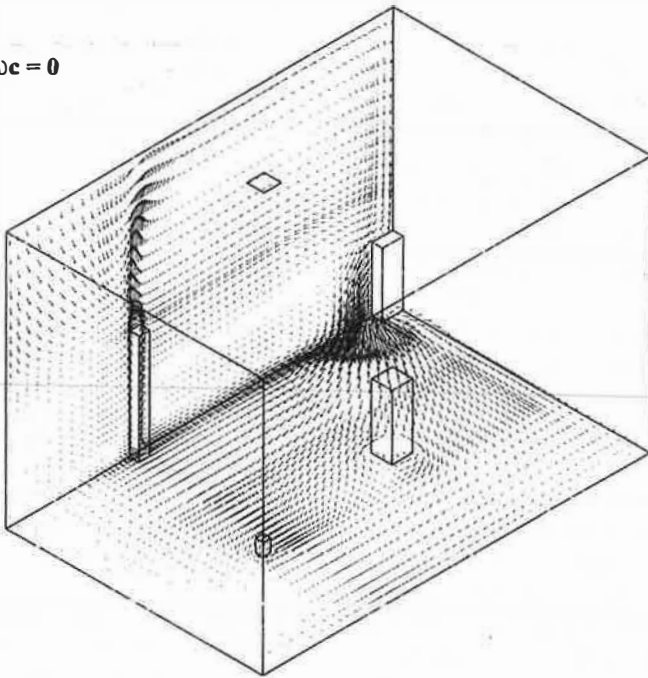
Fitzner, K. (1996) Displacement Ventilation and Cooled Ceilings, Results of Laboratory Tests and Practical Installations, *Proceedings of Indoor Air '96*, Vol. 1, pp. 41-50, Nagoya, Japan.

Fitzner, K. and Krühne, H. (1995) Displacement Flow and Cooled Ceiling, *Proceedings of Healthy Buildings '95*, Vol. 3, pp. 1187 - 1193, Milan, Italy.

Mundt, E. (1996) *The Performance of Displacement Ventilation Systems - Experimental and Theoretical Studies*, Ph.D.-thesis, ISSN 0284 - 141X, ISRN KTH/TT/M--38--SE, Bulletin no 38, Building Services Engineering, Royal Institute of Technology, Stockholm, Sweden.

Patankar, S.V. (1980), *Numerical Heat Transfer and Fluid Flow*, ISBN 0-89116-522-3, Hemisphere Publishing Corporation.

**Case I,  $\omega_c = 0$**



**Case I,  $\omega_c = 1$**

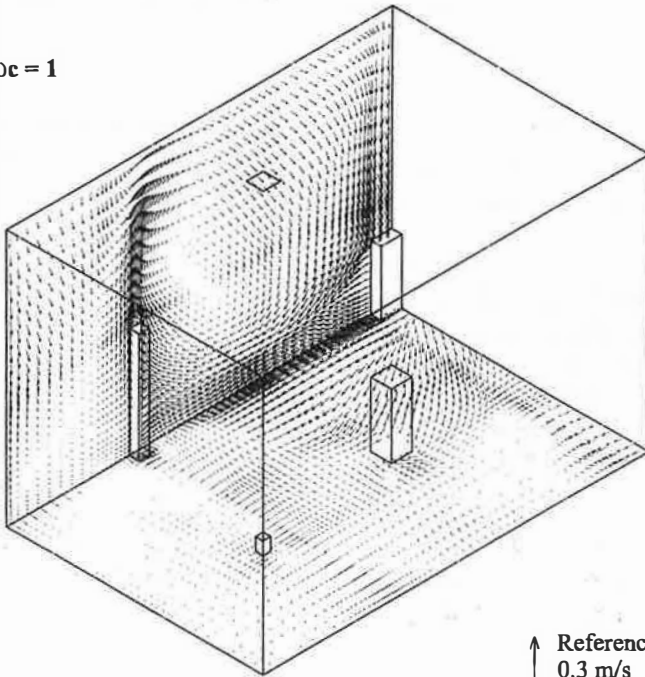


Figure 3. Vector plots of flow field in Case I for a relative cooling capacity of the cooled ceiling  $\omega_c = 0$  (top) and  $\omega_c = 1$  (bottom). Results from the symmetry plane ( $z = 0$  m) and a horizontal plane 0.1 m above the floor ( $y = 0.1$  m) are shown.

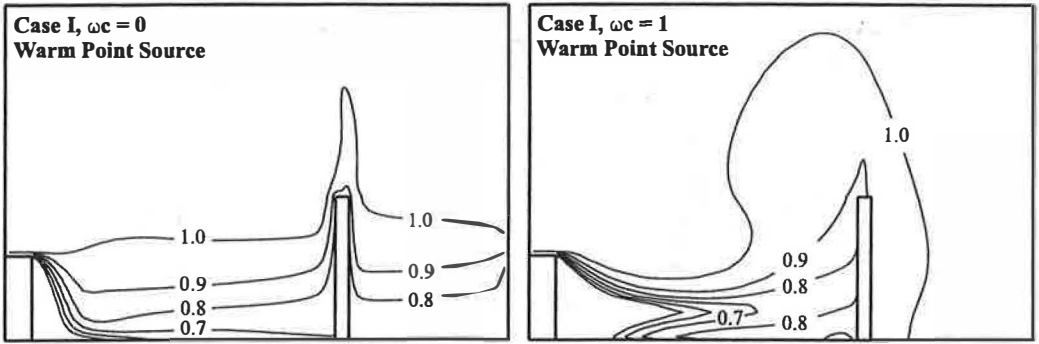


Figure 4. Dimensionless contaminant concentration distribution in the symmetry plane in Case I for  $\omega_c = 0$  (left) and  $\omega_c = 1$  (right). Warm point contaminant source. The concentrations are made dimensionless by dividing by the return concentration.

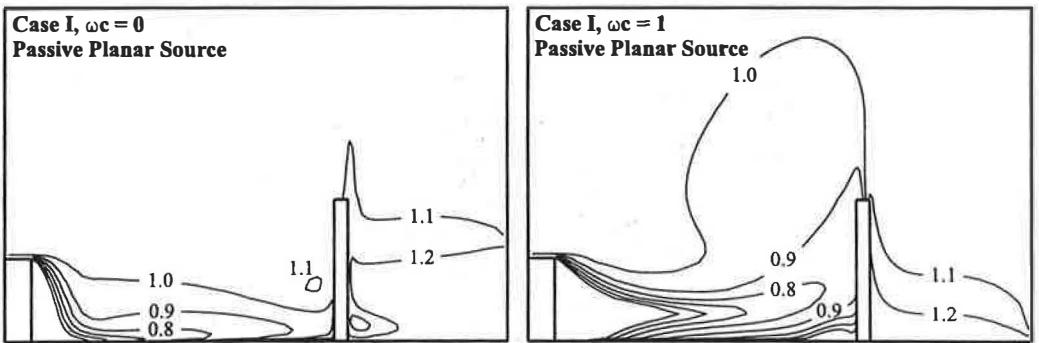
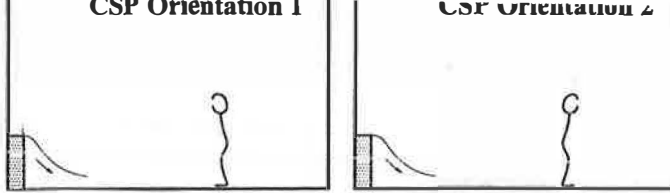
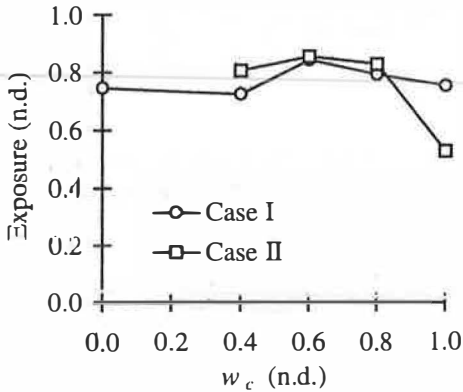


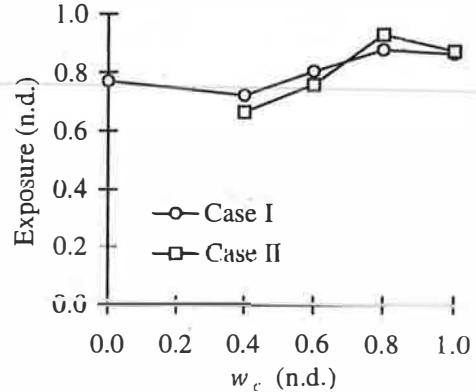
Figure 5. Dimensionless contaminant concentration distribution in the symmetry plane in Case I for  $\omega_c = 0$  (left) and  $\omega_c = 1$  (right). Passive planar contaminant source. The concentrations are made dimensionless by dividing by the return concentration.



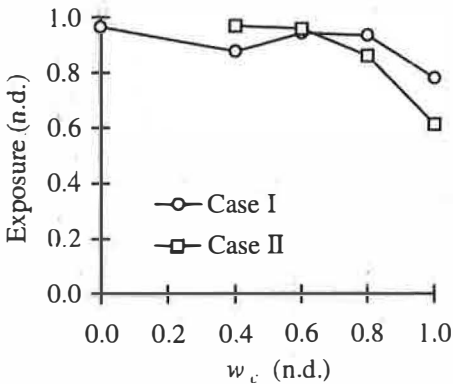
**Warm Point Source  
CSP Orientation 1**



**Warm Point Source  
CSP Orientation 2**



**Passive Planar Source  
CSP Orientation 1**



**Passive Planar Source  
CSP Orientation 2**

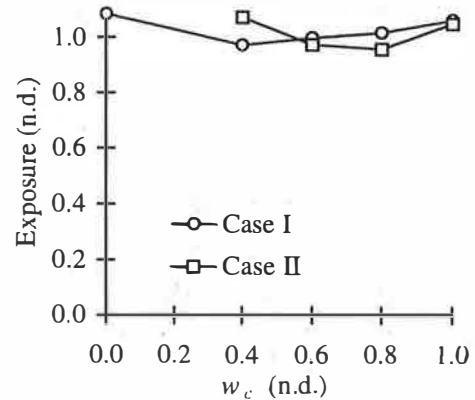


Figure 6. Dimensionless personal exposure of the Computer Simulated Person (CSP) versus relative cooling capacity of the cooled ceiling,  $w_c$ , in case of the warm point source (top) and the passive planar source (bottom). The personal exposure corresponds to the contaminant concentration in the nearest cell along the CSP in a height of 1.5 m. The concentrations are made dimensionless by dividing by the return concentration.