11509

MEASUREMENTS AT AND SIMULATIONS OF THE (IMPROVED) DESK DISPLACEMENT VENTILATION CONCEPT

M.G.L.C. Loomans

Faculty of Architecture, Building and Planning,
Eindhoven University of Technology, The Netherlands

ABSTRACT

Steady state and transient Computational Fluid Dynamics (CFD)-simulations of a climate chamber configuration have been compared with similar full scale measurement results. The configuration comprises a task conditioning concept that applies the displacement ventilation principle. Earlier studies have indicated that this concept is not feasible for standard office configurations. CFD is applied for the investigation of an improved version of the concept. An example of such an improvement is given.

For the steady state situation different boundary and cooling load conditions have been investigated. The presented results indicate the importance of a correct modelling of the wall heat transfer for this type of flow problem. Currently the applied CFD program does not allow a practical improvement of the results, e.g., via an imposed heat transfer coefficient.

Finally, a transient measurement and CFD-simulation result are compared. Given this comparison, an improved version is evaluated numerically.

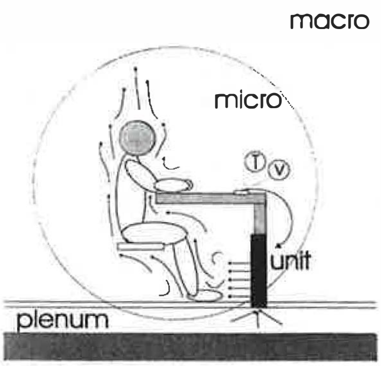

Figure 1 DDV-concept

KEYWORDS

full-scale experiments, displacement ventilation, CFD, demand control

INTRODUCTION

Computational Fluid Dynamics (CFD) is used widely for the simulation of the indoor airflow field, despite its known limitations in the modelling of the non-isothermal, low turbulent indoor airflow. The validation of CFD-results for this type of flow remains restricted as the measurement of a complete indoor airflow field is difficult and the number of available measurement data limited. Nevertheless, the application of CFD for the design of a new ventilation concept is attractive due to often arising experimental restrictions and the time involved. This paper describes the attempt to apply a combined experimental and CFD approach for the improvement of a ventilation concept, the desk displacement ventilation (DDV-) concept (Loomans et al. 1996).

The DDV-concept combines the principles of displacement ventilation and task conditioning. It introduces the supply air below the desk (Figure 1), applying the rules set by the displacement ventilation principle: introduction of air over a relatively large area at low impulse.

The applicability of the concept has been discussed from preliminary steady state measurement and CFD-simulation results and from an additional transient study (Loomans and Rutten (1997) and Loomans et al. (1998)). The results indicate that the DDV-concept is not able to operate as a task conditioning system. Close to the occupant, the transient characteristics of the system, the response time and the damping, do not differ significantly from those of a normally operated displacement ventilation system.

The response time is defined as the time interval between the moment of change of the setting and the point at which $1\text{-}e^{-1}$ of the end-value is reached. The damping is determined from the resulting effect of a change at the position of interest, relative to the set-point change at the inlet. Indicating this effect by f, the damping then is defined as $(1\text{-}f)\times100\%$.

The time response as well as the damping close to the occupant were found to have a too high value; the response time is distinctly larger than one minute and the damping is comparable to the average room values. A time response of less than one minute and a damping that is significantly smaller than the room averaged value have been postulated if the system is to be regarded as a task conditioning system. Under normal office conditions the concept therefore must be supported by a second supply system that allows the introduction of the air close the head.

As the experimental investigation of an improvement of the system is difficult, a new option could be investigated with the help of numerical simulation.

Table 1 Applied heat sources in the experimental set-up

heat sources	power [W]	convective contribution
mannequin	125	49 %
PC-simulator (1)	104	63 %
PC-simulator (2)	102	63 %
desk lighting	24	39 %
ceiling lighting	152	39 %

This paper discusses full scale measurement and CFD-simulation results of the DDV-concept. Results are obtained for different boundary and cooling load conditions. Furthermore, a comparison between transient measurement results and simulations is given, based on which an improved version of the DDV-concept is investigated numerically.

EXPERIMENTAL SET-UP

For this work a full scale climate chamber experimental set-up has been designed to test the performance of the DDV-concept on thermal comfort and air quality parameters. An office model, $L\times B\times H = 5.16\times3.6\times2.7$ m^3 (0.2 m plenum height included), has been built in a larger climate chamber ($5.16\times9.7\times2.7$ m^3) at the Eindhoven University of Technology. The floor, the ceiling and two walls are directly temperature controlled; the remaining two walls are controlled indirectly. An air handling unit is used to condition the air to the plenum. A standard displacement ventilation unit has been used.

Different heat sources have been applied. A simplified thermal mannequin has been designed that represents the human body, both geometrically and in respect of the heat distribution (Loomans and Rutten 1997). Furthermore, PC-simulators and lighting are applied. Table 1 summarizes the heat sources and the consumed power. The convective part of the heat input was determined from comparative infrared thermography.

Omnidirectional low-velocity anemometers have been applied for velocity measurements and thermocouples and positive temperature coefficient resistors (PTC's) for temperature measurements. The accuracy of the velocity measurements has been determined at ± 0.02 m/s (velocity > 0.05 m/s). The thermocouple measurements are accurate within ±0.1°C and the PTC's within registration accuracy (±0.125°C).

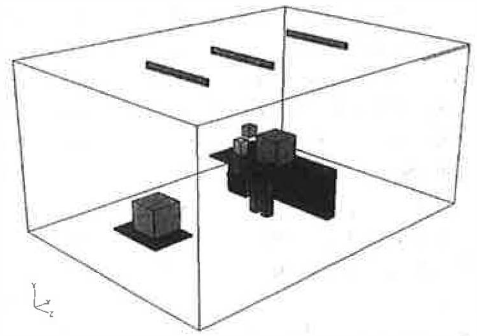

Figure 2 Experimental and CFD-simulation configuration

In the transient experiments additional PTC's were positioned close to the thermal mannequin to measure the temperature course in the boundary layer of the mannequin as a function of time.

NUMERICAL TECHNIQUE

CFD has been used to calculate the air flow pattern for the DDV-configuration. Figure 2 shows the configuration that is used for the simulations and which corresponds with the experimental one. For the current work Fluent/UNS (Fluent 1996) has been applied.

The CFD analysis is conducted applying a low-Reynolds number modified RNG-k-ε model. Standard wall functions have been applied. Furthermore, an adapted version has been used where the heat conduction coefficient and the laminar viscosity at the first grid cells near floor and ceiling are changed to allow the heat transfer characteristics to approach experimental values as presented in Chen and Zheng (1992; Loomans and Rutten 1997). Currently it is not possible to prescribe an internal heat transfer coefficient for the applied CFD-model.

The heat flux at the heat sources is corrected for radiant heat transfer, as determined from measurements (see Table 1). The flow rate, wall temperatures and the inlet temperature are taken from the experimental results.

Table 2 Case description (steady state)

case	air change rate $[h^{-1}]$	heat input [W]	T_{inlet}-T_{wall} [°C]
1	1	125	- 2.5
2	2	125	- 3
3	1	500	- 2.5
4	1.5	500	- 3
5	2	500	- 3

Table 3 Case description (transient)

air change rate $[h^{-1}]$	heat input [W]	T_{inlet}-T_{wall} [°C]
1.5	125	$-6 \rightarrow -3$

CASES

Several cases have been investigated experimentally and numerically, steady state and transient. Set-point values for the steady state cases are summarized in Table 2. For the transient situation one measurement result was used to compare with the transient CFD-simulation (see Table 3). The inlet temperature was increased from 17°C to 20°C within a short time interval. Wall temperatures were maintained at 23°C.

Recent results have indicated a significant air leakage in the supply ducts of the experimental set-up outside the test room. Measurements of the local mean age of the air at the exhaust, for the same configurations, however allow the determination of the actual flow rate in the room (Roos 1998). The currently available results have been used for the subsequent steady state simulations. Due to time constraints, for the transient simulation the applied flow rate couldn't be corrected yet.

For each steady state measurement case, the air temperature has been measured at 520 positions. The values have been corrected for radiant heat transfer to the surrounding walls, but not for the radiant heat transfer from the heat source(s) in the room.

The absolute velocity has been measured at 132 positions for each steady state case. In the calculation of the velocity, the flow direction and calibration position have been taken into account. Measurement values were checked on the lower velocity range (0.05 m/s). As the velocity at most positions in the room remained under the lower velocity range for more than 90% of the registrations, measurements were concentrated in the area close to the inlet and in the plume.

All results are available in *x-y-z* format for further data-analysis.

EXPERIMENTAL RESULTS

Figure 3 presents an example of the temperature distribution at different planes in the room as measured for case 4. From the figure the buoyant plumes above the heat sources (lighting and PC-simulator) are shown. Furthermore, at the inlet side of the desk the temperature gradient near the floor is higher than at the other side of the desk. The horizontal temperature gradients in these planes are small compared to the vertical gradient, except near the plumes.

The mean dimensionless temperature at height *h* is determined from six positions in the room, outside the plume (x = 1.50 m / 2.00 m / 4.50 m ; z = 0.675 m / 2.925 m).

$$\theta_h = \frac{\sum_{n=1}^{6}\left(T_{h,n} - T_{inlet}\right)/6}{T_{exhaust} - T_{inlet}}, \qquad (1)$$

where θ_h = temperature at height *h*
T = air temperature

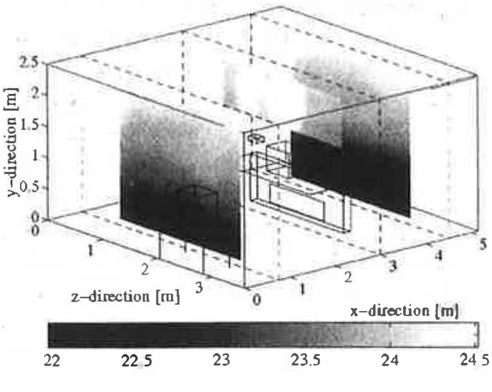

Figure 3 Measured temperature course [°C]; case 4 (x =0.75/3.0/4.5m)

In Figure 4 results for the dimensionless temperature profiles are shown for the different cases. In Figure 4(a) two measurement results are given for each case. The reproducibility of the measurements is good.

The following remarks can be made:

- The maximum temperature in the room is not reached at ceiling height. The temperature controlled ceiling results in a large temperature gradient near the ceiling. The exhaust is a 0.03 m high rectangular opening situated just beneath the ceiling. The ceiling operates as a cooled ceiling.
- The dimensionless temperature profile is correlated with the flow rate. This is in agreement with Mundt (1996). For higher cooling loads, the profile is less sensitive to the flow rate. This is mainly explained from the larger temperature difference between exhaust and inlet (see Table 5).
- For case 3 the cooling capacity of the ceiling is ~80% of the total cooling load. From the temperature profile and from visualisations the flow pattern is regarded as a displacement ventilation pattern. These results are in agreement with the findings of Krühne (1995). In the current experiments however the flow rate at the inlet is changed, whereas Krühne changed the inlet temperature, to adapt the convective cooling/cooled ceiling capacity relation. This experimental difference will

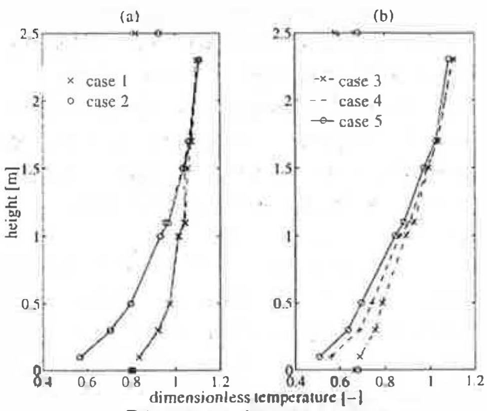

Figure 4 Dimensionless temperature profiles steady state cases

Table 4 Velocity [m/s]; case 4 plume thermal mannequin

height [m]	position back of the head	position forehead
1.5	0.19	0.26
2.0	0.28	0.25
2.3	0.26	0.23

Table 5 Relation between temperature gradient and plume velocity

case	[°C]*	[°C/m]**	[m/s]***	[m/s]****
1	2.7	0.3	0.31	0.32
2	3.1	0.8	0.30	0.19
3	3.5	0.7	0.27	0.27
4	4.0	1.0	0.28	0.25
5	3.8	1.0	0.27	0.23

* $T_{exhaust}$ - T_{inlet}
** vertical temperature gradient: $T_{2.3m}$ - $T_{0.1m}$/2.2 m
*** velocity 0.8 m above head at 2.0 m height
**** velocity in front of head at 2.0 m height

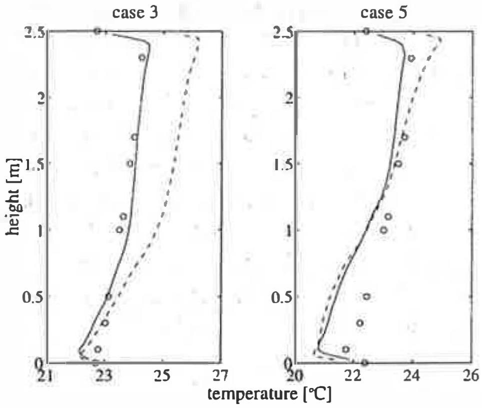

Figure 5 Temperature profile

result in differences in the temperature gradient near the floor at higher cooled ceiling capacity levels. In that case deviations with the results of Krühne may be possible, especially with regard to the contaminant distribution (see Roos 1998).

Velocities are small and below the specification level, except for the buoyant plumes. For the thermal mannequin, velocities were measured at different heights at positions right above the back of the head and at 0.02 m in front of the forehead (see Table 4). The measured velocities are higher than the results presented by Mundt (1996). The difference is due to the form of the heat source and the higher heat load. The measured velocities indicate the flow pattern at the mannequin. Near the mannequin the highest flow velocity is found at the front of the mannequin. This is confirmed by visualisation.

The plume velocities show a relation with the vertical temperature gradient (see Table 5). It is interesting to note that the plume direction is partly influenced by the flow rate, especially for the cases with a low heat input. For a higher flow rate the measurements indicate that the plume is bent backwards, though only slightly. This effect may be explained from the higher level of the interface height for the higher flow rate. The interface height is the height were the ventilation flow rate equals the convection flow rate in the plume(s).

SIMULATIONS

In Loomans and Rutten (1997) preliminary CFD-simulation results are described for a case corresponding with case 1. Results are shown for simulations in which the standard wall functions are applied. Deviations in the simulated temperatures were large, especially near the floor. Given the discretisation better results have been obtained by imposing a higher heat transfer at the wall.

The applied CFD-program does not allow the definition of a heat transfer coefficient as part of the boundary condition definition. In the simulations therefore the first grid node at the floor and the ceiling have been adapted by raising the heat conduction coefficient and the laminar viscosity. This approach gave reasonable results for the preliminary investigation.

The same approach has been adopted for the current results. A new grid was created that incorporated all the present heat sources. Careful attention has been given to the agreement of the grid distribution near the wall. Figure 5 presents the temperature profiles for the measurements and the simulations for case 3 and 5. The profiles have been determined analogous to the dimensionless temperature profile.

When standard wall functions are applied (in both simulations $\overline{y^+}=11$) the agreement between measurement and simulation is unsatisfactory. Application of the approach described above improves the results but is seen to be case-dependent. Given the obtained results, a large part of the difference between measurement and calculation is however explained from the underestimation of the wall heat transfer. It is not possible to improve the simulation results practically applying the current version of the CFD-program.

The simulated flow field near the thermal mannequin confirms the visualised flow field. Highest velocities are found at the front of the mannequin. Given the available measurement results and the results of Mundt (1996), the simulated centre plume velocity is too high and the plume width too small. The simulated flow rate above the mannequin is up to 50% smaller than the average figures presented in Mundt (1996) and is closely related to the vertical temperature gradient. This difference contributes to the deviations as found between the two types of simulations and in comparison to the experiments.

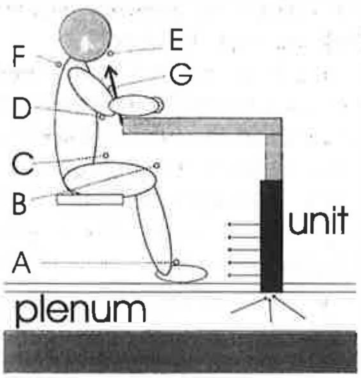

Figure 6 Improved DDV-concept + Measurement locations

TRANSIENT RESULTS

The results of the research into the applicability of the displacement ventilation concept for the control of a micro climate are described in Loomans et al. (1998). The main conclusion is that the desk displacement ventilation concept cannot be used solely as an individually controlled air conditioning system for a standard office configuration. A parallel system is necessary to accomplish a fast response to a change in settings by the occupant. CFD has been applied to investigate the response improvement of an additional system that is incorporated in the desk displacement ventilation concept.

Figure 6 presents the combination of the desk displacement unit and a variant of the so-called Climadesk (Wyon 1995); a small slot in the desk top. Here the displacement ventilation system is used to allow a base ventilation. The flow rate through the desk slot is small (5-10% of the base flow).

First, the transient CFD-simulation of the original concept has been compared with measurement results obtained close to the mannequin for a temperature increase at the inlet as given in Table 3. In the measurement set-up the desk slot is not present. In the simulations the time step was set to 60 sec. For each time step 1000 iterations were solved if there was a temperature variation over the time step. If

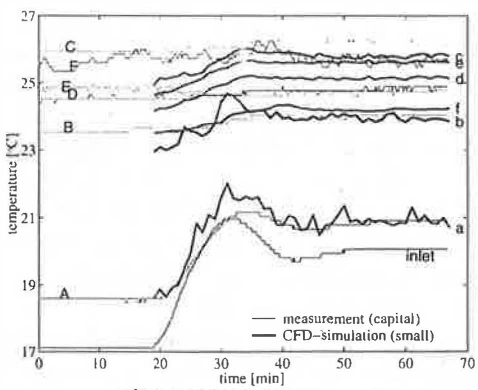

Figure 7 Comparison transient temperature course (original concept)

not, 250 iterations were solved. Steady state simulations indicated that the number of iterations for each time step was ample to reach a stable solution. The raised heat transfer variant was applied.

Figure 7 presents the temperature course as a result of a temperature variation at the inlet for the experiment and the simulation. The letters in Figure 7 correspond with the positions as indicated in Figure 6.

Differences in the absolute values mainly result from the discretisation of the flow problem. At positions close to the inlet the results agree relatively well with the measurement results. At higher positions the influence of a lower heat transfer at the floor (with reference to the steady state simulation results) and a too high flow rate (~25%) are evidenced by the reduced damping.

The calculations require an extended simulation time duration. From the comparison it is however concluded that the application of CFD for the qualitative investigation of a time dependent flow problem is feasible.

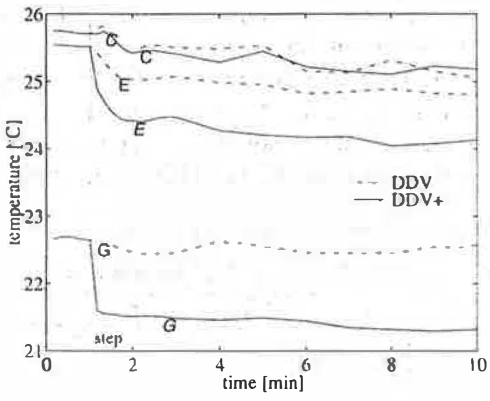

Figure 8 Temperature course (improved concept)

Table 6 Damping [%]

position	A	B	C	D	E	F	G
DDV	15	71	73	76	72	92	92
DDV+	35	67	75	66	45	84	48

Figure 8 presents the temperature as function of the time for three positions near the occupant. For the *DDV* case only the temperature at the displacement ventilation unit is lowered step-wise by 2.7°C. For the *DDV+* case, besides the displacement ventilation unit, the desk slot is operated. The temperature conditions at the desk slot are the same as for the unit. The flow rate is 6 m^3/h. The velocity at position *G* due to the flow is 0.27 m/s, whereas at position *E* no significant increase in the prevailing velocity is calculated.

In Table 6 the damping is given for the two cases. The steady state situation after 50 minutes is taken as an end-point. The positions are indicated in Figure 6.

The positive effect of the desk slot is shown by the large decrease of the damping at mouth height (position *E*). As the damping for the *DDV* case is underestimated compared to the measurement results (see Figure 7), the improvement will be better. At position *A* the CFD solution shows larger fluctuations, which in the end result in a deviating value for the damping.

Figure 8 indicates that the response time to the change in the inlet conditions at position *E* is also improved. The simulation time step was set to 10 sec for the first three minutes of the simulation process. After that a time step of 60 sec was applied.

CONCLUSIONS

Full-scale measurement results are described for the DDV-concept. These results have been obtained for the research into the applicability of the concept for standard office environments. Furthermore, with these results extended comparison with CFD-simulations is possible.

In an earlier study the applicability of the concept has been described and found inadequate for standard office arrangements. In this paper steady state results for different boundary conditions are described and compared. An example is given of an improved version of the DDV-concept as investigated numerically.

In the experimental set-up the ceiling operates as a cooled ceiling for most of the investigated cases. A displacement type of flow pattern is maintained. Contrary to the experiments of Krühne (1995), the cooling capacity of the ceiling is determined from a changing flow rate at constant inlet temperature. Thus a temperature gradient will remain present. This experimental difference and the constant temperature walls may result in deviations with the results of Krühne at a higher cooled ceiling capacity level, especially with regard to the contaminant distribution (see Roos 1998).

The agreement between the steady state experiments and CFD-simulations is unsatisfactory when standard wall functions are applied. Improved results are obtained when the wall heat transfer is increased. The presented results indicate the importance of the wall heat transfer in this type of flow problem. They subscribe to the earlier findings. Better results are expected when the wall heat transfer characteristics can be controlled more directly or when the near wall flow field can be solved more accurately.

Finally, a transient numerical study has been performed to investigate an improved version of the DDV-concept. The comparison between measurement and CFD-simulation of the original concept indicated the feasibility of applying CFD for this type of qualitative study.

The addition of a so called desk slot (compare with the Climadesk (Wyon 1995)) improves the transient characteristics of the thermal parameters, close to the head, considerably. In the example the desk slot is operated constantly at the desired supply air temperature. Another option would be to let the desk slot react directly to a change set by the occupant for a restricted period of time. After this time period (e.g. 5 minutes) the air supply through the desk slot would gradually reduce to zero. This type of adjustment would mitigate the short-term restrictions of the displacement ventilation system. Furthermore, it would provide an answer to the fact that often changes in the thermal requirements are short-term based.

The application of CFD for this type of comparative study of ventilation concepts is feasible. However, if absolute values are required, comparison with experimental results is necessary.

REFERENCES

Chen, Q. and Zheng, J. (1992) Significant questions in predicting room air motion, *ASHRAE Transactions*, Vol.98, part 1, pp.929-939.

Fluent (1996) *Fluent/UNS User's Guide*, Fluent Inc., Lebanon, USA

Krühne, H. (1995) *Experimentelle und theoretische Untersuchungen zur Quellluftströmung*, thesis, Technische Universität Berlin, Germany.

Loomans, M.G.L.C., van Mook, F. and Rutten, P.G.S. (1996) The introduction of the desk displacement ventilation concept, *Proc. ROOMVENT '96*, Vol.1, pp.99-106, Yokohama, Japan.

Loomans, M.G.L.C. and Rutten, P.G.S. (1997) Task conditioning + displacement ventilation, 1+1>2 ?, *Proc. Healthy Buildings/ IAQ '97*, Vol.2, pp.305-310, Washington D.C., USA.

Loomans, M.G.L.C., Rutten, P.G.S. and Schellen, H.L. (1998) The applicability of displacement ventilation for individual control of a micro climate, *ASHRAE Proceedings Healthy Buildings/IAQ'97* (in preparation)

Mundt, E. (1996) *The performance of displacement ventilation systems*, thesis, Kungl Tekniska Högskolan, Stockholm, Sweden

Roos, A. (1998) The air exchange efficiency of the desk displacement ventilation concept. Theory, measurements and simulations, *Proc. ROOMVENT'98*, Stockholm, Sweden

Wyon, D.P. (1995) Thermal manikin experiments on Climadesk. *Proc. from the workshop on Task/Ambient Conditioning Systems in Commercial Buildings*, ed. F.Bauman, CEDR, University of California, Berkeley, USA