CFD MODELLING OF MOISTURE INTERACTIONS BETWEEN AIR AND CONSTRUCTIONS

Lone Hedegaard Mortensen[1], Monika Woloszyn[2], Raluca Hohota[3] and Gilles Rusaouën[2]
[1]Department of Civil Engineering, Technical University of Denmark, Lyngby, Denmark
[2]CETHIL (Thermal Sciences Centre) UMR CNRS 5008 - UCBL - INSA Lyon, France
[3]Faculty of Building Services and Equipment, Technical University of Civil Engineering Bucharest – UTCB, Romania

ABSTRACT

There is a strong demand for accurate moisture modelling since moisture poses a risk for both the constructions and the indoor climate. Thus, in this investigation there is special focus on moisture modelling. The paper describes a new model based on a CFD tool that is enhanced to include both detailed modelling of airflows in rooms and heat and moisture transfer in walls by applying them as fluid walls. In a 3D configuration the impact of different boundary conditions are investigated and the results are discussed. The changes of boundary conditions that are studied are velocity, moisture and temperature conditions for room air.

INTRODUCTION

Moisture plays a significant role in the development of many processes that are harmful to both the quality of indoor air and the conditions of the constructions in the building envelope. Therefore it is anticipated that moisture modelling can help to ensure design of healthier indoor environments and constructions. In building simulation models it is common to consider just one node for the air in a room and to assume fully mixed conditions. If a more detailed analysis of the air distribution in the room is needed it can be obtained by use of CFD simulations.

The focus of this investigation is on the multi-dimensional moisture interactions between constructions and the room air. The moisture flow between air and construction depends strongly upon the airflow conditions close to the surface, which is influenced by the airflow patterns in the room. Thus, it is important to investigate the airflows carefully and to estimate their influence on the moisture transport. Where other models have focused on either air and moisture flows in rooms or on heat and moisture transfers in walls, the model described in this paper tries to combine the detailed modelling of both. Hence, the attention of this paper is on moisture transfers in both air and the surrounding building envelope by use of an enhanced CFD model.

CFD is a very interesting tool for modelling and the potential of CFD simulations is great. CFD provides possibilities for developing new state of the art models that provides easy visualization of the results that can replace the very costly experiments, and after all the goal is to ensure better design of buildings. Other researchers have already tried to use the advantages of CFD for building simulation. Negrão, 1995; Bartak et al., 2002; Zhia et al., 2002; Zhia and Chen, 2003 have all attempted to couple CFD and energy simulations of building envelopes. In relation to moisture interactions Clarke et al. (1999) developed a model that could predict conditions leading to mould and recently Hohota (2004) has shown that CFD models can be used to predict condensation on surfaces.

In this paper the CFD model, Fluent (version 6.1) is used. This work is not combining two existing models but enhances an existing CFD model to include heat and moisture transport in walls. The new scheme for modelling the moisture transfers between the room air and the surrounding walls is to use fluid walls. The aim is to obtain more information about what happens in microclimates that can be critical. One example of such a microclimate is behind a piece of furniture placed close to a cold external wall or in a corner, which represent a thermal bridge with limited airflow on the surface. The idea is to study how the diffusion and the airflows influence the vapour content at the internal surface. This is very important since it is in areas like this that mould growth can occur and it has a negative impact on the indoor climate.

MODEL

In this model the air is represented as a mixture of dry air and water vapour and the walls are modelled like immobile fluids with ordinary wall characteristics as material properties. This enables modelling of moisture transfers within the walls. When the walls are modelled as fluids the user can add a diffusion model for moisture. The diffusion model simply has two boundary vapour contents and then performs a linear regression between the values. Alternatively the wall could have been modelled as a solid but solids modelled in Fluent do not automatically contain a diffusion model, so then a user defined function for the solids would need to be programmed. User defined functions cause some amount of overhead in the execution of Fluent.

The fluid wall model includes heat conduction through the fluid walls but not radiation in the room. The thermal part of the model approach with fluid walls has been validated by a simulation where the temperatures found on the interior surface of the walls have been compared to experimentally obtained values. The experiments were performed by Hohota (2004). The comparison verified a good thermal coupling in the fluid wall approach where the measured temperatures on the internal surface of the wall were similar to the temperatures found by use of the model.

The moisture transfer between the wall and the room air in the model is programmed by user defined functions. More exactly the moisture in the model is transferred by the fluxes. The two fluids, room air and wall, are separated by another "wall" (barrier) in the program so they are unable to interact. To understand the phenomena one could imagine a boundary that is in contact with the room air and a copy of this as its own shadow, which is in contact with the wall. The two faces, the real and the shadow, are placed at the same position but are not in contact. This is illustrated in Figure 1 where the real and the shadow wall for illustrational purpose are placed a small distance apart.

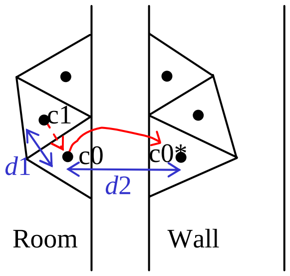

Figure 1 A close up of the face separating the two fluid zones, room air and wall. On each side of the face there are two meshes with same face geometry

As it can be seen in the figure the two faces are not in contact but have the same face mesh. However, this does not mean that the volume size of the mesh element generated from it is the same for corresponding face meshes. To make the vapour content in the two fluids interact a flux transfer is programmed. First step is to perform a test to determine the direction of the moisture transfer as shown in Equation 1 and then perform the transfer.

(1) $$m_{w,c0} > m_{w,c0*}$$

In the equation the water vapour content in the air ($m_{w,\ c0}$) is compared to the vapour content in the wall ($m_{w,c0*}$). The cell in the air c0 refer to a cell next to the surface whereas the * notation refer to the cell adjacent to c0 but in the wall. If the content in the air is highest then the transfer is from the air to the wall or else from the wall to the air. The transfer principle is shown in Equation 2.

(2) $$m_{w,c0*}^{n} = m_{w,c0*}^{n-1} + \frac{\left(m_{w,c1}^{n-1} - m_{w,c0}^{n-1}\right)}{d1} \cdot d2$$

In this case it is assumed that the room air contains the highest amount of water vapour. Then the new water vapour content in [kg/kg] of the mesh element c0* at iteration *n* is equal to the previous vapour content in the mesh element plus the flux coming to the corresponding mesh element c0 in the air. The flux arriving to the cell c0 is proportional to the vapour content of its neighbour cell c1 at the previous time step minus c0's own vapour content in the previous step, and this is weighed by the distance between the centres of c1 and c0 and the distance between c0 and c0* to avoid divergence. In addition to this transfer, in order to assure mass conservation, the exact same amount of water vapour content is withdrawn from the c0 cell.

SIMULATIONS

The objective of the developed model was to investigate the vapour content in microclimates. The study is performed by use of a case. The chosen case is just one example of an interesting microclimate that can provide some indication of what to expect in areas with reduced air circulation and/or in combination with cold surfaces. Here it is chosen to use a single room of 2x2x2m^3, with an air inlet in one side and outflow of air on the opposite side. The two walls with the inlet and outlet are simulated as immobile fluid walls with thicknesses of 0.1m and models as described in the previous section. The remaining wall boundaries are modelled as internal surfaces. The wall with the inlet is in contact with the outdoor air (0°C) whereas all others are internal walls with a temperature of 22°C. The inlet is a rectangular duct with dimension 0.2x0.1m^2 that continues 0.1m inside the room where it ends in an orifice slit of 0.02x0.05m^2.

In real rooms or offices there are moisture sources in terms of person and activities. So to mimic a room by simulation it is necessary to include some moisture sources. The inlet is a source but if all surrounding constructions are moisture tight the result is obvious and the room will have the same vapour content as the inlet air. To keep the geometry simple it is chosen to use some 'wet walls' as moisture sources.

The floor and the two sides of the room are simulated with moisture tight surfaces and the ceiling and the remaining 2 walls as moisture sources/sinks with constant vapour content. The vapour content of air on the outside of the inlet wall is constantly 4.0g/kg corresponding to a relative humidity (RH) of 100% and for the outlet wall constantly 10.0g/kg corresponding to 60%RH on the surface that faces away from the object room. The RH at the ceiling is constantly 48% RH (8g/kg). The last moisture source in the room is the inlet, which has moisture level as in the outdoor air but this is heated to 20°C in order to avoid modelling of radiators in the model and hence the inlet air has 27%RH (4.0g/kg).

In Figure 2 the geometry of the tested case is shown. It can be seen that a 0.45x1.0x1.6m^3 piece of furniture is placed in the corner near one of the side walls and the inlet wall. The air gap between the walls and the furniture is 0.02m and 0.05m between the furniture and the floor.

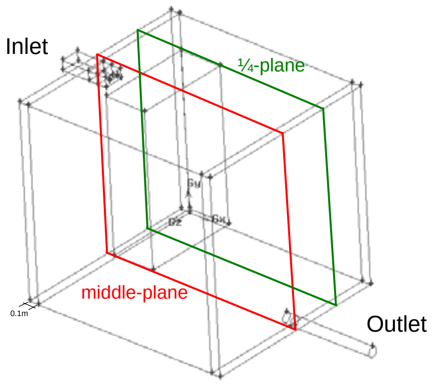

Figure 2 Geometry of the 3D model also showing placement of result planes in the room. It should be noted that the middle-plane is right at the end of the furniture and in the centre of the inlet and outlet

The furniture is included in the case in order to analyze the microclimate behind it. This includes detailed investigation of the effect of the airflows behind the furniture and analyzes the impact on the vapour content in both air and constructions.

All tests are performed in Fluent in 3ddp (3D double precision). This code is based on ordinary conservation equations for mass, momentum and energy. Within the code the used models are energy, realizable viscous model with enhanced wall treatment and species transport. The realizable k-epsilon model is used since it produces more accurate results for boundary layer flows than the standard k-ε model. The gravity is -9.81m/s^2. The walls are simulated as fluids with material properties of cellular concrete and are fixed to have no velocities within the walls and are declared as laminar zones. In the materials the mixture consists of water vapour (H_2O), immobile wall fluid and air. In Table 1 more details about the used material properties for air and the fluid walls are given.

In all tests it was chosen to use an unstructured mesh in order to avoid a guidance of the flow. Furthermore, an unstructured mesh is better at dealing with corners and complex geometry than a structured mesh. Another advantage of the unstructured mesh is that it is much easier to refine in certain zones where more information on the flow or the distribution of temperature or moisture is valuable. However, the unstructured mesh can cause problems near the walls and this is compensated for by use of a so-called enhanced wall treatment (Teodosiu *et al.*, 2003).

Table 1 Material properties for air and walls

MATERIAL PROPERTIES FOR AIR AND FLUID WALLS	
AIR	
Density, ρ	1.225kg/m^3
Heat capacity, c_p	1006J/kg·K
Thermal conductivity, λ	0.0242W/m·K
Mass Diffusivity	1.55·10^{-5}m^2·s
FLUID WALLS	
Density, ρ	400kg/m^3
Heat capacity, c_p	960J/kg·K
Thermal conductivity, λ	0.1W/m·K
Mass Diffusivity	1.55·10^{-5}m^2·s

Table 2 Imposed boundary conditions for case 1 (reference). The boundary conditions are for the inlet air and the external side of the walls

BOUNDARY CONDITIONS			
Inlet air	v	0.167m/s, air chg. rate 1.5h^{-1}	
	t_{air}	20°C	
	H_2O	4 g/kg (RH 27%, 20°C)	
	d_h	0.133m	
	TI	6.0%	
Wall temperatures	t_{walls}	22°C for all except in-/outlet	
	t_{outlet}	22°C	on external side of the fluid walls
	t_{inlet}	0°C	
Vapour content	H_2O	8g/kg	(48%RH) ceiling
	H_2O	4g/kg	(100%RH) inlet wall
	H_2O	10g/kg	(60%RH) outlet wall

Simulated cases

The mesh used in the model is unstructured and the grid contains about ~1,000,000 tetrahedral elements in the first part of the simulation where the general airflow is determined. In the second part of the simulation the mesh is refined in the area between the furniture and the wall so the amount of elements is increased to 1,200,000.

The tested case was a single room, see Figure 2. Several variations of the case have been tested. The tests have only concerned 3d cases since they were non-symmetrical. In the simulated cases the inlet wall is a sink and both ceiling and outlet walls are moisture sources. The total net moisture load from the walls is 38g/h. A total of 5 cases have been tested. All variations have been compared to case 1, which is chosen as a reference. The reference case has boundary conditions as seen in Table 2. To investigate the importance of the inlet parameters; humidity, temperature and velocity, 3 variations have been tested, cases 2-4. The effect of a change in temperature on the outside surface of the inlet wall is studied in case 5 with a higher temperature. All variations are listed in Table 3.

Table 3 The 5 simulated cases, note that in case 3 the higher inlet humidity is due to a lower temp.

SIMULATED CASES				
	Inlet humidity [g/kg] (RH)	Inlet temp. [°C]	Inlet velocity [m/s]	Outs. inlet wall temp. [°C]
Case 1	4 (27%)	20	0.167	0
Case 2	8 (54%)	20	0.167	0
Case 3	4 (74%)	5	0.167	0
Case 4	4 (27%)	20	0.333	0
Case 5	4 (27%)	20	0.167	10

RESULTS

All simulations have been continued for 3300 iterations where the last 300 iterations are done with a refined mesh in the region behind the furniture. After the iterations the residuals found for the continuity are in the range $6.88 \cdot 10^{-5}$-$2.38 \cdot 10^{-4}$, which is low and hence convergence is assumed reached.

The focus of the results is on the microclimatic conditions behind the furniture and more specifically on the moisture. Each of the variations is compared with the reference, case 1. For convenience the results are given as planes at two different locations in the simulated room. The placements of the planes are shown in Figure 2.

First results of case 1 are shown. In Figures 3 and 4 the moisture distribution in the room is shown in values of RH. The average RH of the air in the entire room is 44%RH, 20.9°C (6.7g/kg). On the entire internal surface of the inlet wall RH average is 46%RH, 20.7°C (6.9g/kg) but where it faces the furniture it has 47%RH and 20.3°C (7.0g/kg). So the furniture has an influence on the moisture and together with the temperature the differences from behind the furniture to its front side is 3%RH.

In Figure 5 the temperature distribution is given. The average air temperature in the entire room is 20.9°C. The average temperature of the furniture is 20.7°C. It should be noted that there is a linear temperature distribution in the inlet wall.

The airflows in the room for case 1 are shown in Figure 6 and 7. It can be seen that the airflow follows the ceiling and spreads out on the opposite wall. The velocities in the rest of the room are very low and most are in the range 0.02-0.03m/s. Maximum velocity is 2.4m/s at the orifice inlet.

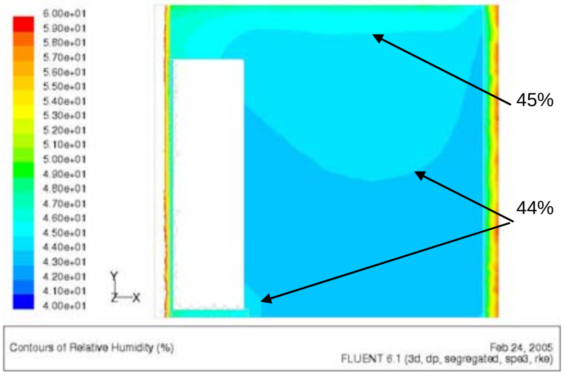

Figure 3 RH distributions at the ¼-plane in the room (case 1). In the white area in the left wall indicate that the moisture level is out of scale. The scale of RH is from 40% in the furniture to 60% at the external wall

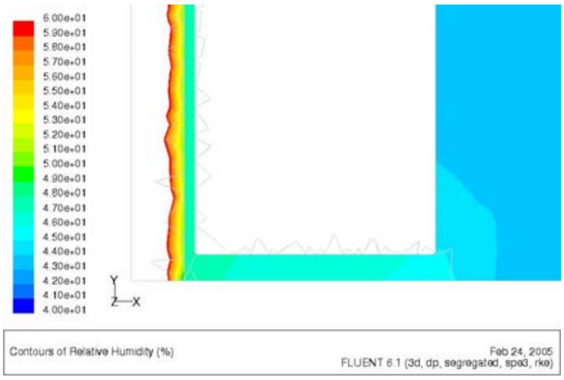

Figure 4 Close up of the RH distributions at the ¼-plane (case 1). The RH behind the furniture is 47%. Under the furniture the RH is 47% (7.0g/kg) in the corner and decreases slowly to 44% (6.6g/kg) in the room

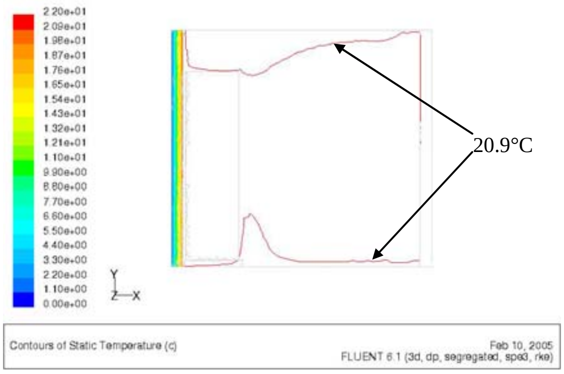

Figure 5 Temperature distributions in the ¼-plane (case 1). The isothermal lines indicate where the temperatures changes between 20 steps in the range 0-22°C. In most of the room air the temperature is slightly less than 20.9°C

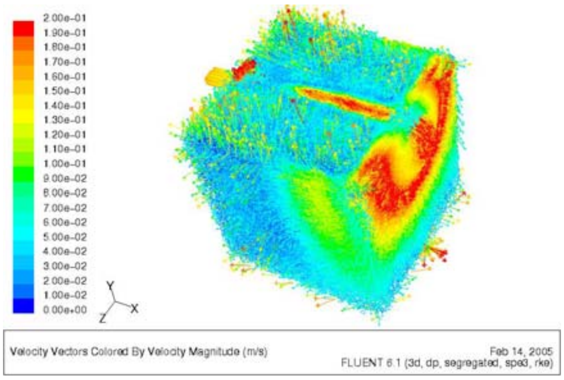

Figure 6 Vector plot of the velocities in the room for case 1. The highest velocities are from the inlet and near the ceiling and on the outlet wall where the jet spreads out. Note that only velocities in the range 0-0.2m/s is shown

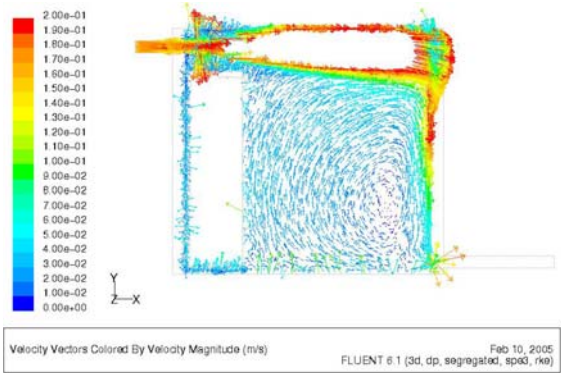

Figure 7 Velocity distributions in the middle-plane for case 1

Figure 8 show the velocities at the ¼-plane. The average velocity of the room air in the ¼-plane is 0.025m/s. The blue colour behind the furniture indicates that the airflow velocity is low but still slightly higher than the average in the ¼-plane since the range is 0.03-0.04m/s. To gain more information about the airflow pattern behind the furniture a close up is shown in Figure 9. In the figure it is possible to see the direction of the airflow behind the furniture. The flow falls to the floor between the wall and the furniture and rises on the face turning to the outlet wall.

In case 2 the moisture load in the inlet air was 8g/kg at 20°C corresponding to air with 54%RH. The main difference between case 1 and case 2 is that the moisture level in the room is increased. There is no difference in the temperatures and the velocity distributions in the room. In the room air there is an average of 52%RH (8.0g/kg) and on the internal surface of the inlet wall the average is 53%RH (20.7°C). On the wall behind the furniture there is 54%RH so again there is a difference from the room level.

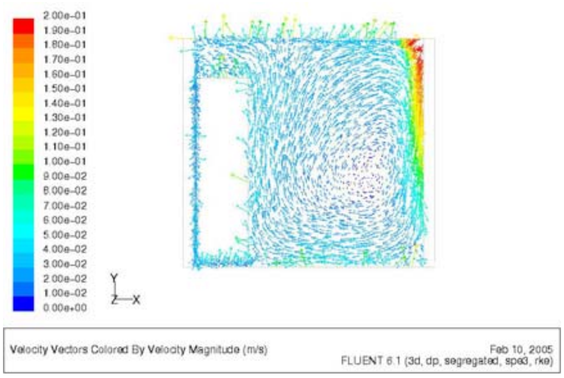

Figure 8 Vector plot of the velocity distribution in ¼-plane (case 1). The highest velocities are at the outlet wall where the inlet jet has spread out

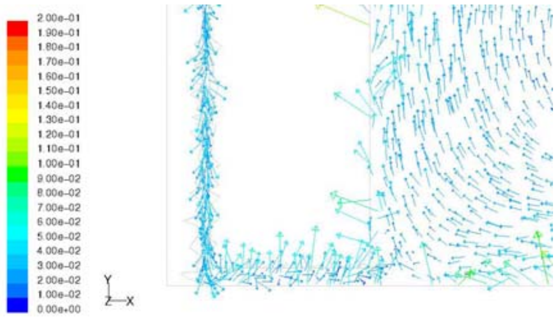

Figure 9 A close up of the corner with the furniture in the ¼-plane in case 1. The vectors indicate the direction of the air flow

In case 3 a lower inlet temperature of 5°C was applied. In this test the inlet jet drops to the floor as expected. In the room air the averages are 47%RH and 19.5°C (6.6g/kg). On the internal surface of the inlet wall there is an average of 47%RH (19.7°C). The distribution of temperatures in the room is shown in Figure 10. The falling inlet jet causes a new distribution of the velocities in the room, where the airflow between the wall and the furniture as well as on the other side of the furniture is streaming up. This is shown in Figures 11 and 12.

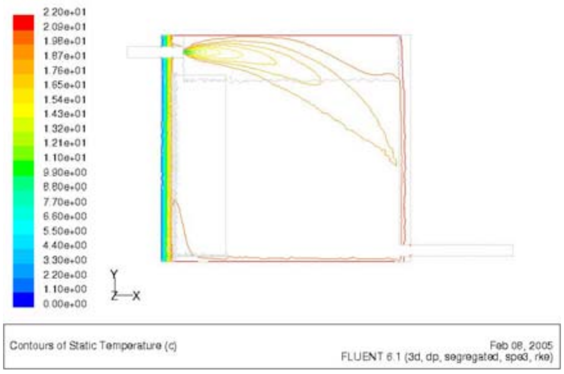

Figure 10 Temperature distributions in the middle-plane for case 3. Note that the inlet jet drops as seen by the temperatures

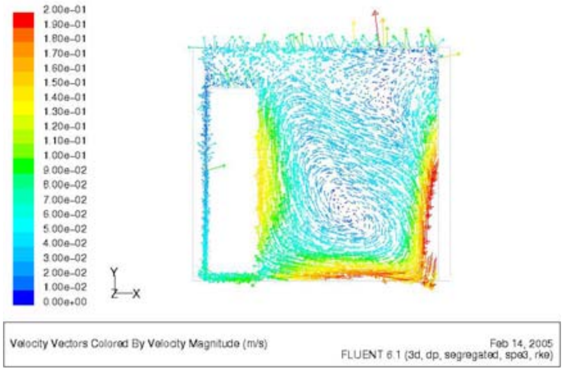

Figure 11 Vector illustrations of velocities in the ¼-plane for case 3. The maximum velocity is near the outlet wall and the floor. The small white areas indicate that the velocity is higher than 0.2m/s

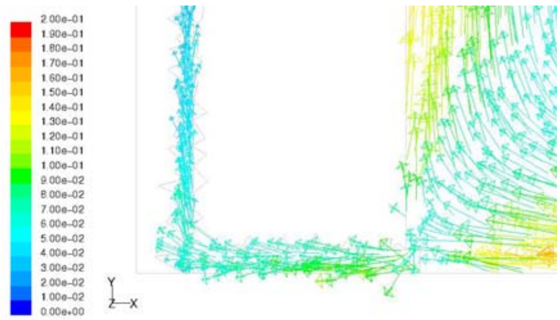

Figure 12 A close up of the velocities around the furniture for case 3. The values under the furniture are in the range 0.07-0.09m/s while the range is 0.04-0.06m/s behind it

In case 4 the inlet velocity is doubled to 0.333m/s corresponding to an air change rate of $3h^{-1}$. This results in average temperature of 20.8°C and RH of 42%RH. On the internal surface of the inlet wall RH average is 43%RH (20.6°C). These values are all lower than in case 1. The airflow distribution in the room has also changed and this is shown in Figure 13. The airflow follows the ceiling from the inlet and spreads on the outlet wall. Then the air is pressed down and it follows the floor until it reaches the furniture where it rises again. Between the wall and the furniture the airflow drops to the floor and the velocity is in the range 0.01-0.03m/s. Between the floor and the furniture the velocity range is 0.02-0.1m/s, which indicates that the air is more turbulent.

The last variation, case 5 was to increase the temperature on the exterior side of the inlet wall from 0°C to 10°C. The results show an average temperature and RH in the room air of 20.9°C and 44%RH respectively, and on the internal surface of the inlet wall the average is 46%RH (20.8°C). These results are almost the same as in case 1. However, there is a change in the temperature distribution in the room as can be seen in Figure 14. The average temperature of the furniture is 20.8°C.

The room velocities are similar to case 1 but are slightly higher. Between the furniture and floor the range of velocities is 0.01-0.09m/s.

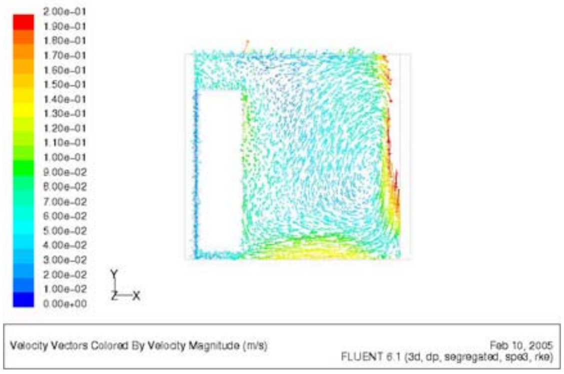

Figure 13 Velocity distributions in the ¼-plane for case 4. The highest velocities are at the outlet wall and the lowest behind the furniture

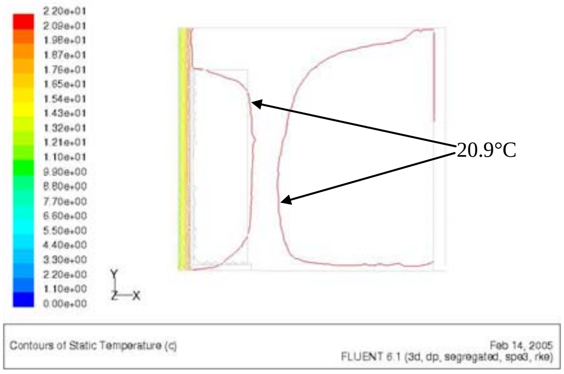

Figure 14 Temperature distributions for case 5 with an external temperature of 10°C (left wall).The highest temperatures are near the floor and ceiling

DISCUSSION

Generally the results are as expected since the airflow patterns in the room follow the ceiling and there is an almost isothermal condition in the room. There is a linear stratification in the fluid walls for both the temperature and the moisture distribution.

The simulation of case 1 showed a vapour content in the room air corresponding to 44%RH (6.7g/kg). This was expected since it is steady-state simulations and the moisture load from the surrounding walls is in equilibrium with 48%RH, 8.0g/kg (ceiling) and 60%RH, 10.0g/kg (outlet wall) in relation to the dryer inlet air of 27%RH, 4.0g/kg. The average air temperature in the entire room is 20.9°C and this is logical since the inlet air is 20°C and the internal walls 22°C and the cold external wall 0°C. The average temperature of the furniture is 20.7°C and the slightly colder temperature is expected since it is placed near the cold external wall. On the internal surface of the inlet wall, where it faces the furniture, it has

47%RH and 20.3°C (7.0g/kg). On the surface of the furniture facing the room there is 44%RH, 20.9°C (6.7g/kg). So the furniture has an influence on the moisture and together with the temperature the differences from behind the furniture to its front side is 3%RH. In this case it is not a problem but in more severe cases the influence of the furniture on differences in moisture levels is highly interesting. In addition to case 1 a similar dry case without any moisture was simulated and the thermal results showed the same average room air temperature and similar temperature distribution.

In the reference case the air change rate of $1.5h^{-1}$ ensures air velocities in the room below 0.2m/s except at the ceiling in front of the inlet and on the top part of the outlet wall. In the rest of the room the velocities are very low and most are in the range 0.02-0.03m/s. These low values would not cause drafts in an indoor environment and therefore they seem reasonable. Behind the furniture the airflow velocity is also low and in the range 0.03-0.04m/s. This means that the flow behind the furniture is slightly higher than in the room in general and it must be explained from the flow along the room surfaces and thereby almost still air in the areas just a little away from the construction surfaces. The flow behind the furniture falls to the floor and rises on the face of the furniture turning towards the outlet wall.

The variation with a doubled moisture load in the inlet air gave almost the same results as the reference case but the moisture level in the room air is increased to an average of 52%RH (8.0g/kg). This was obvious since a bigger moisture load should lead to higher relative humidities but it is still not a risk for the indoor climate.

A lower inlet temperature of 5°C causes the inlet jet to drop to the floor because the density of cold air is higher than warm air. The falling inlet jet affects the airflow distribution in the room. The impact is that the air is moved up on both sides of the furniture. The average RH and temperature for the room air was 47% and 19.5°C. Because of the highly temperature dependent RH it is reasonable that a lower temperature gives a higher RH.

The test with doubled inlet velocity of 0.333m/s corresponding to an air change rate of $3h^{-1}$ gave an average temperature of 20.8°C and RH of 42%RH (6.4g/kg). Both values are lower than for case 1 but the result is obvious since the air coming in is quite dry (27% RH, 4.0g/kg) and when the air change rate is increased from $1.5^{-}h$ to $3^{-}h$ the equilibrium vapour content must be lower. The increased air change rate also cools the room air because more air of 20°C replaces the warmer air. However, this should have the opposite effect on the RH, which should be increased by higher temperatures but the much drier air entering the room dominates this effect. The airflow distribution in the room also changed. It follows the ceiling from the inlet and spreads on the outlet wall and then it is moved down and follows the floor until it reaches the furniture where it rises again. Behind the furniture the airflow drops to the floor.

In the last variation the temperature on the exterior side of the inlet wall was changed from 0°C to 10°C. The average results were the same as in case 1. However, the temperature distribution in the room changed a little and e.g. the average temperature of the furniture was 20.8°C, which is 0.1°C warmer than for case 1. The change is realistic since the exterior wall was warmer and this affects the furniture, which is placed near the wall.

The RH values found for all the simulated cases are all low, between 42%RH and 52%RH. The found levels seem reasonable but the model results should be compared to measured values in order to assure validation. The found low range means that the moisture does not pose a risk to either the indoor climate or the constructions. On the internal surface of the inlet wall the worst case (double inlet moisture load) gave an average of 53%RH (20.7°C), which should not cause problems. If a decreased temperature on the external surface of the inlet wall was used this could give a lower surface temperature and thus maybe a higher RH on the surface. Such a situation would provide a more severe case for studying the effect of the microclimate behind the furniture. However, the purpose of this paper was to demonstrate that it is possible in the first place to perform this type of investigation.

In general the airflow in the room is found to influence the airflow direction behind the furniture. This means that the placement of inlet and outlet in modelling of microclimates is very important and it could change the flow patterns completely.

In this simplified model no radiation model was used. This could cause a possible error in the results of the temperature and thereby influence the result for the microclimate in the corner behind the furniture.

Yet another thing that can influence the microclimate that has not been taken into account is the moisture interaction with the furniture. This might not be that far from the truth since furnishing often have varnished surfaces, which is quite moisture tight. However, in many real situations the backside of furnishing is vapour permeable because it is untreated and in that case it will interact with the vapour content in the surrounding air.

When modelling the moisture transport in the room only one diffusion coefficient for the fluid mixture consisting of fluid wall, air and vapour has been used. This implies that it is not considered that the vapour permeability of the materials influences the vapour diffusion through the materials. Thus the prediction of the surface conditions is not exact. In further developments it would be interesting to take into account diffusion coefficients of the wall materials. Another development of the model could be to implement an approach where the walls are modelled as solids with an inherent diffusion transport because then it would be possible to distinguish between permeable and vapour tight wall materials. Moreover, it would be interesting to simulate other cases with lower air change rate, bigger room and more realistic moisture sources.

CONCLUSION

It was found that the simulations gave reasonable results. This is important, because here the CFD model is used in a non-standardized way (e.g. use of fluid to represent solid wall behaviour) and their correct behaviour in the situation needed to be confirmed.

For all the tested cases the results showed an inlet jet that follows the ceiling and spreads out on the opposite outlet wall except for a case with decreased inlet temperature where the increased density of the air made the jet fall towards the floor. It was found that the airflow patterns in the room are of great importance for the flow direction behind the furniture. Therefore accurate modelling of both inlet and outlet is very important.

The low RH found in the microclimate behind the furniture does not cause moisture problems. But the furniture does influence the moisture levels so for more severe cases this type of investigation can be interesting. It is also found that the model is able to calculate the moisture level and the results seem reasonable. However, the model needs to be validated by comparison with experimental data.

It is a problem that the simulations presented in this paper use the same diffusivity for all fluids. Thereby all materials would give the same results for the moisture levels except that different materials would give different temperatures, which also influences the moisture levels. Thus, this model needs further development.

A disadvantage of using a CFD model is that it is only suitable for steady-state predictions. The explanation is the long computational time needed, so an in-stationary simulation would be too time consuming.

ACKNOWLEDGMENT

The work was carried out with support of the Technical Research Council of Denmark.

REFERENCES

Bartak, B., Beausoleil-Morrison, I., Clarke, J.A., Denev, J., Drkal, F., Lain, M., Macdonald, I.A.., Melikov, A., Popiolek, Z. and Stankov, P., 2002. Integrating CFD and building simulation. *Building and Environment*, **37**, 865-871.

Clarke, J A, Johnstone, C.M., Kelly, N.J., McLean, R.C., Anderson, J.A., Rowan, N.J. and Smith, J.E., 1999. A Technique for the Prediction of the Conditions Leading to Mould Growth in Buildings, *Building and Environment*, **34**, 515-521.

Fluent Inc., 1999. FLUENT® User's Guide, Lebanon, NH, USA: Fluent Inc.

Hohota, R., 2004. Modélisation de l'humidité dans un code CFD (basses vitesses en grande cavité). Comparison avec l'experimental, Ph.D. thesis, Laboratoire CETHIL -l'INSA de Lyon, France.

Negrão, C.O.R., 1995. Conflation of computational fluid dynamics and building thermal simulation. Ph.D. Thesis, University of Strathclyde, Glasgow, UK.

Teodosiu, C., Hohota, R., Rusaouën, G. and Woloszyn, M., 2003. Numerical prediction of indoor air humidity and its effect on the indoor environment. *Building and Environment* **38**, 655-664.

Zhia, Z., Chen, Q., Haves, P. and Klems, J.H., 2002. On approaches to couple energy simulations and computational fluid dynamics programs. *Building and Environment*, **37**, 857-864.

Zhia, Z. and Chen, Q., 2003. Solution characters of iterative coupling between energy simulation and CFD programs. *Energy and Buildings*, **35,** 493-505.

NOMENCLATURE

c0	Face mesh element in room air next to the wall surface (Fig. 1)
c1	Mesh element next to the face mesh element
$m_{w,c0}$	Water vapour content [kg/kg] in c0 (at the cell centre)
$m^{n}_{w,c0*}$	Water vapour content [kg/kg] in c0* at iteration *n*
$d1$	Distance between c1 and c0 [m]
v	Velocity [m/s]
t_{air}	Air temperature [°C]
t_{walls}	Temperature on the exterior side of walls [°C]
TI	Turbulence intensity [%]
d_h	Hydraulic diameter [m]