Air Distribution in Rooms, (ROOMVENT 2000)
Editor: H.B. Awbi

13339

ON THE COMBINED APPLICATION OF THERMAL AND CFD MODELLING IN THE DESIGN OF NATURALLY VENTILATED INDUSTRIAL HALLS

Viktor Dorer [1] and Alois Schälin [2]

[1] EMPA, Swiss Federal Laboratories for Materials Testing and Research,
CH-8600 Duebendorf, Switzerland
[2] Air Flow Consulting, Langmauerstr, 6, CH-8006 Zurich, Switzerland

ABSTRACT

For indoor thermal environment engineering and heating system dimensioning, naturally ventilated spaces impose difficulties due to the interaction of indoor and outdoor air flows and due to their variation in time and space. Thermal building simulation models basically assume mixed air flow conditions in the individual zones, but are able to dynamically model the building masses and the heat exchange between them and the zone air. CFD codes on the other hand can predict the air flow field in a zone in great detail - also in transient cases - but the calculations normally are based on pre-set surface temperature boundary conditions. Therefore, in practice, both dynamic building simulation and CFD calculations are used for proper building design. This is illustrated in this paper in two cases, where both simulation methods were used. The first case is a naturally ventilated test laboratory hall, where overheating risk studies were performed by a combination of thermal modelling and multi-zone air flow modelling, while for winter cases, the draft risk from cold air under the roof glazing was studied using transient CFD calculations. The second case is one of three large central parcel distribution centres, newly built by Swiss Post, where CFD simulations were used to study the inflow of cold air from the truck docking stations. Thermal building simulations where made to provide surface temperatures, especially of the floor, and the overall air flow rates through the building leakage, and to allow for the dimensioning of the heating system. The paper concludes with generally applicable recommendations on the combined use of thermal and CFD modelling.

KEYWORDS

Natural ventilation, industrial ventilation, thermal modelling, transient CFD modelling, applications

INTRODUCTION

Dynamic thermal building simulation is used for the determination of cooling loads and comfort conditions in terms of surface and air temperatures. Temperatures are determined as a functions of

time, for periods up to one year. CFD codes on the other hand can predict the air flow field in a zone i great detail – also in transient cases, but only for time periods in the range of minutes.
Thermal building simulation models basically assume mixed air flow conditions in the individua zones, but are able to dynamically model the building masses and the heat exchange between them an the zone air. CFD codes on the other hand normally are based on pre-set surface temperature boundary conditions. Therefore, in practice, both dynamic building simulation and CFD calculation are used for proper building design. This is illustrated in the two cases described below.

CASE 1: LABORATORY TEST HALL

The building

The large civil engineering test laboratory hall at EMPA (Figure 1) is refurbished. Large parts of the roof are glazed for maximum daylight use. Solid walls are better insulated.

Purpose of the study

The simulation study should give answers to a) overheating risk under summer conditions, b) temperature reduction potential using passive cooling by natural night-time ventilation, c) different ventilation opening control strategies in terms of compliance with thermal comfort requirements and d) risk of draught due to cold air streams falling from the cold roof windows in the winter season.

Figure 1: Picture of the laboratory hall with the glazed roof during retrofit work

Approach

Due to the thermally driven air exchange and the large building masses involved, the problem must be studied using a dynamic thermal building model with an integrated ventilation model. The study was performed for a representative section of the hall. A network model was established for COMIS, considering doors, openings at floor level as well as the large openable ventilation hood on the roof. Relations were established for the air exchange rate in function of the temperatur difference between inside and outside for different opening configurations. The effect of a temperatur gradient in the hall was evaluated additionally. As a conservative approach, wind effects wer neglected. These relations between air exchange rate and temperature difference were then integrate as the ventilation model in the thermal model, the TRNSYS multi-zone type, considering the hall an the room below the thick concrete test floor slab. For the hall, a room model with two air temperatur nodes (one for the occupied zone and one for the rest of the hall) and geometrically detailed radiatio exchange is used.

For the determination of down draught risk in the winter case, 3-dimensional and transient CF computations were performed, using the AEA CFX-TASCflow code. Boundary conditions wer defined from the results of the thermal simulations.

Results

Thermal comfort evaluation

The thermal comfort was evaluated with hourly mean values of the air temperature in the occupie zone, plotted against the maximum 1-h mean outdoor temperature value of the day. Only the perio from April 1st to Oct 30th, and only working hours (7 am to 6 pm) are considered. The minimum a maximum allowable comfort temperatures are adapted to the usual activity and clothing levels of th workers in the hall (see figure 3).

Passive cooling by night-time ventilation and closing the openings if the outside air temperature is higher than the air temperature in the occupant zone have both a significant effect on the resulting peak inside temperatures. The openings should also be closed if the outdoor temperature at night falls below a certain threshold, in order to prevent too low temperatures in the hall in the morning.

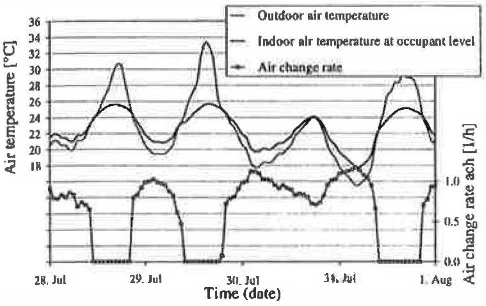

Figure 2: Air change rate, outdoor air temperature (Ta) and room air temperature in the occupied zone (Ti) for a four day summer period, ventilation openings opened 0-24 h, but not if Ti > Ta.

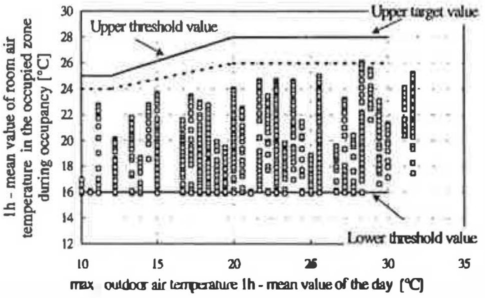

Figure 3: Comfort evaluation with hourly temperature values plotted in relation to max. daily outdoor temperature, and upper and lower limits for acceptable comfort

Winter case

The draught risk due to cold air pillows under the roof glazing, dropping into the occupied zone, was determined by 3-dimensional and transient CFD calculations. The results show a strong intermittent behaviour. As can be seen from figure 4 , velocities do not exceed values over 0.2m/sec. Therefore, the draft risk was assumed to be marginal.

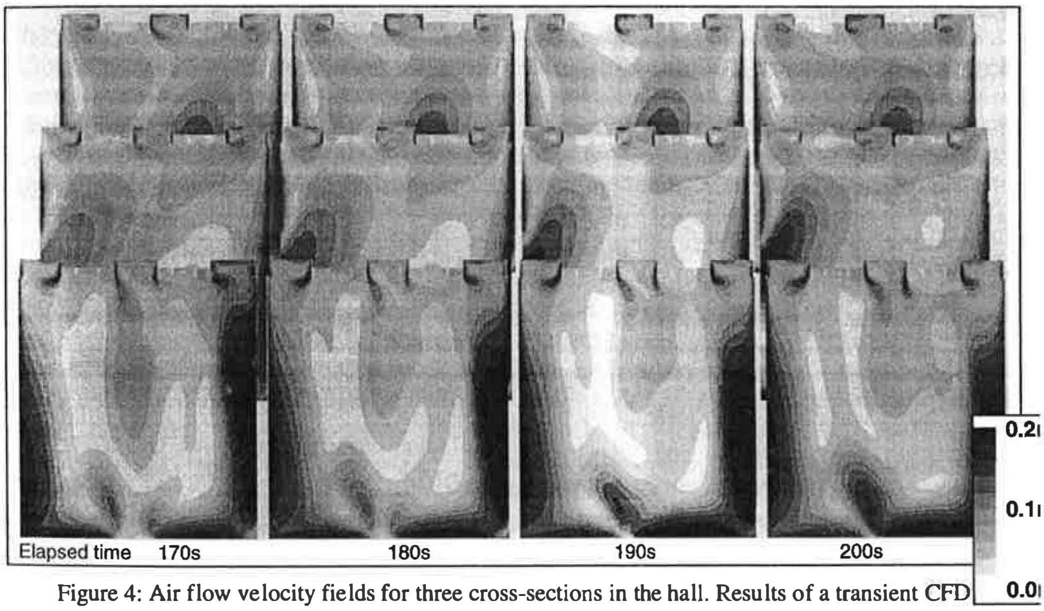

Figure 4: Air flow velocity fields for three cross-sections in the hall. Results of a transient CFD calculation over a time period of 200 sec. Scale 0..0.2m/s

CASE 2: POST PARCEL DISTRIBUTION CENTRE

The building

Schweizer BauJournal 2/99

Schweizer BauJournal 2/99

Figure 5: Outside view of the building with the truck docking stations

Figure 6: Inside view with the distribution lines

The Post parcel distribution centre is a huge building with one single large space with a ground floor area of about 100m by 120 m, and with a height of 20m. About 130 lorry docking doors are distributed around the building facade.

When a lorry is docked, an opening of approx. 0.5 m^2 remains, allowing for air exchange driven by thermal and wind forces. Depending on the docking schedule (up to 80 lorries can be docked at the same time) and the wind situation, significant draught problems may arise in the hall, of course especially in the vicinity of the docking stations.

Purpose of the study

Due to the many docking stations, thermal comfort and draught in winter time was of concern. Specifically, the need for additional air tightening of the docking stations had to be checked. In addition, the radiant heating system had to be dimensioned.

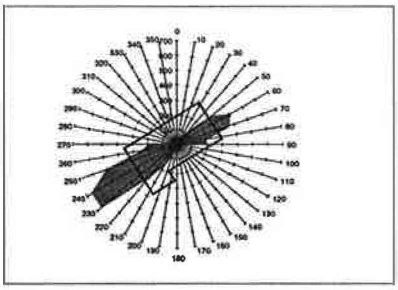

Figure 7: Wind velocity distribution and orientation of building

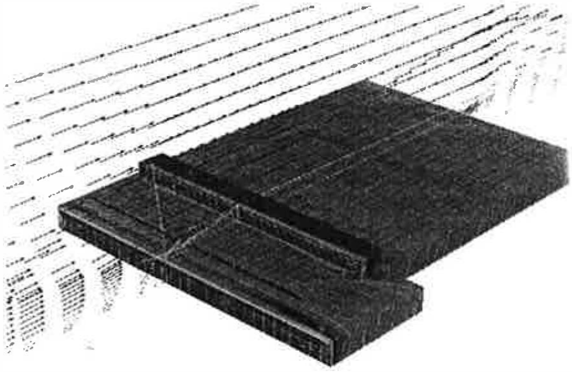

Figure 8: Pressure distribution on the building envelope with wind from South-West (perpendicular to the building front on the left hand side)

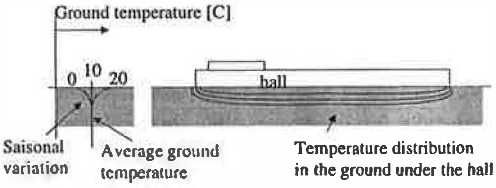

Figure 9: Sketch of temperature distribution in the ground below the hall

Approach

First, critical wind situations were determined (figure 7). For these, pressure distributions on the building facade were determined using CFD (figure 8). The pressure distribution was used for the COMIS natural ventilation network model, which was integrated in the TRNSYS thermal multi-zone model. Individual sections of the hall were modelled as separate zones. With the large floor area, the thermal interaction of floor slab and the ground is also of great importance. This has been studied using a two-dimensional finite-element (FE) heat conduction model. With the thermal simulations, comfort conditions in the docking zones and in the distribution sections could be determined (figure 10). A CFD model of the hall was established with boundary conditions from the thermal simulations. Inside temperatures and velocities were calculated for a few critical cases (figure 11) and thermal comfort and draught risk were evaluated on the basis of PPD values, including mean radiant temperatures.

Results

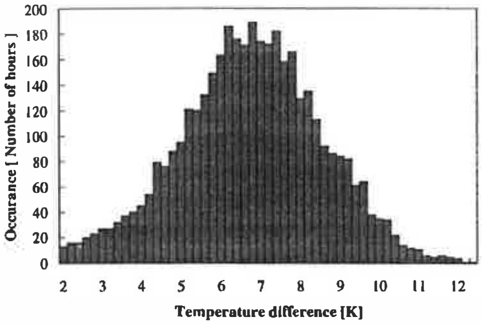

Figure 10: Temporal distribution of the deviations of the temperature near the docking stations in relation to the temperature in the middle of the hall. Results of the TRNSYS-COMIS simulation.

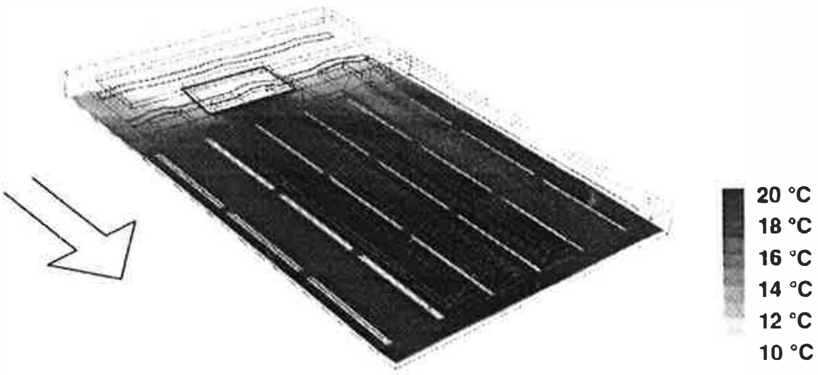

Figure 11: Results of CFD calculation: Air temperature distribution in the hall 1m above ground for a case with wind from West (from left parallel to long facade)

CONCLUSIONS

For the design of naturally ventilated industrial halls, thermal comfort and air quality analysis requires dynamic thermal simulation with an integrated network ventilation model and CFD simulation. Figure 12 shows a possible interaction between these two models.

In order to facilitate the interaction between these codes, development efforts are undertaken by several groups in order to improve the links between CAD, thermal models and CFD models in the form of integrated design software tools.

The dynamic CFD computations in case 1 show a strong intermittent behaviour. Two-dimensional and steady state CFD models produce results which rarely would have reflected the real situation, even in this case with constant boundary conditions. Therefore, draft risk evaluations often require transient CFD simulations.

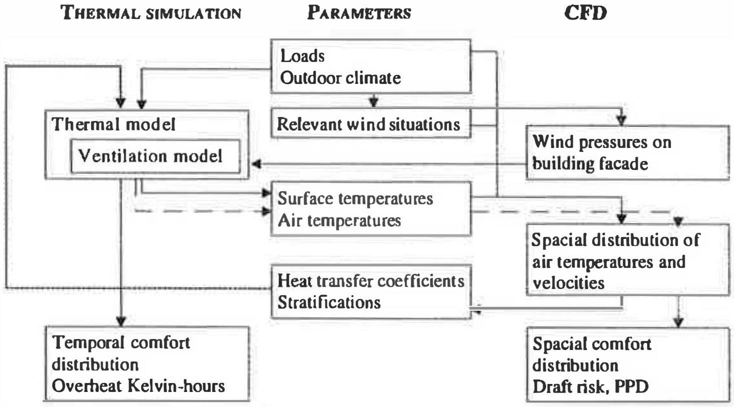

Figure 12: Interaction between thermal, ventilation network and CFD modelling (dashed line: calculation loop)

More information on the application of thermal and CFD simulation tools may also be found in the INVENT Industrial Air Technology Design Guide Book, which is to be published soon.

ACKNOWLEDGEMENTS

The thermal simulations using TRNSYS in the two cases and the finite element heat conduction simulation in case 2 were made by A. Weber, M. Koschenz and R. Weber at EMPA.

REFERENCES

AIVC Technical Note TN 40. (1993). Overview of combined modelling of heat transport and air movement. Air Infiltration and Ventilation Centre, Coventry, GB

Dorer V., Weber A. (1999). Air, contaminant and heat transfer models: Integration and application. *Energy and Buildings* **30**, 97-104

Hensen J.L.M. (1991). *On the thermal interaction of building structure and heating and ventilating system*, Thesis Technical University of Eindhoven, The Netherlands, 1991

Negrão C.O.R. (1998). Integration of computational fluid dynamics with building thermal and mass flow simulation. *Energy and Buildings* **27:2**

Ott F. (1996). *Numerical coupling of air flow, radiation and thermal behaviour of the building*, (in German), Thesis No 11805, Federal Institute of Technology, Zuerich, Switzerland, 1996

Schild P. (1997). *Accurate prediction of indoor climate in glazed enclosures*, PhD thesis 1997:27 , NTNU Trondheim, Norway, ISBN 82-471-0057-6

Schaelin, A: Comfort problems in indoor spaces open to the outdoor environment. *Proceedings Indoor Air 99*, Edinburgh, Scotland.