



AALBORG UNIVERSITY
DENMARK

Aalborg Universitet

Energy Consumption and Indoor Environment Predicted by a Combination of Computational Fluid Dynamics and Building Energy Performance Simulation

Nielsen, Peter Vilhelm

Published in:

Proceedings of Workshop "Integrated Building Simulation", Gliwice-Wisla, 2003

Publication date:

2003

Document Version

Early version, also known as pre-print

[Link to publication from Aalborg University](#)

Citation for published version (APA):

Nielsen, P. V. (2003). Energy Consumption and Indoor Environment Predicted by a Combination of Computational Fluid Dynamics and Building Energy Performance Simulation. In Baranowski, A. (Ed.), Proceedings of Workshop "Integrated Building Simulation", Gliwice-Wisla, 2003 (pp. 35-46). Aalborg : Silesian University of Technology.

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- ? Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- ? You may not further distribute the material or use it for any profit-making activity or commercial gain
- ? You may freely distribute the URL identifying the publication in the public portal ?

Take down policy

If you believe that this document breaches copyright please contact us at vbn@aub.aau.dk providing details, and we will remove access to the work immediately and investigate your claim.



**PROCEEDINGS OF WORKSHOP
"INTEGRATED BUILDING SIMULATION"**

GLIWICE - WISŁA 2003

**edited by
Andrzej BARANOWSKI**

***CENTRE FOR ENERGY EFFICIENT TECHNOLOGIES AND SYSTEMS
IN INDOOR ENVIRONMENT ENGINEERING***

www.ener-indoor.pl

email: kowito@kowito.ise.polsl.gliwice.pl

ENERGY CONSUMPTION AND INDOOR ENVIRONMENT PREDICTED BY A COMBINATION OF COMPUTATIONAL FLUID DYNAMICS AND BUILDING ENERGY PERFORMANCE SIMULATION

Peter V. Nielsen, Aalborg University, Sohngaardsholmsvej 57, 9000 Aalborg, Denmark

ABSTRACT

An interconnection between a building energy performance simulation program and a Computational Fluid Dynamics program (CFD) for room air distribution is introduced for improvement of the predictions of both the energy consumption and the indoor environment.

The article describes a calculation of the energy consumption in a large building where the building energy simulation program is modified by CFD predictions of the flow between three zones that are connected by pressure and buoyancy-driven air flow through open areas. The two programs are interconnected in an iterative procedure. The article shows also an evaluation of the air quality in the main area of the buildings based on CFD predictions.

It is demonstrated that an interconnection between a CFD program and a building energy performance simulation program will improve both the energy consumption data and the prediction of thermal comfort and air quality in a selected area of the building.

KEY WORDS

Air flow pattern, Air quality, CFD, Energy, Ventilation efficiency, Building energy performance simulation.

INTRODUCTION

An interconnection between a building energy performance simulation program and a CFD program for room air distribution will improve the predictions of both the energy consumption and the indoor environment.

In order to predict the air distribution in a room it is necessary to have a description of the boundary conditions as e.g. surface temperature or heat transmission. This is difficult in situations where radiation and free convection are important but the boundary conditions can be obtained by a building energy performance simulation program that takes account of radiation, conduction in the structure and transient responses of the building.

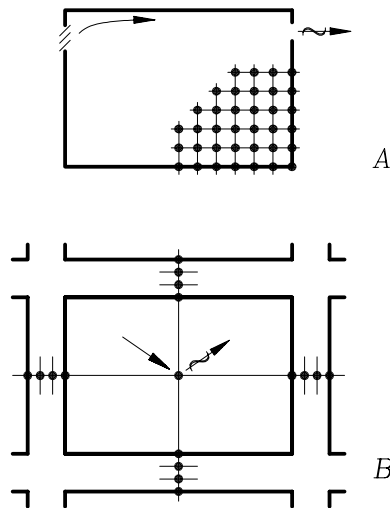


Figure 1. Principles for Computational Fluid Dynamics (A), and building energy performance simulation (B).

Figures 1A and 1B show the characteristics of a CFD program and a building energy performance simulation program. The CFD program predicts the flow in a room by solving all the flow equations, including the energy transport equation, in a grid covering the volume of the room, see Figure 1A. The solution domain is surrounded by boundary conditions such as air terminal devices, return openings, surface temperatures or energy flow at the surfaces. The building energy performance simulation program, see Figure 1B, often describes the energy flow in the room air movement by a single grid point. The energy flow in the walls, the floor and the ceiling is described by a grid number sufficient to predict the detailed dynamic energy flow and the consumption of the whole building during a period.

Figures 1A and 1B indicate the improvements that can be obtained by an interconnection between the two programs. The building energy performance simulation program is able to predict the temperature distribution, or energy flow, through all the surfaces in the building, and the found values are used as boundary conditions for the CFD program. The CFD program is, on the other hand, able to predict the air flow in the rooms of the building and thereby give an improved description of the energy flow between the surfaces of the building.

The method can be structured in different ways. A building energy performance simulation program can be connected to a separate CFD program where the CFD program makes the prediction of the energy flow in the selected situation. It is also possible to work with a CFD program that is extended to include the possibility of finding a combined solution of radiation, conduction and thermal storage, parallel to the CFD solution of the flow field. This model is often called a conjugate heat transfer model. Another possibility is to add a CFD code to be used in selected rooms to extend a large building energy performance simulation program.

Some of the earliest publications based on the combined use of CFD and building energy performance simulation were given by Chen [1] and Holmes et al. [3]. New examples of conjugate heat transfer models are given by Moser et al. [8], Kato et al. [4] and Schild [9].

This article describes an energy and indoor air quality study of the Central Library of North Jutland. The combination of the two programs is used, especially to predict the energy flow in a space divided into several rooms with open connections and, furthermore, to study the indoor air quality in the main hall.

Building energy performance simulation

The Central Library of North Jutland consists of 170 rooms ventilated partly by a CAV system partly by a VAV system.

Radiator heating is used in most of the rooms, but the main hall is heated by an air conditioning system.



Figure 2. Library hall and openings to two other lending departments.

The main library hall is connected by large openings to two other rooms as shown in Figure 2. Figure 3 shows the library hall as zone 1 (Z1), and the two other rooms as zone 2 (Z2) and zone 3 (Z3). The figure shows a vertical connection between zone 2 and zone 3 and horizontal connections between zone 1 and zone 2 and between zone 1 and zone 3 as well. All three zones are equipped with supply and return openings for individual air distribution in the different spaces. Pressure and temperature differences induce an air movement between the three zones, and this movement will in certain situations be supported by thermal flow from radiators located in zone 2 below the vertical openings to zone 3.

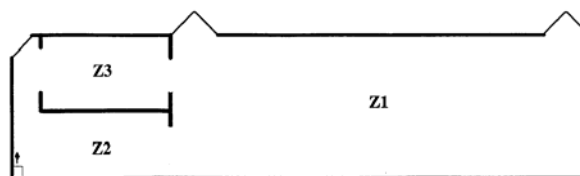


Figure 3. The library hall, Z1, and two other zones, Z2 and Z3, with open connections to each other.

A building energy performance program called tsbi3, developed by the Danish Building Research Institute, calculates the energy consumption of the building. Benchmark tests of this program and of

several other programs are given by Lomas et al. [5].

Rooms with identical load profiles, identical heating and ventilation principles and identical orientation of windows with identical window to floor ratio are grouped in a single zone, and 170 rooms are simulated as a total of 27 zones. The simulation program is not able to predict the flow between the zones 1, 2 and 3 as a function of the pressure and temperature distribution in the zones, and as an initial guess it is assumed that the energy flow between the zones is low and therefore unimportant.

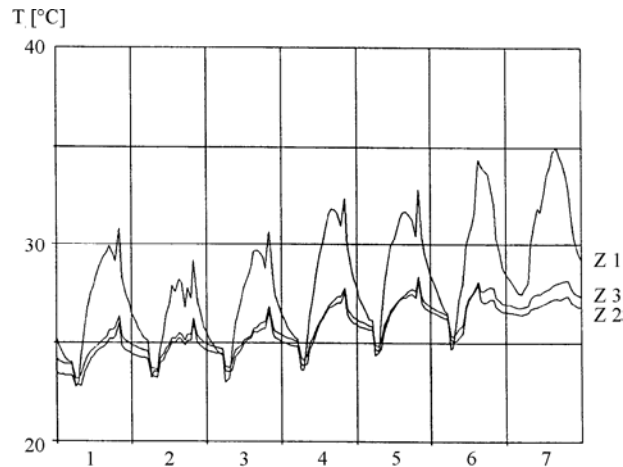


Figure 4. The temperature development in the zones 1, 2 and 3 during week 23. It is assumed that there is no air exchange between the zones.

Figure 4 shows the development of temperatures in the three zones during a week in June (week 23) when the building is loaded according to the Danish reference year. The library hall, zone 1, is heated by solar radiation through skylights, and in certain periods it will obtain a temperature level that is 8°C above the temperature in the surrounding zones. It is obvious that this temperature difference will induce an energy flow between the zones and, therefore, it is necessary to extend the tsbi3 program to include information on the air movement in the open connections in the building.

Building energy performance simulation combined with Computational Fluid Dynamics

This chapter shows the possibilities that can be obtained by performing a combined simulation with tsbi3 and the CFD program FLOVENT.

It is very time-consuming to run a CFD program and it is therefore necessary to study the possibilities of simplified predictions of the energy flow. In some situations it is possible to obtain good results in a two-dimensional geometry, see e.g. Nielsen [6], but in the case described in this article it is both necessary to preserve the three-dimensional effect of the flow around the openings and to work with the actual vertical dimensions. Figure 5 shows the simplified three-dimensional geometry that is selected for the prediction of the energy flow. The geometry represents a compromise with correct geometry around the openings. The ratio between the total flow area and the floor area of the three zones is the same in the CFD prediction and in full scale in the building. Air supply and return openings are only defined in a coarse grid sufficient for momentum and energy flow predictions but insufficient for a detailed analysis of the air velocity distribution in the occupied zone.

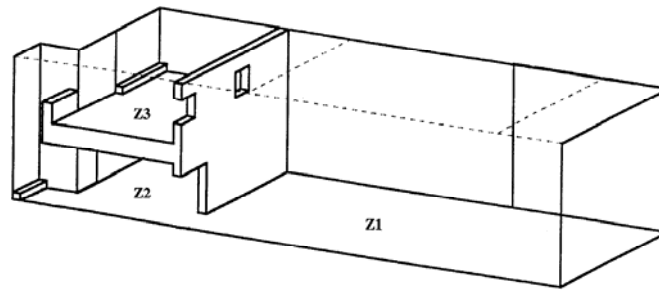


Figure 5. Three-dimensional CFD model for prediction of air exchange between the zones Z1, Z2 and Z3.

It is not only necessary to simplify the geometry in the CFD predictions it is also necessary to restrict the number of cases during the reference year in which the CFD predictions are performed. Interesting situations are both cases where the temperature difference is large between the zones 1, 2 and 3 because they may represent large energy flow between the zones and cases where the load and the system will generate vertical temperature gradients. Those are situations that can be handled by the CFD program but the energy simulation program may be unable to predict the situation correctly because the flow regime is only represented by one grid point, which is insufficient to represent flow in openings and a vertical temperature gradient.

Three cases are selected because they represent the above-mentioned problem. The first case (Case A) is the situation on the 7th January at 12 p.m. The ventilation in the library hall (zone 1) is closed down at night and over the weekend to save energy, whereas the two other zones (zone 2 and zone 3) are heated and they will add an energy flow to the library hall.

The second case (Case B) is the situation on the 8th January at 11 a.m. The ventilation system in the library hall (zone 1) is reheating the hall after the setback. The surface and the construction in the hall are cold and in this situation a vertical temperature gradient may be obtained.

The third and last case (Case C) is the situation on the 6th June at 4 p.m. The temperature in the library hall will be very high due to solar radiation and large temperature differences will be obtained between zone 1, zone 2 and zone 3 as shown in Figure 4.

In all three cases the boundary conditions for the energy flow are obtained by the tsbi3 program. They are based on given values of the heat transfer coefficient. The boundary conditions could also be given as a temperature distribution, but it is well known that it is difficult to make an accurate prediction of the heat transfer coefficient based on the wall functions in a CFD program, see Chen [2] and Nielsen [7].

It is necessary to obtain the final solution by an iteration between tsbi3 and the CFD program because a change in the air flow between the three zones will change the temperature level and cause a change in the energy flow in the surrounding surfaces.

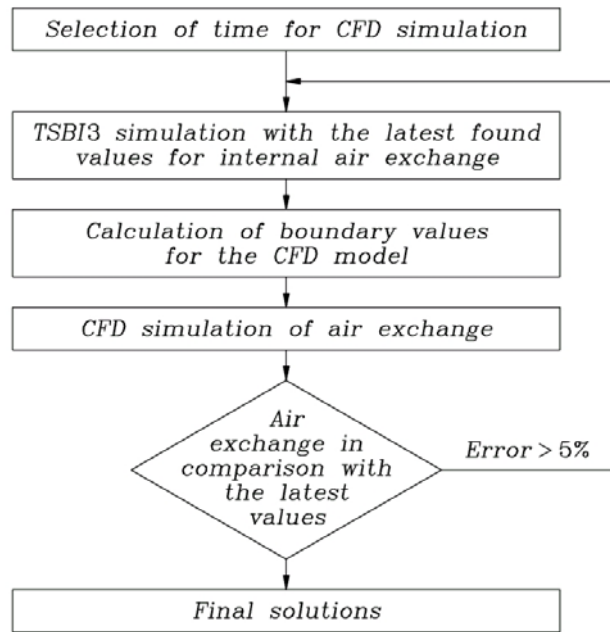


Figure 6. Flow chart that shows the iterations between the building energy performance simulation program and the Computational Fluid Dynamics program.

Figure 6 shows the stages in this iteration. The initial tsbi3 predictions are made without air exchange between the zones 1, 2 and 3. The corresponding CFD predictions give an air exchange that is introduced into the tsbi3 program. The new energy flow at the surfaces is introduced into the CFD program, which will result in a new air exchange between the three zones. The iterations between the two programs continue till the change in the air flow is below 5% compared with the latest values. Similarity between the predictions from the tsbi3 program and the CFD program will often be obtained after 3 - 6 iterations.

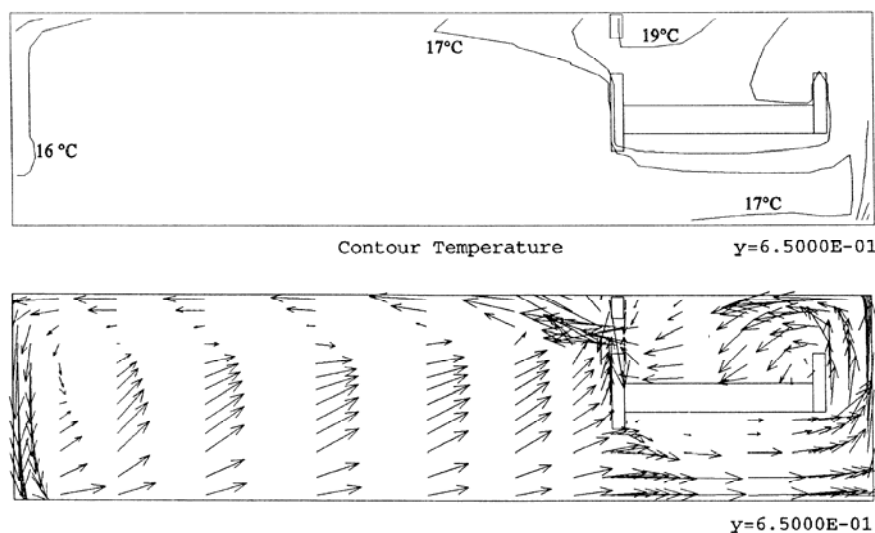


Figure 7. CFD predictions of temperature distribution and air movement between the zones 1, 2 and 3 in Case A, 7th January at 12 p.m.

Figure 7 shows the CFD prediction of the flow during night setback in the library hall (Case A). There is an air flow, and therefore also an energy flow, between the zones 1, 2 and 3. The floors in zone 1 and

zone 2 are cooled down at a similar rate, which causes the same temperature in both rooms at the level of the occupied zone. The radiators in zone 2 generate a flow upwards in the opening between zone 2 and zone 3, and the room zone 3 is always heated to the set value. The instantaneous heat output from the radiators in zone 1 is 22% larger than the value obtained if the energy flow is ignored between the three zones.

Cold draught from ceiling and walls and heating from the floor create a large mixing flow in the library hall. This flow will restrict any tendency to a formation of a vertical temperature gradient and complete agreement between the results from the two programs is obtained after three iterations.

A vertical temperature gradient will be obtained in case of reheating and it will not be possible to reach a converged solution for both programs when the solution is based on identical heat flow through all surfaces. Case B, 8th January at 11 a.m., shows this situation. The CFD program predicts a vertical temperature gradient that gives a heat flow at the floor, which is opposite to the heat flow found from the tsbi3 program. In this case the iteration loop is changed. The predicted floor temperature in zone 1 and zone 2 is transferred from tsbi3 to the CFD program as boundary conditions instead of the heat flow and the supply temperature in zone 1 is restricted to give a room temperature of 21.5°C (set point temperature).

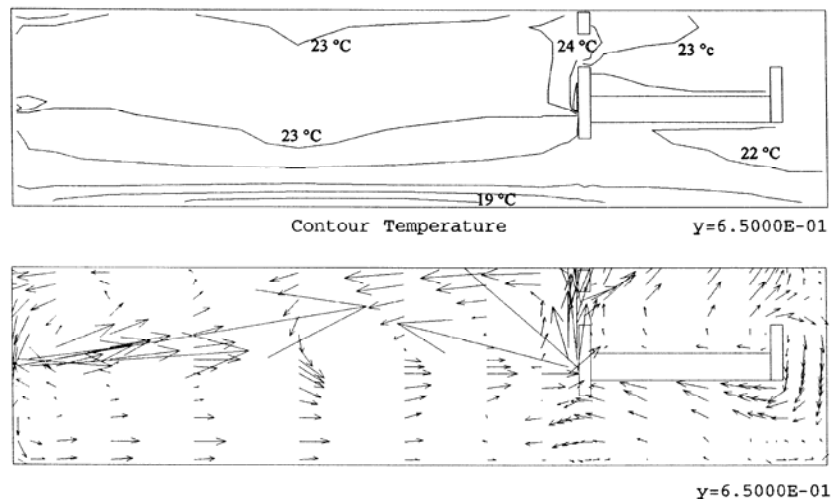


Figure 8. CFD predictions of temperature distribution and air movement between the zones 1, 2 and 3 in Case B, 8th January at 11 a.m.

Figure 8 shows the final temperature and velocity distribution in Case B. A vertical temperature gradient (4°C from floor to ceiling in the library hall) is present, and it must be concluded that an air heating system with supply opening above the occupied zone is not a perfect solution in situations like this where reheating of the hall takes place after night setback.

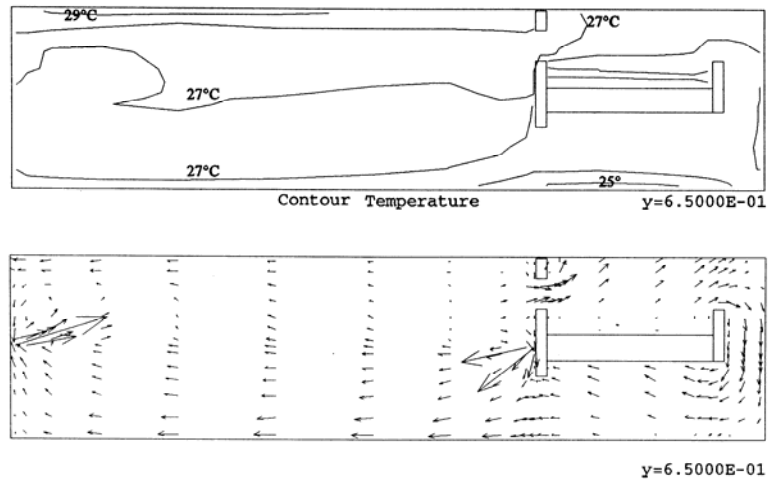


Figure 9. CFD predictions of temperature distribution and air movement between the zones 1, 2 and 3 in Case C, 6th June at 4 a.m.

Figure 9 shows the air distribution found by the CFD program the 6th June at 4 p.m., Case C. The temperature in the library is high due to solar radiation. The highest temperature is found in zone 1 and the lowest in zone 2. The air is flowing from zone 1 into zone 3 and further into zone 2. The flow reduces the temperature in the library hall, zone 1, and increases the level in zone 3.

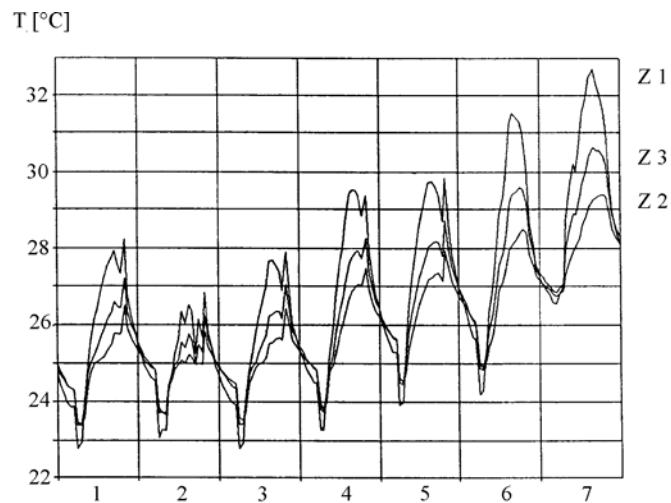


Figure 10. The temperature development in the zones 1,2 and 3 during week 23. The air exchange between the zones is obtained by CFD predictions.

Figure 10 shows the development of the temperatures in the three zones during week 23 if the air exchange is considered in the tsbi3 program. The air flow between the zones causes a decrease in the temperature difference between the three zones as well as a decrease in the temperature level in the library hall.

The highest temperature difference between any of the zones is only 3°C compared with 8°C found without air exchange between the zones, see Figure 4. It must be assumed that by considering the air exchange between the different open zones in the building the quality of the energy consumption calculations will be improved.

The building energy performance program is finally used for the calculation of the yearly heat consumption in the case where flows between zones 1, 2 and 3 are considered. These flows should have been given as a function of the temperature difference, which is not possible in the tsbi3 program. A constant level of the flow, according to the Cases A, B and C, is distributed over the year (A and B in the heating season and C during the summer period). A calculation with these values gives all things considered an energy consumption that is 6.4% larger than the value obtained without flow between the zones 1, 2 and 3. Most of the increased energy consumption is due to the vertical temperature gradient that is simulated by a slightly increased set value in the library hall. Only 1% of the increased energy consumption is a direct result of the flow between the zones 1, 2 and 3.

CFD prediction of air distribution and ventilation efficiency

This chapter describes an evaluation of the air quality in the library hall based on CFD predictions. The predictions are made in a geometry that corresponds to the real layout of the library hall, including bookcases in the occupied zone but without the zones 2 and 3. Flow rates and temperatures through vertical surfaces in the hall introduce the effect of these zones. One of the important aspects is the simulation of air terminal devices that consist of circular openings with a diameter of 16 cm. Figure 2 shows the supply opening in the wall above the occupied zone and the return opening just below the ceiling close to the skylights.

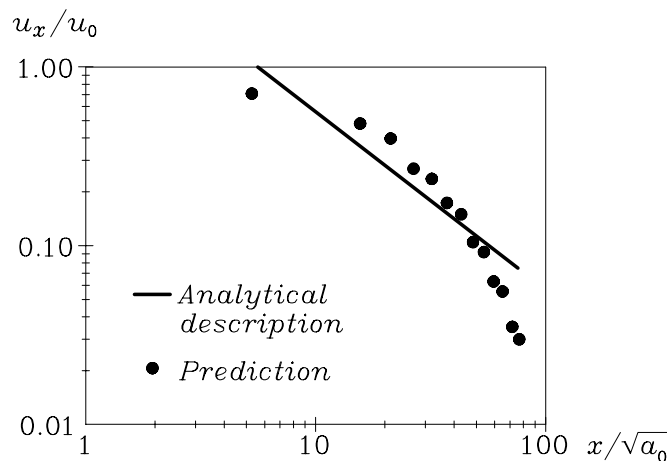


Figure 11. Predicted and calculated velocity decay in a single jet without disturbance from the surroundings.

A special set of predictions, with focus on the flow from a supply opening, has been made to study the necessary number of grid points in the velocity decay zone from an opening. Figure 11 shows the velocity decay from the opening. It is seen that the predictions give a reasonable description of both the momentum flow and the velocity decay up to a distance of $60\sqrt{a_0}$ (8.5 m).

The boundary conditions for the energy flow and the flow in connection with the zones 2 and 3 are given from the combined predictions mentioned in the previous chapter.

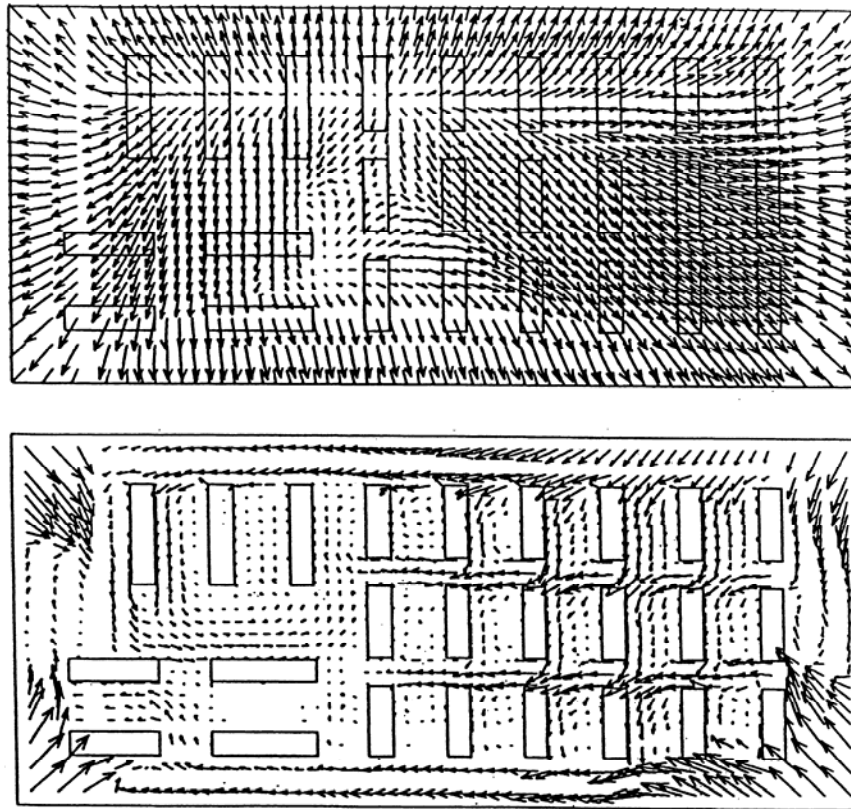


Figure 12. Air movement in two horizontal sections of the library hall on the 8th January. The upper figure shows the predictions at a height of 7 m, and the lowest figure shows the predictions at floor level (0.25 m).

Indoor air quality problems are observed during the winter because the library hall is heated by the ventilation system and the supply air seems to bypass the occupied zone due to the buoyancy effect. Figure 12 shows the air distribution in the hall on the 8th of January at 11 a.m. (corresponding to Case B). Two horizontal sections are shown, one section just below the ceiling and one section 25 cm above the floor. It is obvious that the heated jets ($4.56 \text{ m}^3/\text{s}$, 32.5°C) move upwards and generate a radial flow below the ceiling. Some air will move down into the corners of the hall because three of the corners are without supply openings. The flow in the top right-hand corner of the lower figure is the flow from zone 2, which in this situation will supply air to the occupied zone ($2.07 \text{ m}^3/\text{s}$, 18.6°C). The flow in the bottom right-hand corner of the figure is partly cold downdraught from a glazed area in this corner of the hall, and partly induced flow from the upper part of the building. Figure 12 shows that the velocity level is very low in the main part of the occupied zone, which corresponds to low ventilation efficiency in this area of the hall.

The concentration distribution has been predicted for a situation where the low surfaces in the hall are the sources. The maximum concentration in the occupied zone, c_p / c_R , has a level of 2.05, which corresponds to a local ventilation index, ε_p , of 0.49 (c_p is the concentration in the air and c_R is the mean concentration in the return openings). The average concentration in the occupied zone, c_{op} / c_R , is 1.3 corresponding to a mean ventilation efficiency of 0.77. This is not a high value, and it must be considered that recirculation in the ventilation system will cause a further decrease in the level of fresh air in the occupied zone.

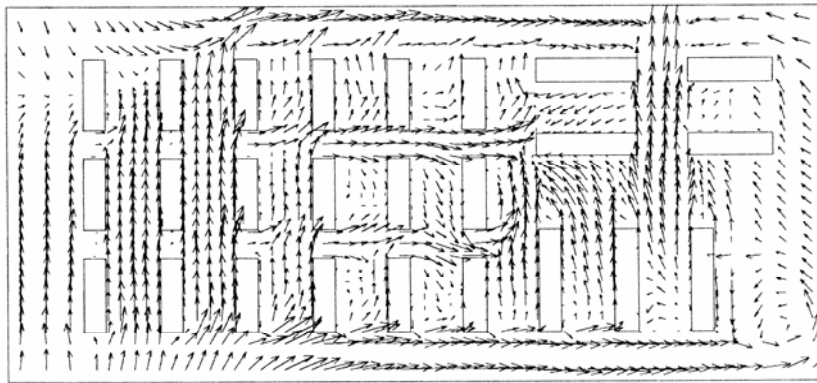


Figure 13. Air movement in the library hall on the 6th June. The figure shows the predictions at floor level (0.25 m).

Figure 13 shows the velocity distribution in the occupied zone of the library hall (velocity at the height of 0.25 m) the 6th June at 4 p.m. (corresponding to Case C). The velocities have the level of 0.3 m/s everywhere in the occupied zone. The concentration distribution is $c_{op} / c_R = 1.12$ corresponding to a better mixing of the contaminant. The mean ventilation efficiency is 0.89, which is a reasonable level for a mixing ventilation system.

CONCLUSIONS

An interconnection between a building energy performance simulation program and a CFD program for room air distribution is introduced to obtain an improvement of the predictions of both the energy consumption and the indoor environment.

The building energy performance simulation program in a large building is modified by CFD predictions of the flow between three zones connected by pressure and buoyancy-driven air flow through open areas. The two programs are interconnected in an iterative procedure.

It is shown that the interconnection between the CFD program and the building energy performance simulation program will improve the calculation of dynamic temperature development in a room as well as the prediction of thermal comfort in a hall taking the flow from neighbouring zones into consideration.

ACKNOWLEDGEMENT

Tryggvi Tryggvason is acknowledged for the participation in this work and for discussion of results and conclusions.

REFERENCES

- [1] Chen, Q. (1988) Indoor Airflow, Air Quality and Energy Consumption of Buildings. Ph.D. Thesis, Delft University of Technology, the Netherlands.
- [2] Chen, Q. and Jiang, Z. (1992) Significant Questions in Predicting Room Air Motion. *ASHRAE Transactions*, Vol. 98, Pt. 1, pp. 929-939.
- [3] Holmes, M.J., Lan, J.K-W., Ruddick, K.G. and Whittle, G.E. (1990) Computation of

Conduction, Convection and Radiation in the Perimeter Zone of an Office Space. *Proceedings ROOMVENT '90*, Oslo, Norway.

- [4] Kato, S., Murakami, S., Shoya, S., Hanyu, F. and Zeng, J. (1995) CFD Analysis of Flow and Temperature Fields in Atrium with Ceiling Height of 130 m. *ASHRAE Transactions*, Vol. 101, Part 2, pp. 1144-1157.
- [5] Lomas, K., Eppel, H., Martin, C. and Bloomfield, D. (1994) Empirical Validation of Thermal Building Simulation Programs using Test Room Data. Vol. 1: Final Report, IEA BCS, Annex 21 & IEA SHC Task 12.
- [6] Nielsen, P.V. (1995) Airflow in a World Exposition Pavilion Studied by Scale-Model Experiments and Computational Fluid Dynamics. *ASHRAE Transactions*, Vol. 101, Pt. 2, pp. 1118-1126.
- [7] Nielsen, P.V. (1998) The Selection of Turbulence Models for Prediction of Room Airflow. *ASHRAE Transactions*, Vol. 104, Pt. 1.
- [8] Moser, A., Off, F., Schälin, A. and Yuan, X. (1995) Numerical Modeling of Heat Transfer by Radiation and Convection in an Atrium with Thermal Inertia. *ASHRAE Transactions*, Vol. 101, Part 2, pp. 1136-1143.
- [9] Schild, P. (1997) Accurate Prediction of Indoor Climate in Glazed Enclosures. Ph.D. Thesis, Dep. of Refrigeration and Air Conditioning, Norwegian University of Science and Technology (NTNU), Norway.