Computational Fluid Dynamics and Building Energy Performance Simulation

Nielsen, Peter Vilhelm; Tryggvason, T.

Published in:
Proceedings of ROOMVENT '98 : Sixth International Conference on Air Distribution in Rooms, Stockholm, Sweden, June 14-17, 1998

Publication date:
1998

Document Version
Publisher's PDF, also known as Version of record

COMPUTATIONAL FLUID DYNAMICS AND BUILDING ENERGY PERFORMANCE SIMULATION

Peter V. Nielsen[1], Tryggvi Tryggvason[2]

[1]Aalborg University, Aalborg, Denmark
[2]Opus ehf, Akureyri, Iceland

ABSTRACT

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

The building energy performance simulation program requires a detailed description of the energy flow in the air movement which can be obtained by a CFD program.

The paper describes an energy consumption calculation in a large building, where the building energy simulation program is modified by CFD predictions of the flow between three zones connected by open areas with pressure and buoyancy driven air flow. The two programs are interconnected in an iterative procedure.

The paper shows also an evaluation of the air quality in the main area of the buildings based on CFD predictions.

It is shown 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.

It is necessary to have a description of the boundary conditions as e.g. surface temperature or heat transmission when the air distribution is predicted in a room. This is especially necessary in constructions where radiation and free convection are important. These boundary conditions can be obtained by a building energy performance simulation program which takes account of radiation, conduction in the structure and transient responses of the building.

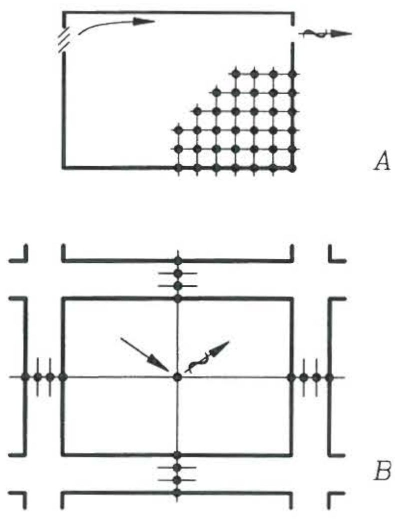


Figure 1 Principles for Computational Fluid Dynamics (A), and building energy performance simulation (B).

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 with a given indoor and outdoor load.

Figures 1A and 1B indicate the improvements which 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 values found 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 selected situation. It is also possible to work with a CFD program which is extended to include the possibility of finding a combined solution of radiation, conduction and thermal storage, in parallel with the CFD solution of the flowfield. This model is often called a conjugate heat transfer model. Another possibility is to extend a large building energy performance simulation program with a CFD code to be used in selected room.

Some of the earliest publications based on the combined use of CFD and building energy performance simulation were given by Chen (1988) and Homes et al. (1990). New examples of conjugate heat transfer models are given by Moser et al. (1995), Kato et al. (1995) and Schild (1997).

One of the main reasons for the use of combined models or a conjugate heat transfer model is, as mentioned earlier, to obtain a good description of the energy flow at surfaces and between rooms in different situations.

This paper describes an energy and indoor air quality study of the Central Library of North Jutland. Combined programs are 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.

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

Most of the rooms are heated by radiators, but the main hall is heated by the air conditioning system.

The main library hall is connected with

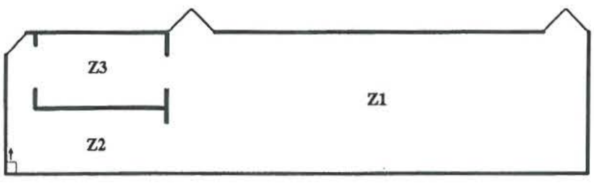


Figure 3 The library hall, Z1, and two other zones, Z2 and Z3, with open connections to each other.

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, 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.

The energy consumption of the building is calculated by a building energy performance program called tsbi3, developed by the Danish Building Research Institute. Benchmark tests of this program and of several other programs are given by Lomas et al. (1994).

Rooms with identical load profiles, identical heating and ventilation principle and identical orientation of windows with equal 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 Z1, Z2 and Z3 as a function of the pressure and temperature distribution in the zones, and it is assumed, as an initial guess, that the energy flow between the zones is low and therefore unimportant.

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 it will in certain periods obtain a temperature level which 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 it is therefore necessary to extend the tsbi3 program to include information on the air movement in the open connections in the building.

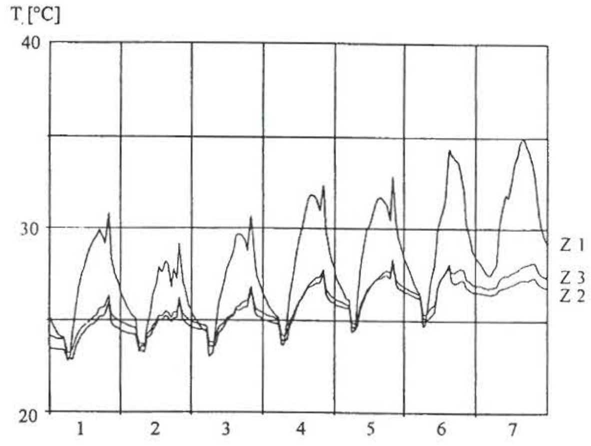


Figure 4 The temperature development in the zones 1, 2 and 3 during week 23. It is assumed that there is no air exchange between the zones.

Building energy performance simulation combined with Computational Fluid Dynamics

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

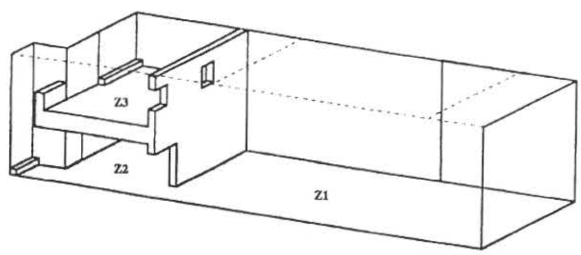


Figure 5 Three-dimensional CFD model for prediction of air exchange between the zones Z1, Z2 and Z3.

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 (1995), but in this case 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 which 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 energy flow predictions but insufficient for a detailed analysis of the air velocity distribution in the occupied zone.

The CFD simulation is made on the fourth day of week 23 at four o'clock in the afternoon. The boundary conditions for the energy flow are obtained by the tsbi3 program and they are based on given values of the heat transfer coefficient. The boundary conditions could also be given as a temperature distribution, but is well known that it is difficult to make an accurate prediction of the heat transfer coefficient with the wall functions in a CFD program, see Chen (1992) and Nielsen (1998).

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 shows the stages in this iteration. The initial tsbi3 predictions are made without air exchange between the zones Z1, Z2 and Z3. The corresponding CFD predictions give an air exchange which is introduced into the tsbi3 program. The new energy flow at the surfaces is introduced into the CFD program resulting in a new air exchange between the three zones. The iterations continue till the change in the air flows is below 5 % compared with the last values. Different situations during the reference year are selected for this analysis, and similarity between the predictions from the tsbi3 program and the CFD program has been obtained after 3 - 6 iterations.

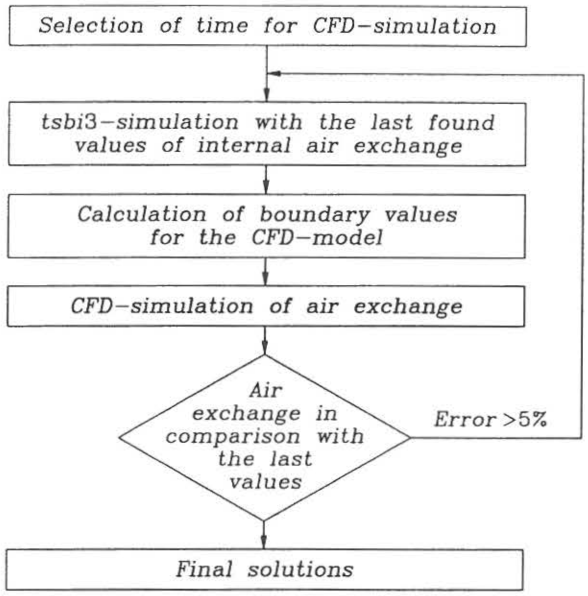
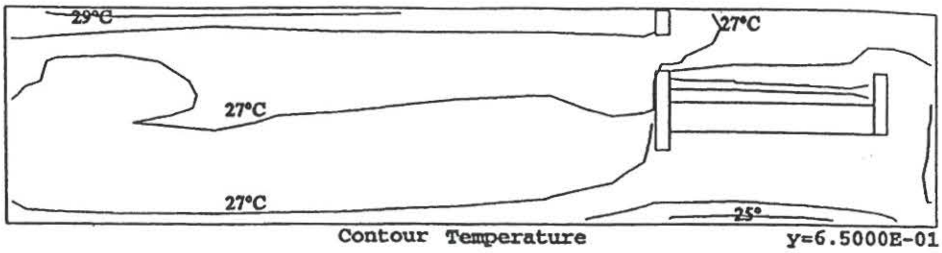


Figure 6 Flow chart which shows the iterations between the building energy performance simulation program and the Computational Fluid Dynamics program.

Figure 7 shows the air distribution found by the CFD program. 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 8 shows the development of the temperatures in the three zones during week 23 if the air exchange is considered. 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



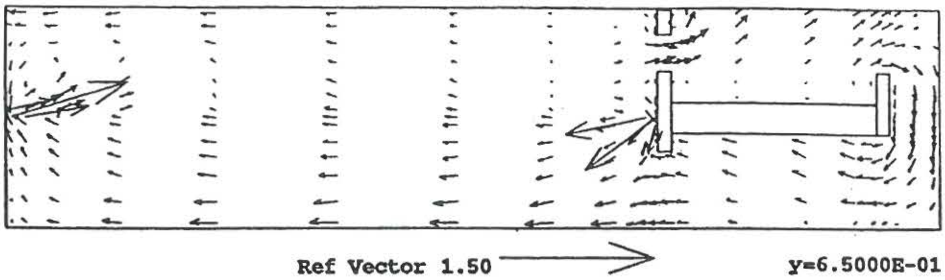


Figure 7 CFD predictions of temperature distribution and air movement between the zones 1, 2 and 3 on the fourth day of week 23 at four in the afternoon.

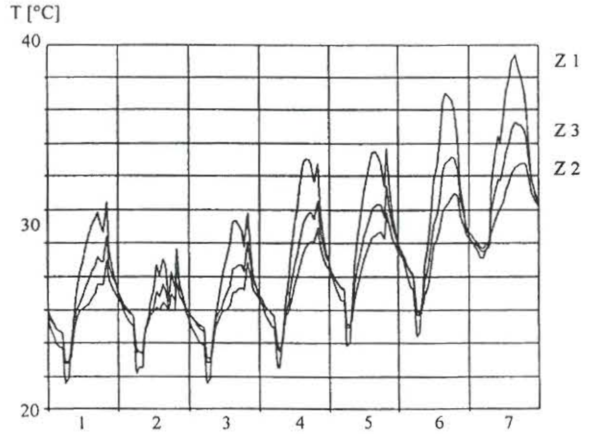


Figure 8 The temperature development in the zones 1,2 and 3 during week 23. The air exchange between the zones is obtained by CFD predictions.

must be assumed that the energy consumption calculations of the highest quality are obtained by considering the air exchange between the different open zones in the building.

Figure 7 shows that the vertical temperature gradient in the rooms is small. Larger gradients can be obtained in other situations where it will be difficult to obtain similarity between predictions made by the two programs. The building energy performance simulation program is only able to cope with situations without vertical gradients because the room air volume is described by one grid point, see Figure 1B. A zonal model with more vertical zones in a room will be necessary in this situation.

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 prediction is made in a geometry which corresponds to the real layout of the library hall, including bookcases in the occupied zone but without the zones 2 and zone 3. The effect of these zones is introduced by flow rates and temperatures through vertical surfaces in the hall. One of the important aspects is the simulation of air terminal devices which 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. 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. It was possible to obtain a sufficient description of both supply openings, bookcases and the flow in the occupied zone by 11400 grid

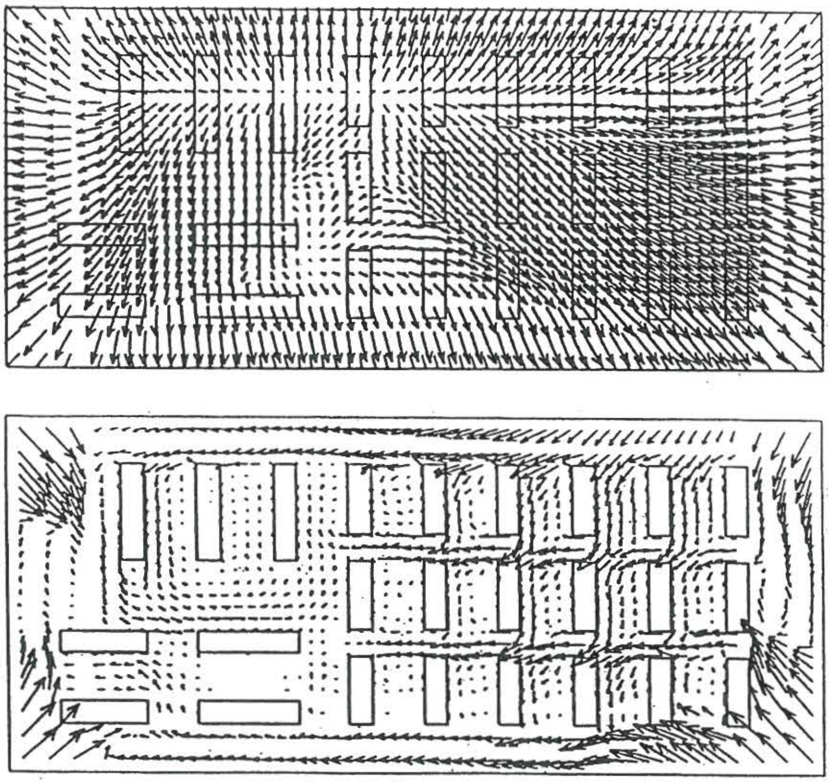


Figure 9 Air movement in two horizontal sections of the library hall. The upper figure shows the predictions at a height of 7 m, and the lowest figure shows the predictions at floor level (0.25 m).

points.

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 last chapter.

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 9 shows the air distribution in the hall on the 8th of January at eleven o'clock in the morning. 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 m^3/s, 32.5°C) move upwards and generate a radial flow below the ceiling. Some air will move down in the corners of the hall because three of the corners are without supply openings. The flow in the top right-hand corner of the figure is the flow from zone 2, which in this situation will supply air to the occupied zone (2.07 m^3/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 9 shows that the velocity level is very low in the main part of the occupied zone, which corresponds to a 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 the 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 further decrease the level of fresh air in the occupied zone.

CONCLUSIONS

An interconnection between a building energy performance simulation program and a CFD program for room air distribution is introduced, and it can be concluded that improvement of the predictions of both the energy consumption and the indoor environment can be achieved.

The building energy performance simulation program in a large building is modified by CFD predictions of the flow between three zones connected by open areas with pressure and buoyancy driven air flow. 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.

REFERENCES

Chen, Q. (1988) Indoor Airflow, Air Quality and Energy Consumption of Buildings. Ph.D. Thesis, Delft University of Technology, the Netherlands.

Chen, Q. and Jiang, Z. (1992) Significant Questions in Predicting Room Air Motion. *ASHRAE Transactions*, Vol. 98, Pt. 1, pp. 929-939.

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.

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.

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.

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.

Nielsen, P.V. (1998) The Selection of Turbulence Models for Prediction of Room Airflow. *ASHRAE Transactions*, Vol. 104, Pt. 1.

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.

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.