Numerical simulation of air flow patterns in auditoria with respect to various ventilation layouts and room geometries

Finn Drangsholt, Prof, dr.ing

Abstract

Computational fluid dynamics have been used to examine the airflow pattern in auditoria with various ventilation layouts and room geometry.

The simulations have shown that the airflow pattern inside this type of premises may be quite complex. The air flow pattern is affected by room geometry, capacity and location of heat sources, aspects of building material and construction, furnishing, ventilation principle, supply air temperature and surrounding climate.
A general trend from the simulations of terraced premises is that the upper part of the occupied zone appears to be the most polluted area with regard to both thermal and atmospheric contaminants. This tendency is independent of the ventilation principle and technical layout.

Introduction

The airflow pattern within a occupied zone can have a considerable impact on the indoor air quality, the thermal comfort and the energy performance of the ventilation system. Even with a sophisticated measuring concept, it is difficult to consider in detail the pattern of the airflow, or the influence from air movement on thermal and pollutant transport.

Computational fluid dynamics (CFD) enables the air flow pattern to be predicted within a zone with a wealth of details, if sufficient computer resources are available. The development of the CFD technique started at the middle of the seventies. However, the applications to building physics are relatively new.

The CFD technique permits sensitivity studies which would be almost impractical to accomplish if carried out by using full scale trials. The parameter variations related to building geometry and technical layout are especially difficult to realise in full scale, due to the cost and restriction of interfering with the daily activity in the room.

CFD simulations have proved to be an extensive supplement to full scale measurements and vice a versa. In this study measurements from an auditorium at University of Trondheim found the basis for setting up boundary conditions and for verifying the CFD results.

All simulations carried out in this study are executed on a CRAY X-MP. In total the author has been sponsored with 75 CPU-hours from the Norwegian Research Council for Science and the Humanities (NAVF).

The KAMELEON II computer code

The KAMELEON code has been developed by SINTEF Applied Thermodynamics and has been continuously improved during the last decade. The software packet includes the graphical pre-processor LIZARD, the post-processor MONITOR and the CFD code KAMELEON written in standard FORTRAN.

The purpose of the LIZARD program is to define geometry by Cartesian co-ordinates, define boundary conditions, create the start field and generate the problem dependent FORTRAN code that is linked to and compiled with the KAMELEON II program. The heart in this program is the graphical construction and boundary designer.

The MONITOR program permits graphical presentation of the computed results. Temperature gradients and mass fraction are presented as iso-contours filled with colours or grey shading as option. The flow field is represented by arrows. The length and the direction of the arrows illustrate the velocity of the flow field. Figure 1. shows the gray shadig scale used in this presentation.

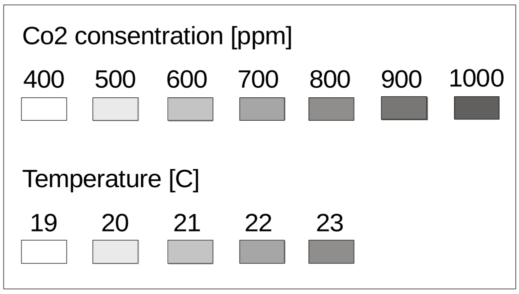

Fig 1. Gray shade scales

The KAMELEON program is the CFD code of the CFD software packet. It contains the 3-dimensional CFD code with a cyclic boundary generator, an obstacle generator and a porous baffle generator. The code also includes several different methods for relaxation, discretization, solving algorithms and matrixes (scalar and vector processing).

The mathematical models /1/ used in the KAMELEON program are based on the theory of continuous fluid mechanics. The equations representing the flow, govern the conservation of mass, momentum and energy. In addition the equation of state for an ideal gas handles the density distribution and a turbulence model simulates the influence of turbulent effects.

These equations give a number of 8 unknowns for each control volume the case is divided into. The total number of unknowns which must be solved simultaneously is found by multiplying the number of unknown parameters with all the partial volumes. The solution of a steady state situation is found by an iterating procedure which is considered as converged when the residue is within a certain limit. When the simulations are transient, the iteration procedure is linked to an automatic time-step generator.

Modelling auditorium EL5

The auditorium EL5 at the University of Trondheim was modelled in three dimensions, using a non-uniform orthogonal grid system. Only half of the room was modelled, presuming that the other half represents a symmetric reflection. The model was subdivided into 29184 control volumes. Each control volume was sized to 25*25*50 cm.

Figure 2 shows a graphical representation of the model. The symmetric section is located at position yz20, (the vertical section on the right side of the figure).

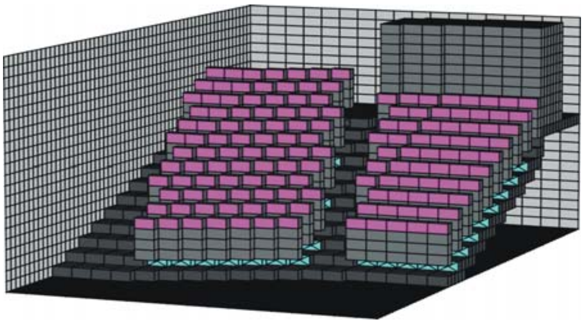

Fig. 2. Graphical representation of the CFD model-EL5. The auditorium is fully occupied

In the graphical pre-processor a control volume is classified either as a boundary cell or as domain cell. A cell and its properties are identified by a colour or grey shades.

A boundary cell may represent a solid adiabatic wall, a solid isothermal wall, a flow inlet/outlet or a symmetrical boundary.

The domain cells are categorised into obstacles, free cells and locked cells. Inside obstacles only the energy equation is solved. Free cells may be partly closed by activating the volume porosity parameter or the surface porosity parameter.

All surfaces in the EL5 auditorium were modelled as isothermal boundaries. Initial surface temperatures were gathered from full scale measurements. The exhaust slots in the ceiling were modelled with outlet boundary cells.

The students were modelled as columns of obstacle cells with a carbon dioxide mass source cell at the top (pink colour). The "students" were interlocked with heat source cells, which symbolised the heat dissipation from the people in the room.

The air inlet devices were modelled as air mass sources (blue colour). Free cells next to the mass source cells were partly closed by surface porosity to obtain correct air inlet device characteristics. The surface porosity acts as a perforated sheet added to the outlet area of the mass source cell.

All simulations were carried out as steady state computations. The boundary conditions for these simulations were measurements sampled at the end of the lecture. This means that the computed result is an instant picture of the situation minutes before a class is finished.

An alternative method to steady state computations, is to make transient simulations. A transient simulation produces an instant picture of the flow field as it progresses with respect to time. However, such computations require substantially more CPU-time than steady state simulations.

Simulations versus measurements - auditorium EL5

The basis for the preliminary CFD studies of the airflow pattern inside the auditorium, is measurements from an almost fully occupied situation at the EL5 auditorium /2/.

Based on these measurements, the walls were set to be isothermal with a surface temperature of 21 °C. The floor was set to be isothermal with a surface temperature of 20 °C. The ceiling was set to be adiabatic, which means that it does not take part in the heat balance.

Figures 3, 4 and 5 show the computed result of the preliminary studies.

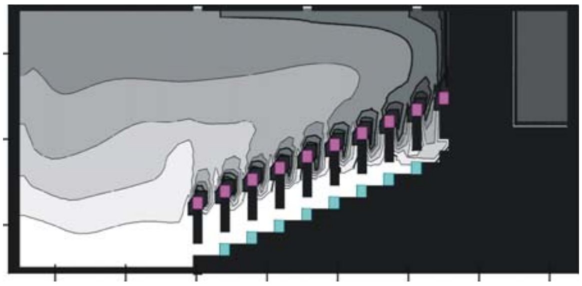

Fig. 3. Concentration chart at section yz19.

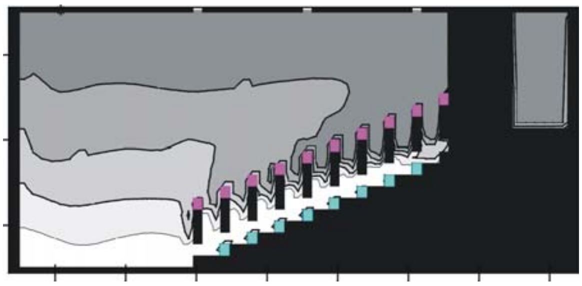

Fig. 4. Temperature chart at section yz19

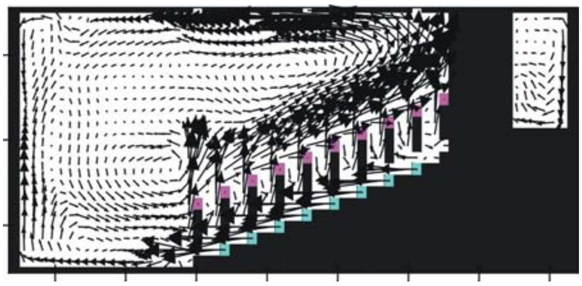

Fig. 5. Air flow pattern at section yz19

The results are presented as graphical charts of temperatures, carbon dioxide concentration and

flow fields. In these computations the carbon dioxide production from each person was set at 20 l/h. The air flow rate was set at 10,000 m^3/h and the number of occupants was set at 300.
The heat production from the students and the lighting is treated somewhat specially. The sensible heat dissipation from a seated adult is about 95 watt. 50 watt is released via convection, the rest is released as irradiation. However, this distribution is only valid for a single person and may vary significantly for a group of persons.

At the time these computations were carried out, there was no radiant model available in the KAMELEON program. Therefore the size of the convection sources were "calibrated" against the measured temperature rise of the air flow passing through the auditorium. These "calibrations" shown that 60 watt/person were released to the air flow at the end of the lecture that formed the basis for the preliminary computations.

In Table 1 temperature and carbon dioxide profiles gathered from the full scale trial of an almost fully occupied auditorium are presented. Comparing these results with the temperature and carbon dioxide charts shown in Figures 3 indicate reasonable conformity between measurements and simulations.

Table 1 Temperature and carbon dioxide profiles - EL5

Height [cm]	Temp. [^oC]	CO_2 [ppm
10	19.6	446
60	20.4	578
110	22.9	858
170	22.7	945
250	22.5	963
350	23.3	868
450	22.8	889

Besides temperature and concentration charts, the simulations also provide flow fields from each computed section. During the full scale trials parts of the air flow pattern were verified by local injection of smoke.

The air flow pattern gives valuable information about the thermodynamic processes taking place inside the auditorium. Following the supply air flow path in Figure 4, shows that the air flows down the terraced floor from the back of the auditorium. It then returns at the "bottom" of the auditorium and follows the convection sources to the back again. Finally the path makes a new return beneath the ceiling towards the front of the auditorium. The "z" path has a total length of approximately 25 m.

The reason for this "z" path is multifactorial. The supply air is flowing down the floor because it is somewhat colder than the room air, and because the floor is terraced. The air returns to the back of the auditorium, because the convection sources (the students) are arranged in a terraced manner causing the convection flow from one source to become entrainment air for the next source. This effect is further emphasised by the fact that the convection sources are entrained by air only from one side and is located next to a vertical surface at the back (the Coanda effect).

The terraced design and the specific flow path cause the climate of this room to be of non homogeneous character. At the front of the auditorium there is a zone that is slightly polluted with a temperature close to the temperature of the supply air. At the back of the auditorium both temperature and pollutant level are significantly higher.

Situations with reduced occupancy

Two different situations with reduced occupancy in the EL5 auditorium were simulated. One simulation has all the students concentrated in the four lowest rows. Another simulation was carried out with the students dispersed all over the room. In both cases the number of occupants have been set at 100, which is approximately 1/3 of the room capacity.

The simulation results are presented in Figures 6, 7, 8 and 9. Two sections, yz6 and yz19 from each computation are presented.

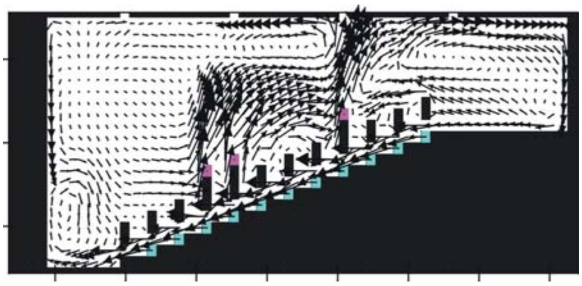

Fig. 6. Air flow pattern at section **yz6**. Auditorium partly occupied - **dispersed** seating pattern.

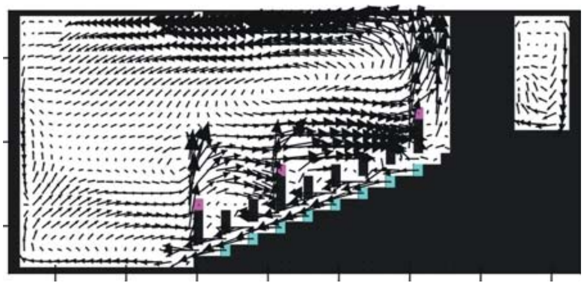

Fig. 7. Air flow pattern at section **yz19**. Auditorium partly occupied - **dispersed** seating pattern

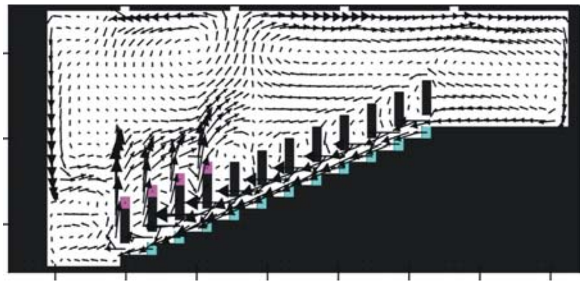

Fig. 8. Air flow pattern at section **yz6**. Auditorium partly occupied - **concentrated** seating pattern

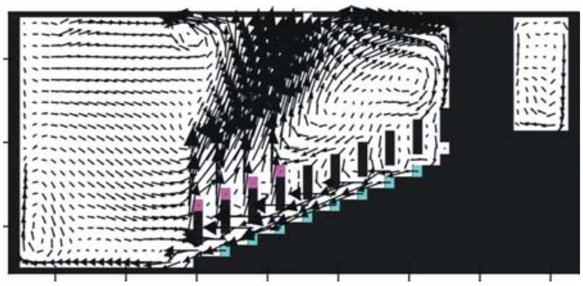

Fig. 9. Air flow pattern at section **yz 19**. Auditorium partly occupied - **concentrated** seating pattern

The simulations were carried out with a ventilation rate of 3,300 m^3/h, which means that the ventilation rate per person was the same as for the fully occupied auditorium. Additional initial conditions remained unchanged compared to the fully occupied situation.

Observing the air flow patterns at the different sections (6..9), shows that the seating pattern has a significant influence on the airflow pattern. Figure 7 (dispersed seating pattern) shows a convection flow that is almost horizontal, while Figure 9 (concentrated seating pattern) shows a convection flow that is almost vertical. The difference is clearly pronounced in sections containing the AV-room (yz19).

Referring to the temperature and concentration charts (not shown), we can observe that the dispersed seating pattern causes more stratification than the concentrated seating pattern. However, both situations display charts showing that the region at the back of the auditorium beneath the ceiling is the most polluted zone with the highest temperature. The area at front of the auditorium represents the coldest and cleanest zone.

Different technical layouts and room geometries

Preliminary CFD simulations have focused on the EL5 auditorium. These simulations have shown that room geometry, ventilation principle, occupancy and seating pattern all are parameters that affect the airflow pattern inside an auditorium. Recommendations applicable for the EL5 auditorium may therefore not be relevant for other auditoria.

This section emphasises design and technical layouts that differ from those found in the EL5 auditorium. The basis for these simulations is layouts and design frequently found in various Norwegian auditoria. All approaches are adapted to the physical size and capacity of the EL5 auditorium, which means that the initial setting and boundary conditions equal those described initially.

The purpose of these simulations is to procure data which can form bases for drawing up of general guidelines and recommendations with respect to sensor location. Limited computer resources however, restrict the possibility of making numerous parameter variations.

Most of these simulations deal with premises based on displacement ventilation. However, some simulations will focus on premises ventilated by dilution.

The results from these simulations must be looked upon as trends, rather than exact solutions. One must bear in mine than none of the following simulations are verified by measurements. All the charts presented are located at section yz19 (see Figure 1). This section represent the centre of the seating pattern.

Air terminal devices - various locations

Three different locations of air terminal devices were simulated. One auditorium has air terminal devices located at the front of the auditorium (10a). Another simulation has air terminal devices located both at the front and the back of the auditorium (10b). A third simulation has the air terminal devices located at the back of the auditorium at the ground level (10c). In this case the seating rows are mobile and may be pushed towards the back wall. All auditoria are ventilated by displacement. Figure 10 sketches the different cases.

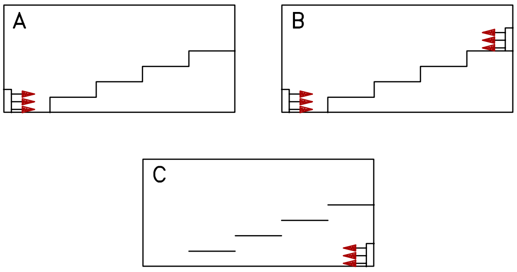

Fig. 10. Various location of the air terminal devices

All three cases have the same room geometry and boundary conditions as the EL5 auditorium. All simulations were carried out with the auditoria fully occupied. In case study (B), 50% of the airflow is supplied at the front of the auditorium and 50% is supplied at the back.

The simulation results are presented in Figures 11 - 16.

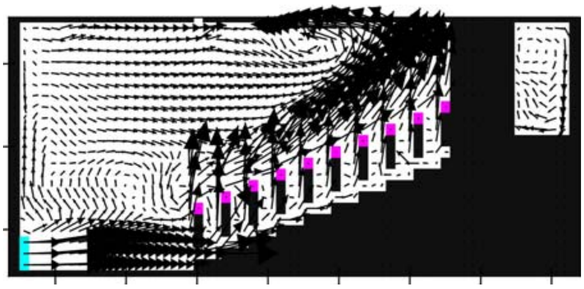

Fig. 11. Airflow pattern at section yz19.
Air terminal devices at the front

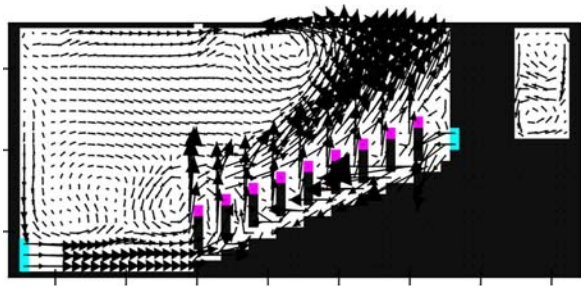

Fig. 12. Airflow pattern at section yz19.
Air terminal devices at the front and back

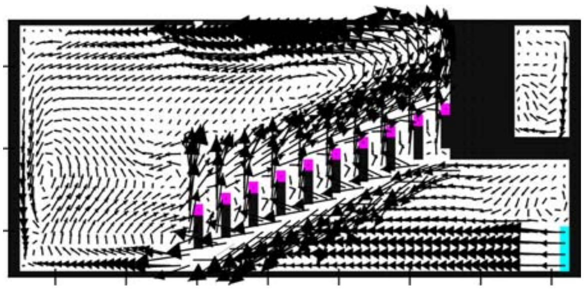

Fig. 13. Airflow pattern at section yz19.
Air terminal devices at the back

What we can observe form these simulations is that the airflow pattern, the temperature and the concentration profile vary slightly from case to case.

Comparing these simulations with the EL5 case study (Figure 2) shows the same trend. We may conclude in these cases that the location of air terminal devices has a small influence on the airflow pattern.

The influence of various room geometries

Three auditoria with different room designs were simulated. All auditoria were ventilated by displacement. One auditorium had a plane floor where all the occupants were sitting at the same level. The second auditorium had a slightly terraced floor, while the third auditorium had a floor gradient set to 10^o. The computer simulations are presented in Figures 14,15 and 16.

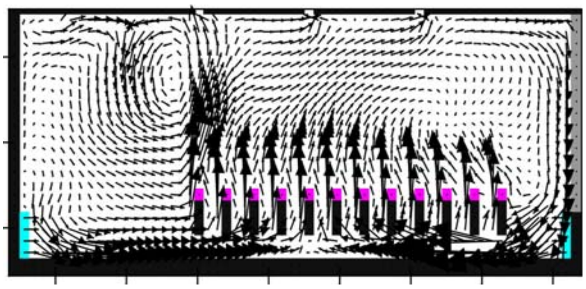

Fig. 14. Airflow pattern at section yz6 .
All seats at the same level

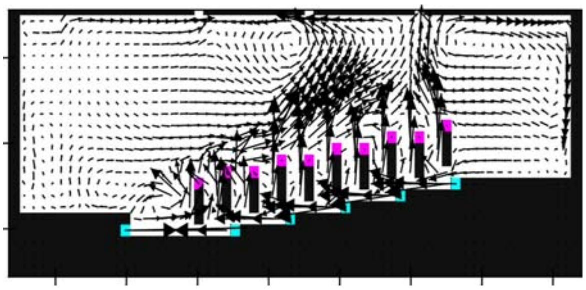

Fig. 15. Airflow pattern at section yz6
Slightly terraced floor (10^o)

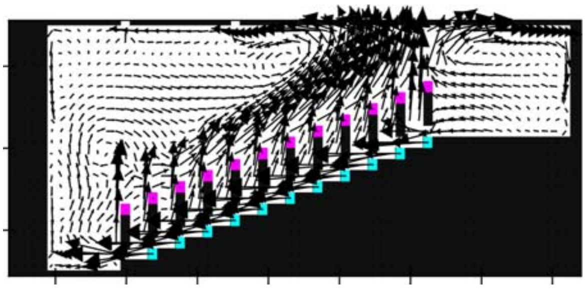

Fig. 16. Airflow pattern at section yz6.
Terraced floor (10^o)

The results show that the floor gradient has a significant influence on the air flow pattern. In auditoria which are slightly terraced or have all the seats on same level, the occupied zone has an environment that is more uniform than if the floor is steeply terraced. In that respect, the presented results are more in accordance with the idea of displacement ventilation, which is to make a clean occupied zone by displacing the contaminants to an upper polluted zone.

The simulation of the non-terraced auditorium (Fig. 14) also reveals the influence of cold draft from the back wall. The fact that the back wall now has full height, causes the thermal convection flow related to boundary condition to be more dominant. From the concentration charts we can observe that this effect causes contaminants at the back of the auditorium to be more concentrated.

Ventilation by the principle of dilution

Three auditoria with different ventilation arrangements were simulated. All auditoria were ventilated by the principle of dilution. All simulations were carried out with room design identical to the EL5 auditorium.

One auditorium had diffusers in the ceiling and exhaust slots at the front of the terraced floor. The second auditorium had diffusers in the ceiling and exhaust slots at the back of the terraced floor. The third auditorium had supply air injected as jets via nozzles in the ceiling, and exhaust slots equally dispersed all over the terraced floor area. Hence, this simulation was carried out with a tilted ceiling.

In contrast to the air terminal devices designed for displacement ventilation, the outlet condition from diffusers and nozzles has a dominant effect on the air flow pattern. To obtain tolerable inlet characteristics, jets are simulated by combining mass source cells with surface porosity cells (perforated sheets).

However, one should bear in mind that limited grid resolution may restrict the conformity between model and reality.

Figures 17 - 22 shows results from the auditorium with diffusers in the ceiling and exhaust outlets at the front. As we may observe, the environment inside the room is rather inconsistent. The most polluted area, is the occupied zone. Lowest concentrations originate next to the boundary surfaces and in the exhaust.

These results are almost identical to the experience gained from the full-scale trial.

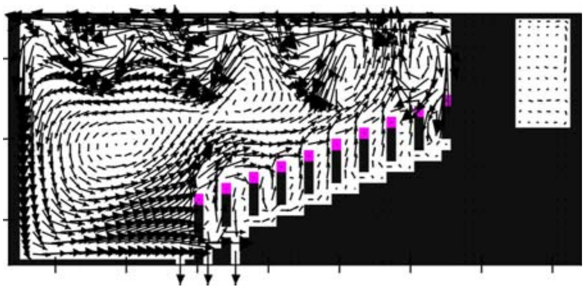

Fig. 17. Airflow pattern at section yz6. Diffusers in the ceiling , exhaust slots at the front

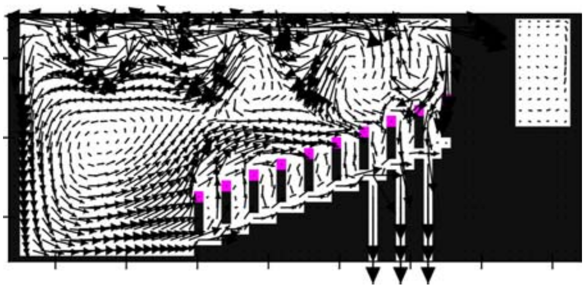

Fig. 18. Airflow pattern at section yz6. Diffusers in the ceiling, exhaust slots at the back.

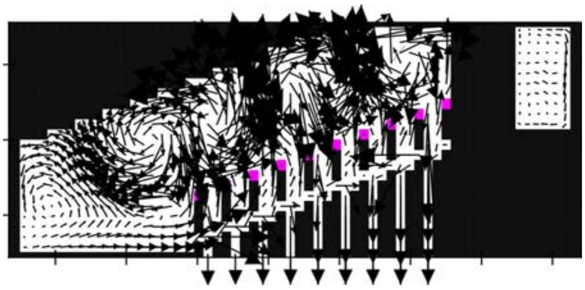

Fig. 19. Airflow pattern at section yz6. Nozzles in the ceiling, exhaust slots under the seats

Repositioning the exhaust slots to the back of the auditorium (Figures 17/19), shows that the environment is somewhat improved. Especial this passes for the lower part of the occupied zone. The concentration gradient however, is still several hundred ppm.

Figures 19/22 show results from the simulation of the auditorium having nozzles in the ceiling and exhaust slots equally dispersed all over the floor. As observed, this particular layout did not manage to establish uniform conditions inside the room.

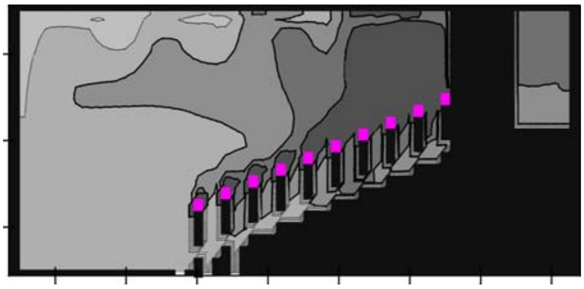

Fig. 20. Concentration chart at section yz19. Diffusers in the ceiling, exhaust slots at the front.

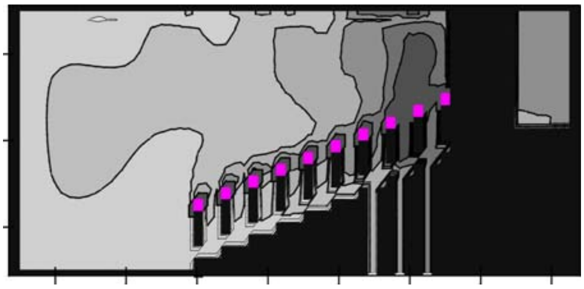

Fig. 21. Concentration chart at section yz19. Diffusers in the ceiling, exhaust slots at the back.

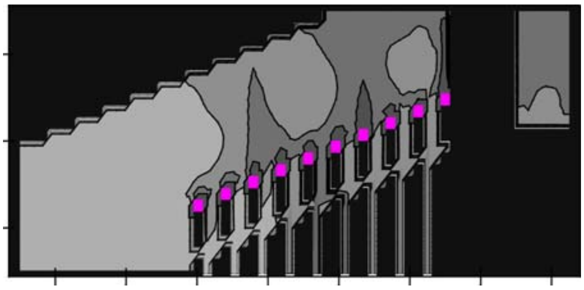

Fig. 22. Concentration chart at section yz19. Nozzles in the ceiling, exhaust slots under the seats.

Efficient elimination of contaminants by dilution, presumes a well mixed air flow pattern. This section has indicated that densely occupied premises generate strong thermal convection flows which complicate the mixing. The result is short-circuiting between the supply and exhaust air. Evidently, more research related to this approach is required.

Closing remarks

CFD simulations of various auditoria have shown that the airflow pattern inside this type of premises may be quite complex. The airflow pattern is affected by several parameters. Those of importance are room geometry, capacity and location of heat source, aspect of building material and construction, furnishing, ventilation principle, supply air temperature, leakages and the surrounding climate.

A general trend from the simulation of terraced auditoria, is that the upper part of the occupied zone appears to be the most polluted area. This tendency seems independent of ventilation principles and technical arrangements.

Even if the occupant load and the seating pattern are varying much, there is a clear trend that the occupied zone at the front of the auditorium will be moderately polluted compared to zones at the back of the auditorium. The tendency is also valid for temperatures.

The thermal climate at the front of terraced auditora is strongly influenced by the supply air temperature when the auditorium is ventilated by displacement. An acceptable thermal climate in this area therefore restricts the minimum supply air temperature

An adequate sensor location both for temperature- and pollutant control (VAV), should be at the back of the auditorium beneath the ceiling. The location will give a good indication of occupancy, but will not cover poorly ventilated zones.

Presuming no leakages and no short-circuiting between the supply and exhaust air, it is also possible to recommend the location of sensors in the exhaust duct. If the sensors are located in the exhaust duct, the ventilation rate must always be kept above a certain minimum.

A small digression with respect to indoor air quality judgement is relevant. Judges entering the room represent new heat sources which may affect the airflow pattern significantly. The result from this kind of judgement may therefore not be representative of the air quality.

References :

/1/ Lakså B. Vembe B.E.
Kameleon II, SINTEF STF15 F91048,
Trondheim 1991

/2/ Drangsholt F
The Applicability of Demand Controlled
Ventilating System for Assembly Halls,
NTH - Trondheim 1992,