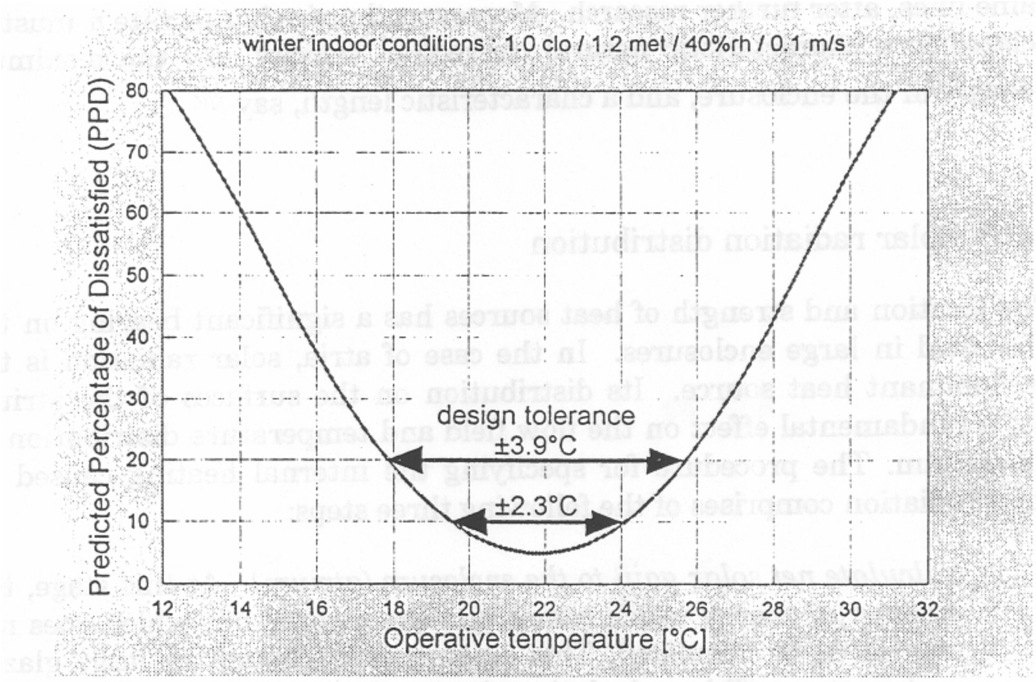


Fig. 1

The accuracy of CFD calculations depends on the integration method, grid size and other parameters. Using a coarse grid causes larger numerical errors. But when modelling atria and other large enclosures, it is computationally costly to use a fine grid throughout the flowfield. Thus for practical applications of CFD, a balance must be struck between computational accuracy and computing cost. The computational accuracy depends not only on the total number of grid points, but also on their distribution over the computational domain. The grid should be most refined in regions of large gradients, in particular near walls, supply jets and outlets.

Borth & Suter (1994) have suggested a dimensionless number G (Equation 11) which can be a guide to determining the required level of grid discretization for 3-dimensional CFD simulation of a room with mixing ventilation.

$$G = \left(\frac{V}{N}\right)^{\frac{2}{3}} \frac{n}{\nu} \qquad (11)$$

In their test case, using a standard k-c turbulence model, a coarse grid of G > 1 was found to be sufficient for only qualitative evaluation of the simulation results. The use of a finer grid of G < 1, was suggested to be adequate for qualitative purposes. These guidelines can not directly be applied to modelling atria which have predominantly buoyancy-driven flow, though it may be possible to develop relevant guidelines along the

- The last distribution method tried was more rigorous. It takes into account the first specular and two diffuse reflections of the solar radiation. See Figure 2.

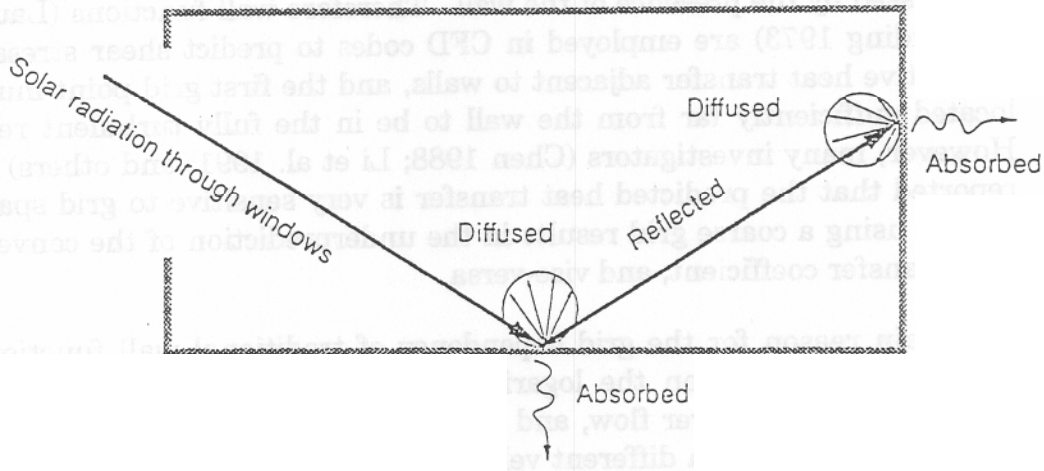


Fig. 2

A heat flux boundary condition was used in all three cases. The results presented and compared with measurements in Figure 3. It is interesting to note that with the distribution based on surface ratio, there was insignificant thermal stratification. Already with the distribution method using only the surface directly heated by the sun, the resulting vertical temperature profile was much closer to that which was measured. The last distribution method was slightly more accurate.

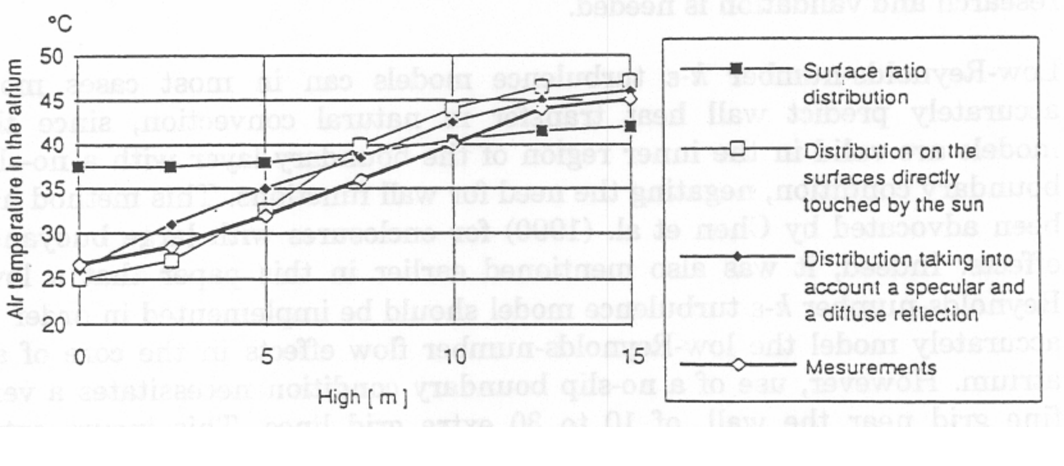


Fig. 3

The current study: The objective of the current study has been to establish the most accurate way of defining natural convection boundary conditions using a k-s model with wall functions, and to quantify the errors involved. A 2-dimensional model of a large atrium (10m high, 10m wide) has been studied with the CFD code Flovent (Figures 4 & 5a). All of the atrium's surfaces are adiabatic except for a high cold wall. The cold wall's U-value (2 W/m^2K) and the steady-state outside design air temperature (0°C) are both explicitly known. Air is supplied at low velocity from the roof (20°C, 0.01m/s) and is extracted through a 0.5m wide slot at the foot of the of the cold wall. The downdraught from the cold wall is therefore 'sucked out' at floor level, such that the air in the core of the atrium is stagnant and thermally stratified. The Rayleigh number for the atrium ($Ra=7x10^{11}$) is very high, implying predominantly buoyancy-driven airflow. For simplicity, solar radiation was not modelled, and surface-to-surface radiation exchange was only modelled in the last case (Figure 5b). The following topics were investigated:

- Methods of defining boundary conditions
- Grid resolution near the cold wall
- The need for surface-to-surface radiation modelling

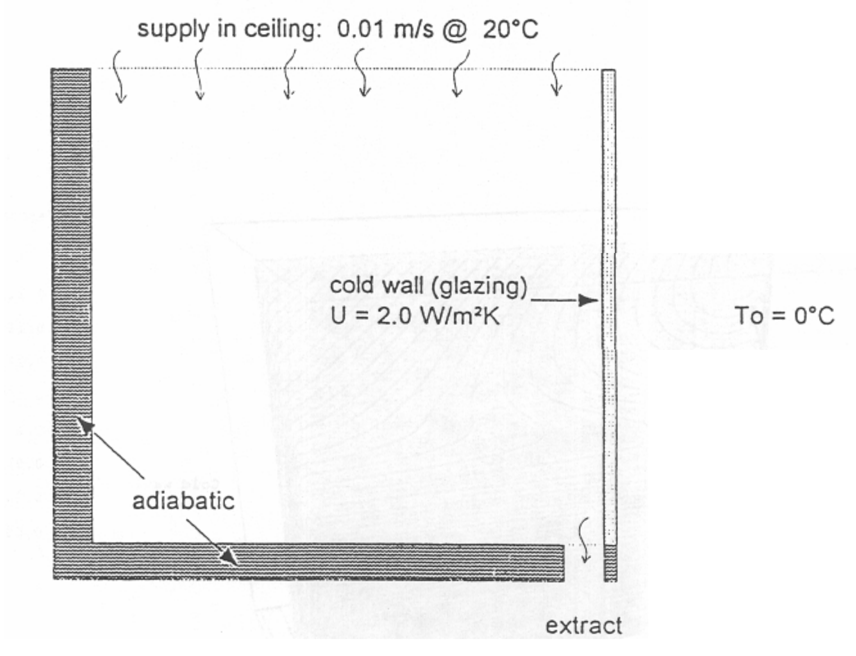


Fig. 4

Study of different boundary condition methods

The following five common methods of prescribing boundary conditions were investigated. For simplicity, each method has been given a shortcode. See also Figure 6.

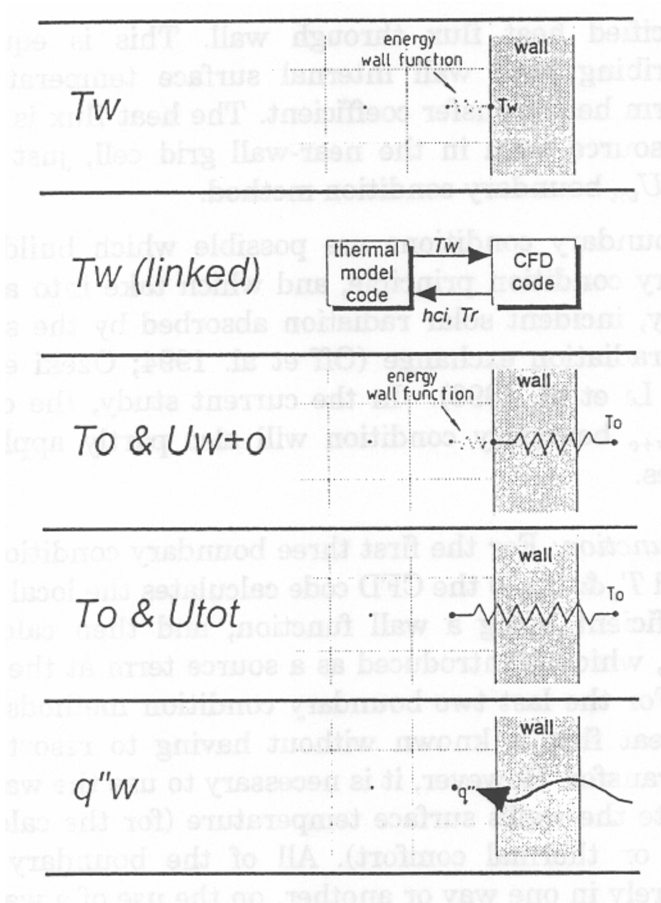


Fig. 6

code	description
T_W	• Specified wall internal-surface temperature.
T_W(linked)	• The CFD code using the T_W boundary condition, is iteratively linked to a building thermal modelling code. The thermal model is fed the calculated bulk convective heat transfer coefficient for each surface, and room bulk temperature, from the CFD code, and returns a recalculated value of T_W to the CFD code. In this study the thermal model was set up on a spreadsheet, and the 'linking was done by hand 7 times at intervals of approximately 500 iterations.

Below is a rendition of how the T_w and q boundary conditions were calculated in the study.

(1) *Decide the internal & external heat transfer coefficients.* In this case, the values given in Norwegian Standard NS 3031 (1987) were used: h_{ci}=2.7 W/m²K; h_o=25 W/m²K. The choice of internal heat transfer coefficients for real buildings is a matter of debate. A comparison of the natural convection heat transfer coefficient correlations for vertical surfaces (external & in enclosures), collated by Dascalaki et al. (1994), which are valid at the Rayleigh number for the modelled atrium, gave a best fit value of 3.0±0.6 W/m²K which is close to, and understandably slightly higher than the value specified in NS 3031.

(2) *Decide the bulk air temperature (heat transfer temperature)*, which is a function of the inlet/outlet air temperatures in the atrium, and the airflow pattern. In this study it was sensible to assume that $T_r=T_s$ since the cold downdraught mixed little with the room air before being sucked out at floor level. For a general enclosure the relationship is $T_r = x \cdot T_s + (1\text{-}x) \cdot T_e$ where x takes a value between 0 and 1.

(3) *Solve the energy balance for the atrium*, using the assumptions from steps (1) & (2) above, to find q and T_w. The energy balance involves the two equations: • $q\,A_w = c_p\,(T_s\text{-}T_e)$

• $q = h_{ci}\,(T_w\text{-}T_r)$ where: $T_w = T_r - (T_r\text{-}T_o) \cdot U_{tot}/h_{ci}$

In this study, the solution was q=27.013 W/m² and T_w=9.995°C.

A good example of applying the above principles to the design of a real atrium, is given by Kondo & Niwa (1992) who used an advanced multi-zone dynamic thermal model for the calculation of their boundary conditions.

The boundary condition method T_o & U_{tot} relies only on one of the aforementioned assumptions; that of the value of h_{ci}. For the boundary condition methods T_o & U_{w+o} and T_w*(linked)*, no such assumptions had to be made, and so they would not be expected to suffer from errors in this respect.

Additional calculations: The CFD results were imported into a spreadsheet for post-processing. The bulk air temperature (T_r), convective heat transfer coefficient (h_{ci}) at every height along the wall (Equation 12), and nondimensional distance from the wall (y^+) (Equation 13) are some of the post-calculated variables.

$$h_{ci_x} = q_{w_x} \cdot (T_r - T_{w_x}) \qquad (12)$$

$$y^+ = \frac{y_p \;\; u_\tau}{\nu} \qquad where \quad u_\tau \;\; C_\mu^{1/4} \;\; k_p^{1/2} \qquad (13)$$

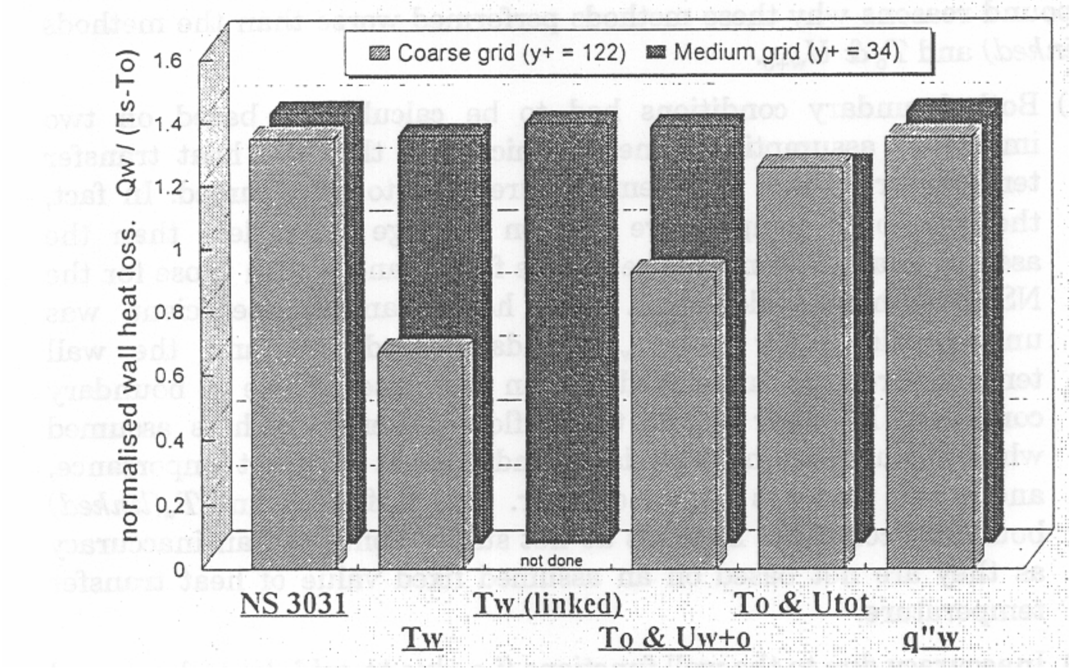

Coarse grid (y+ = 122)
Medium grid (y+ = 34)
normalised wall heat loss. Qw / (Ts-To)
1.6
1.4
1.2
1
0.8
0.6
0.4
0.2
0
not done
NS 3031
Tw
Tw (linked)
To & Uw+o
To & Utot
q"w

Fig. 8a

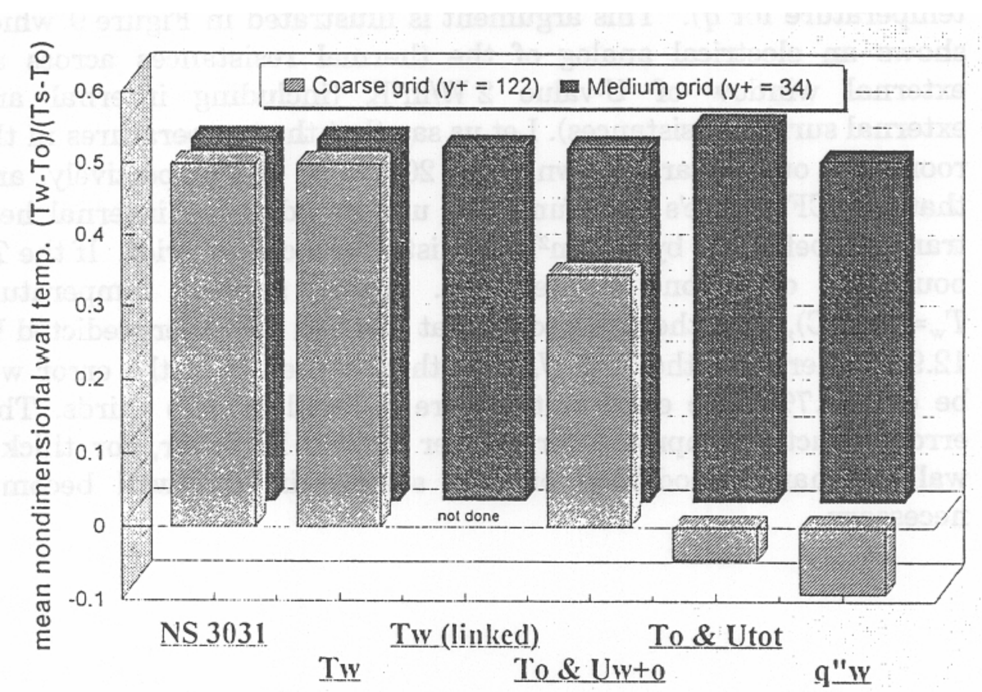

Coarse grid (y+ = 122)
Medium grid (y+ = 34)
mean nondimensional wall temp. (Tw-To)/(Ts-To)
0.6
0.5
0.4
0.3
0.2
0.1
0
-0.1
not done
NS 3031
Tw
Tw (linked)
To & Uw+o
To & Utot
q"w

Fig. 8b

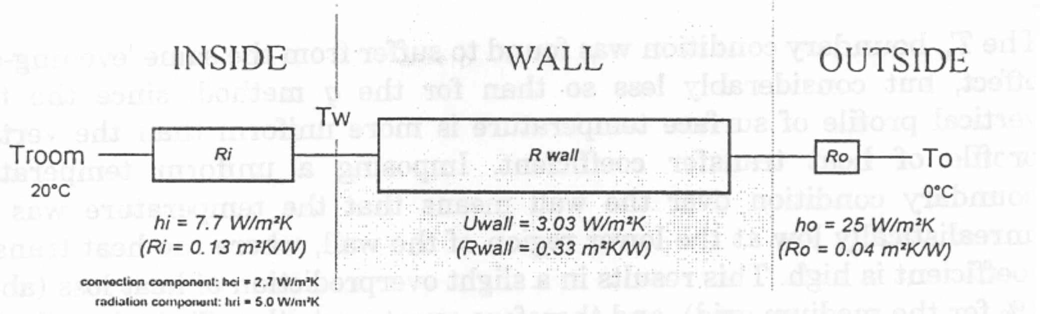


Fig. 9

(3) Enforcing uniform values of T_w, or q along the wall's height is another cause of inaccuracy. For the q boundary condition, this results in an 'evening-out' of wall heat flux, such that the wall heat loss at the top of the cold wall is set unrealistically high and vise versa for the bottom region of the wall. At the top region of the cold wall the convective heat transfer coefficient is low because the cold downdraught is not yet fully developed. The heat transfer coefficient increases with distance down the wall. For the coarse grid, applying the wall functions to calculate the wall surface temperature resulted in large negative temperatures at the top quarter of the wall (-80°C at top of wall), which is physically unrealistic. The wall's mean surface temperature was calculated to be -1.8°C in the case of the coarse grid calculation; i.e. lower than both the room temperature and outside temperature. This is illustrated in Figure 10 which shows the predicted surface temperature profiles for each boundary condition.

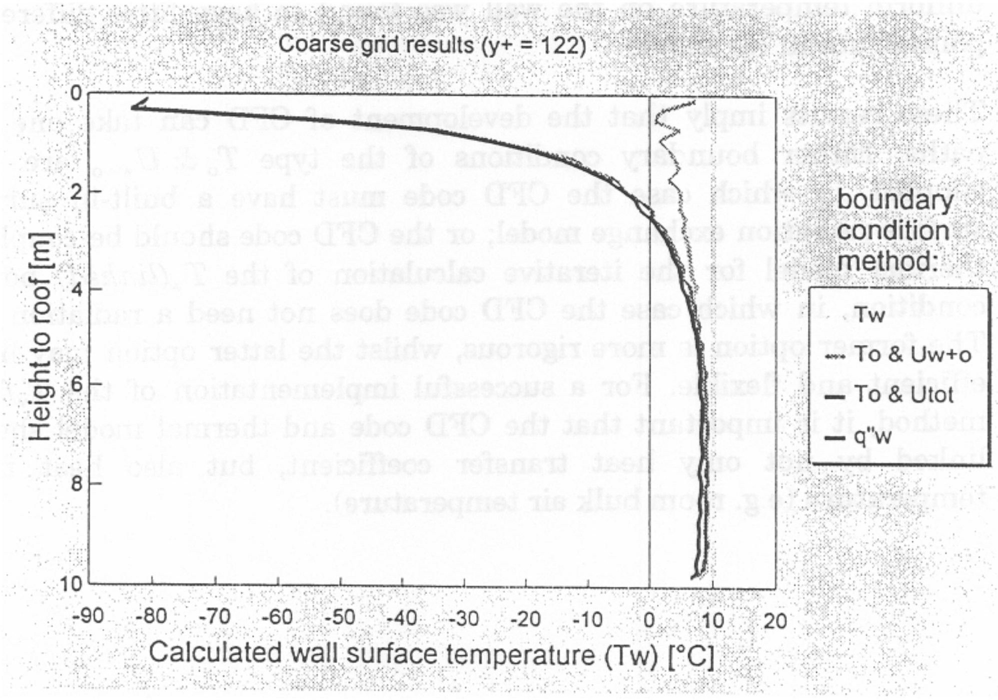


Fig. 10

Further study of grid refinement

A more detailed study was carried out using just the T_o & U_{W+o} boundary condition. Different grades of grid refinement were tested, with the first grid point located in various regions of the downdraught's boundary layer; from within viscous sublayer (Δy=2.2mm,+=2.4) to the turbulent outer region (Δy=1000mm,y^+=207). Figure 11 showsy$^+$ versus predicted mean bulk heat transfer coefficient and mean nondimensional wall temperature. It is clear from the chart that the results are highly grid dependent, due mainly to the unsuitability of the wall functions for natural convection boundary layers.

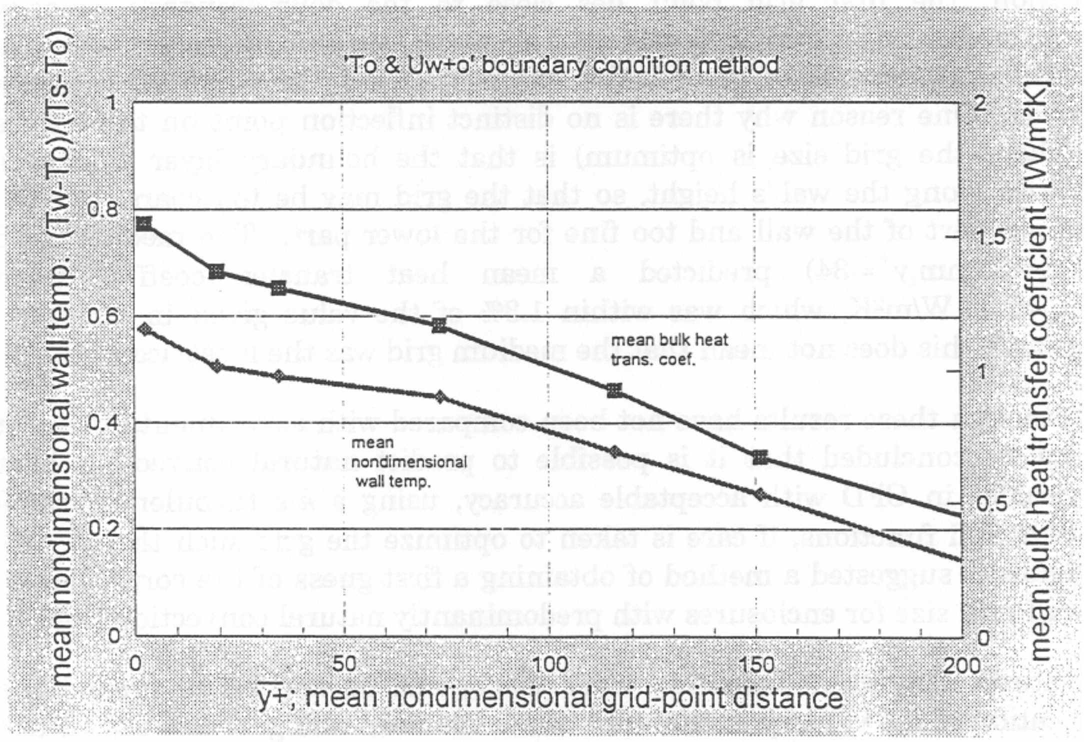


Fig. 11

Combining Equations 14 and 15 gives the following relation for estimating the required near-wall grid element size *(Δy)* in millimeters:

$$y = 5.35(g\beta\Delta T_f)^{-0.1} H^{0.7} \qquad (15)$$

If, after the first CFD simulation, the resulting value of y^+ is unsuitable, then the near-wall grid size should be fine-tuned before performing further simulations. A crude method of fine-tuning the grid is to assume that y is proportional to y^+ (assuming constant wall shear stress through the boundary layer). It is thereby possible to use extrapolation or interpolation to estimate y for y^+=30.

The need for radiation modelling

For the final part of the study, surface-to-surface long-wave thermal radiation was modelled in the CFD simulation of the model atrium. The radiation fluxes were imposed as plane heat sources/sinks on the internal surfaces of the atrium. A simple radiation model to calculate the radiant fluxes, was set up on a spreadsheet. The radiant fluxes were recalculated and entered into the CFD program 11 times, at intervals of approximately 1700 iterations. Figures 5a and 5b show the resulting temperature and flow fields for the simulations without, and with surface-to-surface radiation. It is clear from these figures that the temperature and flow fields are significantly different. It is therefore vital that CFD simulations of atria should include surface-to-surface radiation exchange.

Models for internal heat sources

The most significant heat source in an atrium is solar gain. Two simple ways of introducing solar gain boundary conditions are:

(1) Heat flux specification (q): This is the easiest way, though it is strictly only suitable if the surface is lightweight and well insulated. In this context the *q* boundary condition does not suffer the same degree of poorly predicted wall temperature, as was observed earlier when the *q* specification was applied to model heat transfer through a wall of known U-value. Strictly speaking, the value of *q* should be taken from a dynamic thermal model with surface-to-surface radiation exchange, using the correct distribution of solar gains.

(2) Surface temperature specification (or T_W*(linked)):* In this case, a dynamic thermal modelling program is needed to calculate the temperatures of the different surfaces in the atrium, using the correct distribution of solar gains. If the T_W boundary condition is used then the wall heat flux will be erroneous, whereas the T_W*(linked)* method will more accurately model the wall heat flux.

$$\Delta p = \tfrac{1}{2} f \rho V^2 \quad (17)$$

V	=	average velocity through the opening
p	=	density of the air
f	=	loos coefficient

The effect of the wind on an opening can be taken into account by increasing the relative pressure :

$$P_{stagnation} = p + \tfrac{1}{2} C_p \rho W^2 \quad (18)$$

W	=	wind velocity at the opening level
p	=	density of the outside air
Cp	=	Pressure distribution coefficient at the opening level

The hydrostatic pressure does not need to be included in the - relative pressure because it is already included in the momentum equation which is solved by the programme. But it is important to define the reference temperature and density. Normally the external air temperature and density are chosen.

The infiltration through leakages of the building envelope can also be defined in the similar way. The problem is to find the location of these leakages, their size and pressure drop coefficient.

Sometimes when the overall air exchange rate has been measured, it is possible to simplify the boundary condition by introducing under the neutral axis the total amount of the outside air in a diffuse way (large opening surface) and by extracting the same amount of air over the neutral axis also in a diffuse way. This method has been used in a winter simulation of the ELA atrium with some success.

Steady-state or transient boundary conditions

The effect of thermal capacity can have a large impact on the environment in an atrium. Ozeki et al. (1994) report that the predicted mean room air temperature in a very large atrium was significantly different when a dynamic simulation was carried out. The swings in temperature were damped (peak temperature was 4°C lower) and delayed significantly (by 3 hours). They conclude that a transient analysis is imperative for such enclosures.

Holmes et al (1990) also report on transient CFD simulation, this time of an office. The CFD code was coupled to a dynamic thermal model to evaluate the performance of the HVAC controls. The computational overhead of coupling a dynamic thermal model was 0.5%, the run time being dominated by the heavy computational requirements of the transient CFD analysis.

8.4 Example of CFD applications in atria

8.4.1 University of Neuchâtel

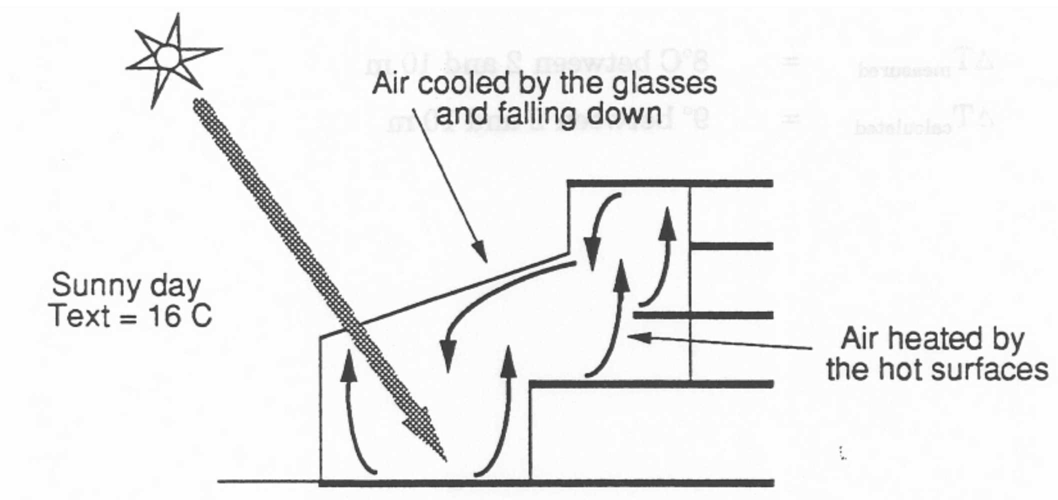
A flow field simulation of some typical cases of the atrium of the university of Neuchatel (CH) has been done during the annex.

This atrium has been presented in chapter 7. 2-D steady calculations have been done for different typical cases :

A) **No solar protections and no passive cooling (no openings)**

No important stratification occurs because the solar gains are directly heating the floor and the wall. The "cold" window surfaces are cooling the air and creating a downdraft flow; the "warm" surfaces as floor and front wall are creating an upwind flow.

These two flows are mixing the air and disturb the stratification of the temperature.



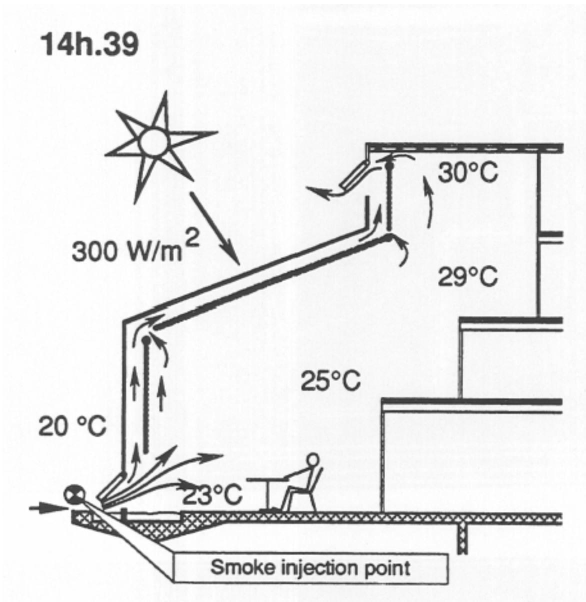
$\Delta T_{measured}$ = 3 to 4°C between 2 and 10 m

$\Delta T_{calculated}$ = 4° between 2 and 10 m

C) Solar protections with passive cooling

The situation is the same as case B but the lower and upper hatches are opened.

The arrows shown in the next figure are representing the flow direction determined with the aid of smoke flow visualisation.

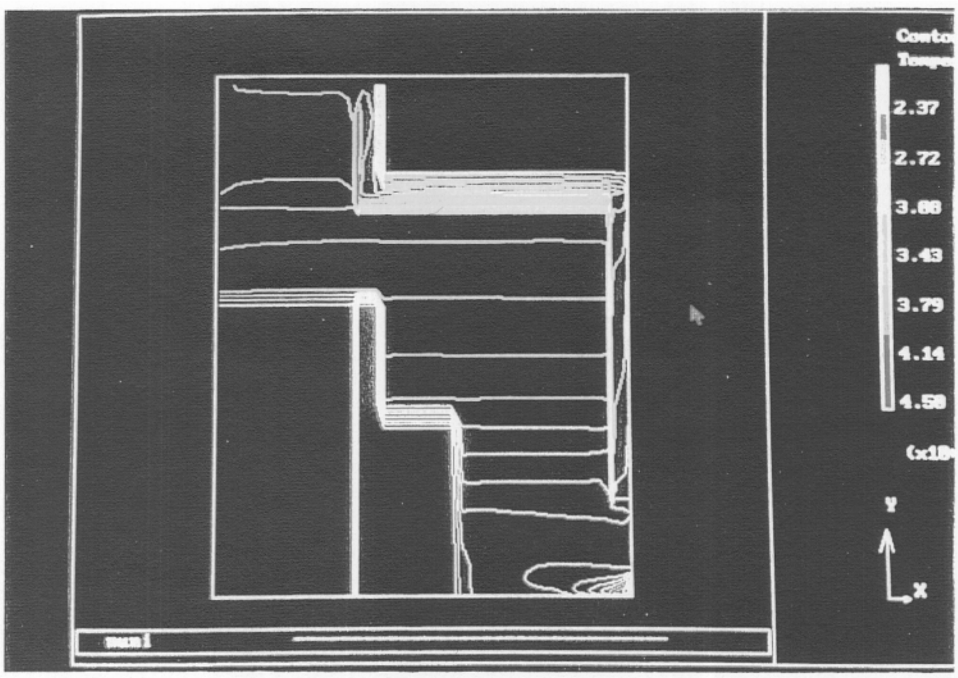


$\Delta T_{measured}$ = 5°C between 2 and 10 m

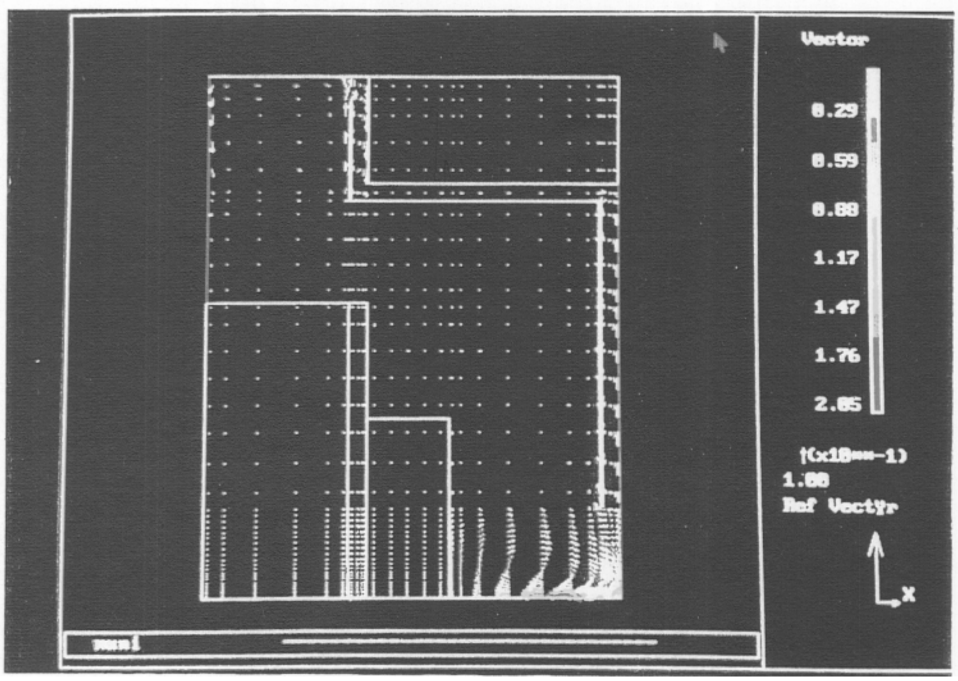
$_{calculated}$ = 4° between 2 and 10 m

D) Solar protections with only low opening

Here only the low hatches have been opened. The results of the CFD calculation: are shown in the next figures.



$\Delta T_{measured}$ = 21°C $\Delta T_{calculalted}$ = 17°C between 2 and 10 m



Although a complete quantitative comparison make no sense, becaus NUNI is a 3-D case and the boundary conditions are complex, one can see that the principal phenomena have been correctly simulated by the program.

8.4.2 Comparison of the measured temperature field in a real atrium with the Flovent and Kameleon calculation

Two typical situations have been calculated with CFD and compared with the measurements :

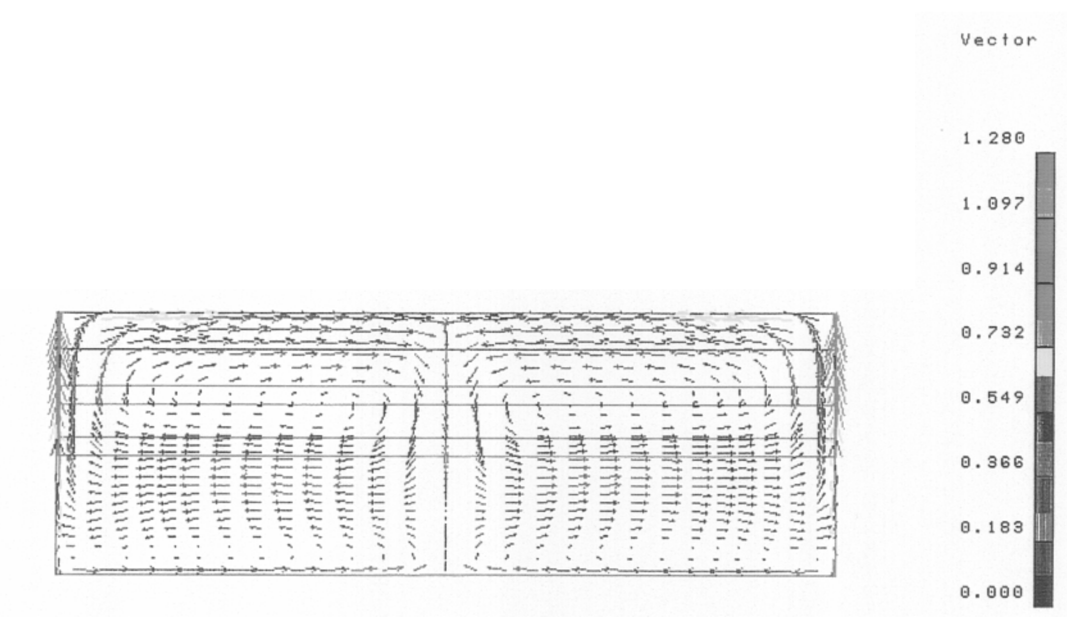
- Winter case with heating convectors (night)
- Summer case with and without opened vents (midday)

The atria used for the comparison is situated in Trondheim (Norway). The data of the building is as follows :

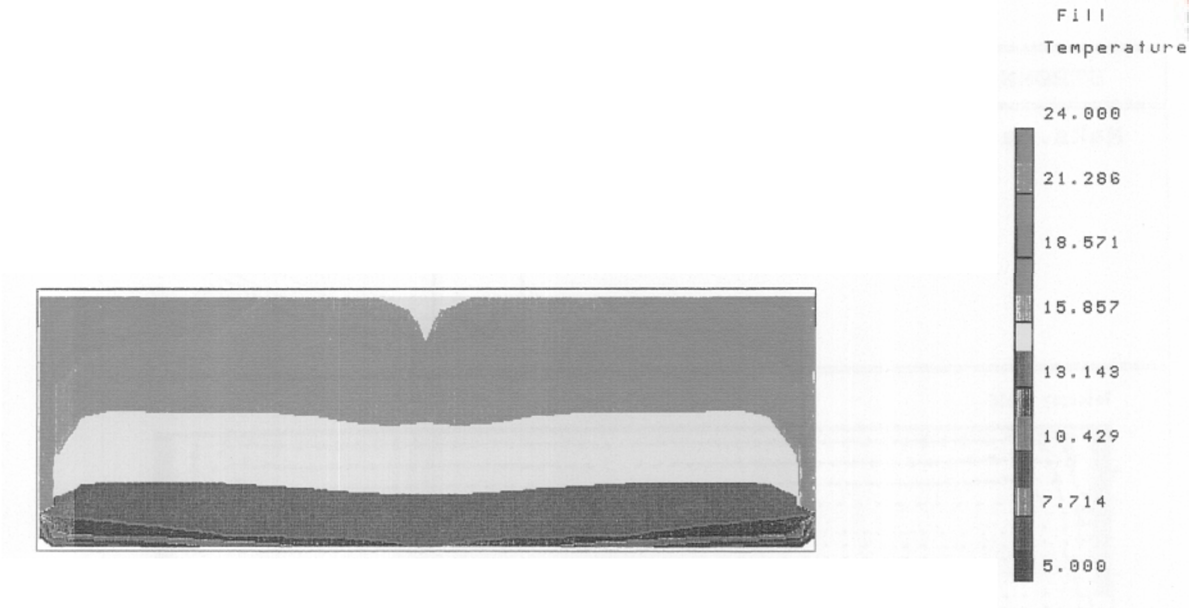
Characteristic measures of one atrium, SG1:	U-values (W/m² K):	Design temperatures:
Length, m: 46	- Atrium glazing: 2.1	- Atria: 15°C
Width, m: 10	- Windows in exterior walls: 2.1	- Offices: 20°C
Max. Heigth,m: 16	- Intermediate wall: 3.2	
Volume, m³ : 7003		

Winter case : night

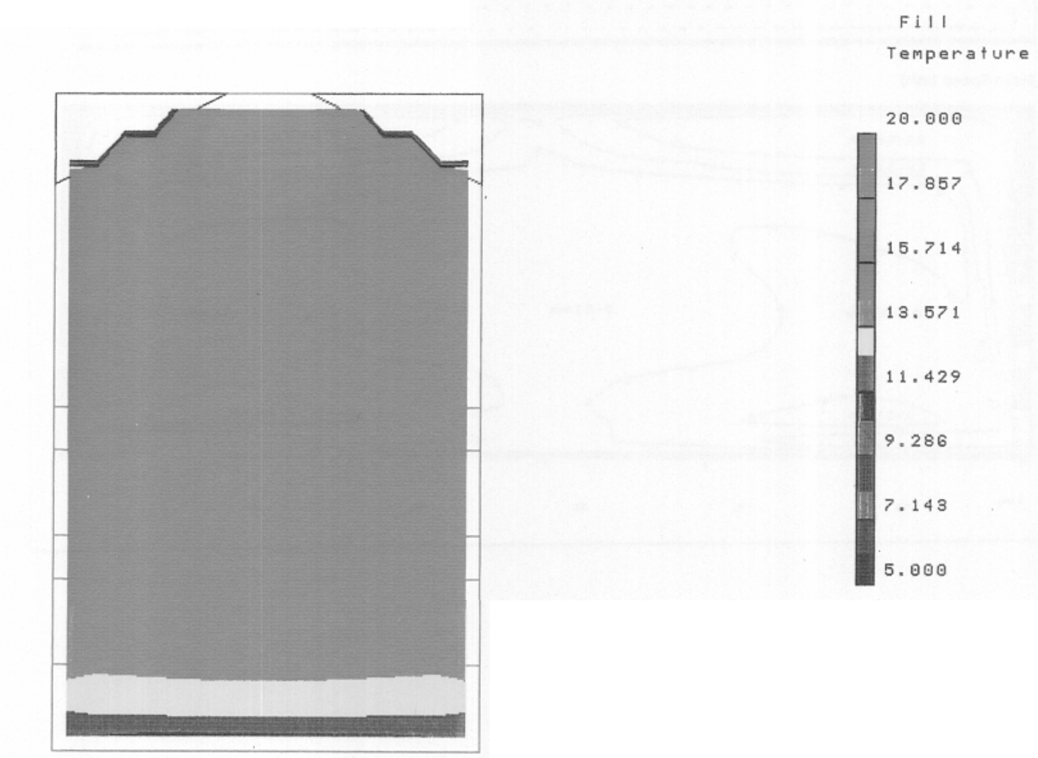
Outside temperature:	-19°C	Unvalue glasses	= 3 W/m^2k
Office temperature:	20°C	Unvalue of the walls	= 2.1 W/m^2k
heat of convectors situated on the gable	100 kW	Infiltration	= 0.5 V/h



Calculated flow field in the middle of the atria

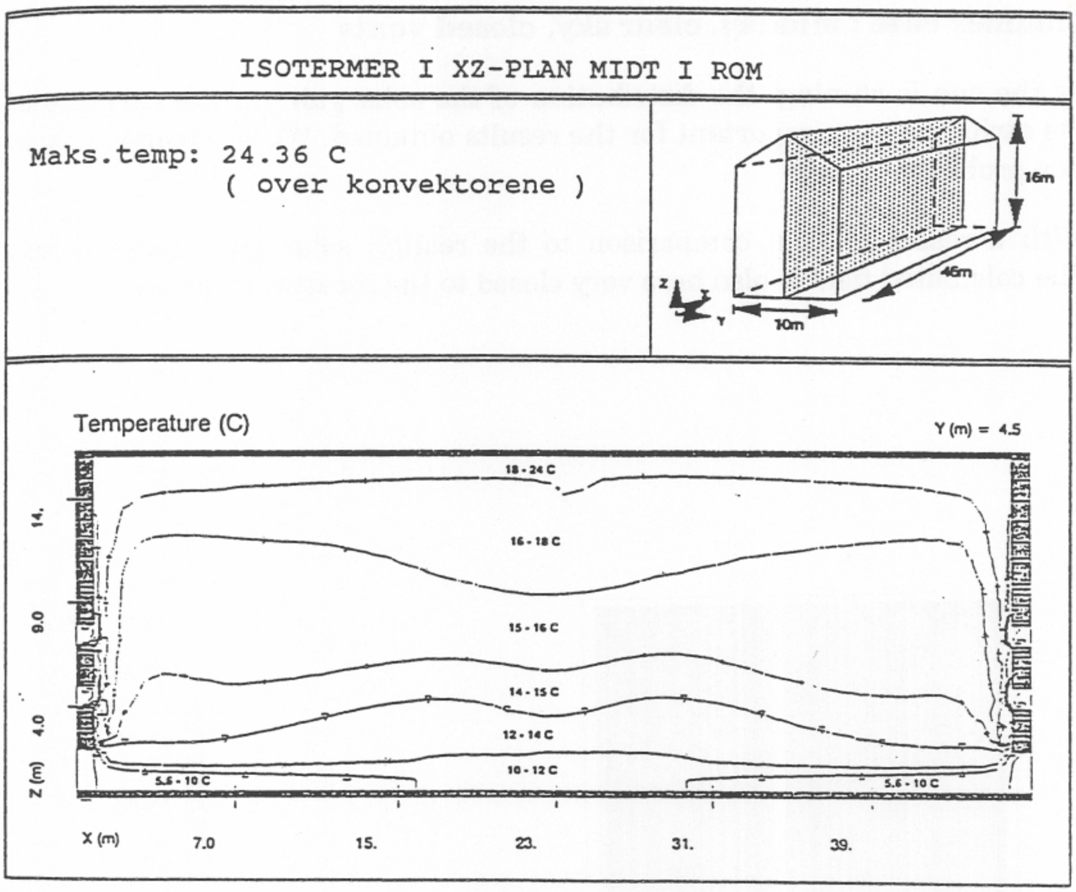


Calculated temperature field in the middle of atria



Calculated and measured temperature field in the section A.

As we can see in the above figure the temperature which has been calculated correspond rather well to the measured values. Similar calculations have been performed with the program Kameleon. The flow and the temperature fields we presented in the next two figures. Also here the comparison is good and shows the ability of CFD if used correctly to predict internal indoors environment.

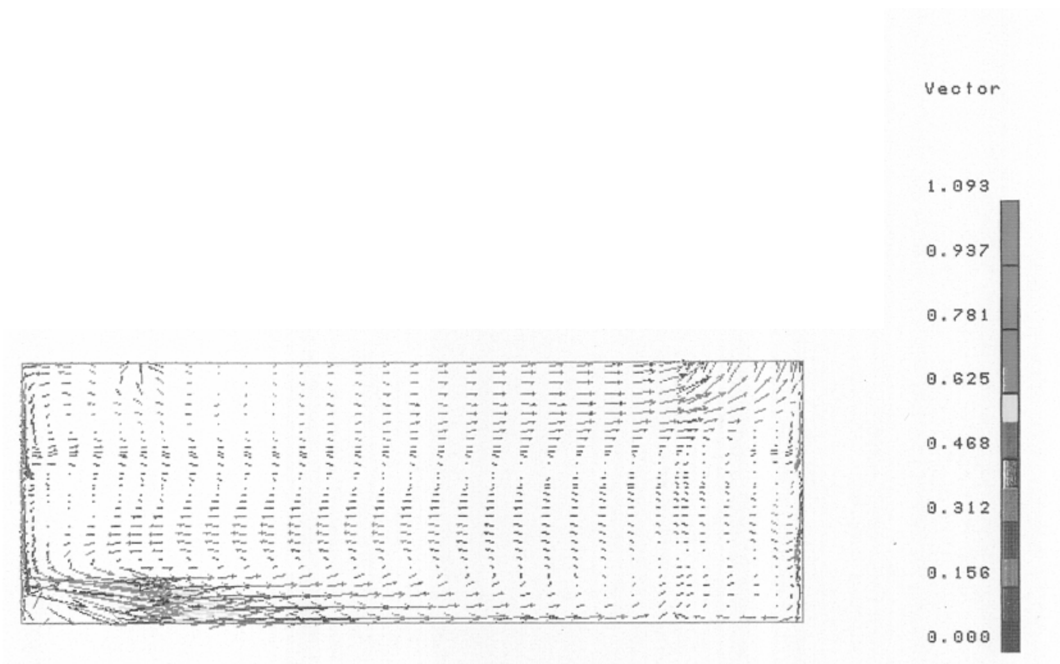

ISOTERMER I XZ-PLAN MIDT I ROM
Maks.temp: 24.36 C
(over konvektorene)
16m
46m
10m
Z
Y
Temperature (C)
Y (m) = 4.5
18 - 24 C
16 - 18 C
15 - 16 C
14 - 15 C
12 - 14 C
10 - 12 C
5.6 - 10 C
5.6 - 10 C
14.
9.0
4.0
Z (m)
X (m)
7.0
15.
23.
31.
39.

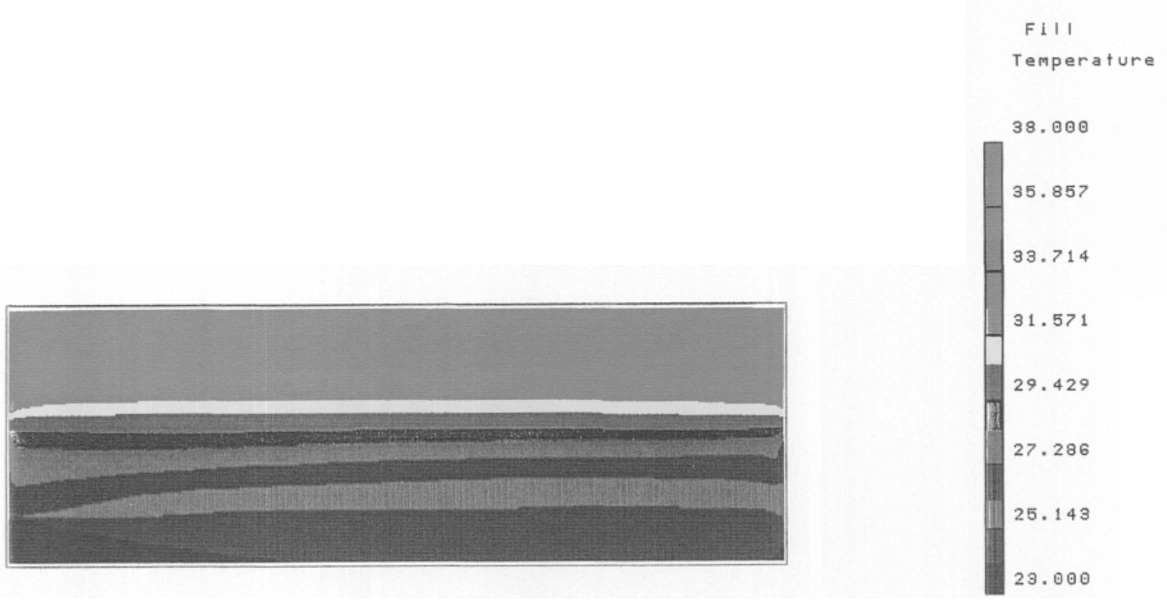
Summer case : Midday, clear sky, opened vents

With opened vents both the temperature stratification and the average temperature level decreases.

A quantitative comparison is difficult because of some unknown values as the wind velocity, and exact opened surfaces.

The qualitative as effect is predicted correctly by the Flovent calculation, some more complete set of data will be collected in the frame of IEA task 26 and will allow a better comparison. The two following figures illustrate the temperature and flow fields calculated with Flovent with 6 m^2 opening area, no wind and external air temperature of 23°C.




Fill
Temperature
38.000
35.857
33.714
31.571
29.429
27.286
25.143
23.000

8.4.3 Validation of simplified model for natural ventilation with CFD calculations

In order to be able to use the atrium also during the summer, it is important to avoid overheating in the atrium. For that two things have to be done :

1. Internal or external shadowing devises
2. Openings -> natural ventilation

The first point will not be discussed here. We will focus the attention on point 2 : cooling by natural ventilation.

Two things are of interest when we try to ventilate an atrium.

a) Is the necessary cooling load achieved (air exchange rate) ?
b) Are the velocity in the region of the occupation zone in atrium not too high (comfort problem) and the temperature too low.

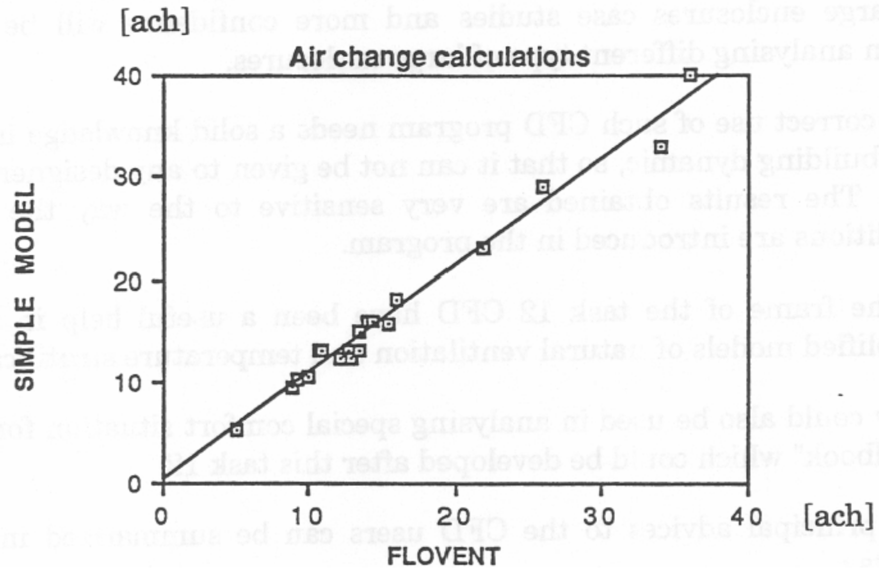
The simplified models should answer to the first question, and allow to determine an air exchange rate of the atrium with the outlet which could be used in the building calculation.

The second question is more difficult to answer with simplified model. Velocity distribution in the occupancy region are given for the moment only by CFD programs. It is for the moment not known if simplified input profiles such as those of wall jets, or displacement ventilation could be applied in this case with some success.

As there is a lack of measurements data in atria, Flovent has been used in order to overcome this problem. The simplified models based on the Bernouille equation have been validated in comparison with the Flovent results in different typical situations.

The figures of the next pages are illustrating some possible situations which could interest the designer.

The next figure shows a global comparison between the results of the simplified models and those obtained with Flovent for different situation and boundary conditions.



These models are really promising in calculating overall air exchange rates when the temperature profile in the atrium is known and well calculated. More detail on these simplified models and their comparison with CFD calculation are given in chapter 6.

- For external walls/floor/roof it is best to use a boundary condition for heat transfer that implicitly depends the outside ambient conditions. Three good ways of doing this have been identified :

  One method is to prescribe the outside air temperature and the U-value between the wall's inside surface and the outside air. The CFD code is left to calculate the internal convective heat transfer coefficient. The CFD code must include a radiation exchange model.

  A second method is to couple the CFD code with a thermal model. Each iteration, the thermal model is fed the calculated bulk convective heat transfer coefficient for each surface, together with the heat transfer (bulk air temperature), and returns recalculated values of surface temperature to the CFD code. This method gives almost identical results to the method above, although in this case the radiation model should be implemented in the thermal model.

  A third method also involves coupling the CFD code with a thermal model. Each iteration, the thermal model is fed the calculated heat transfer temperature from the CFD code, and uses an empirical convective heat transfer coefficient for each surface to recalculate values of wall heat flux that are then fed back to the CFD code. However, the choice of suitable empirical local convective heat transfer coefficients (i.e. between the wall and first row of grid points) is open to debate.

  The first two methods are suggested for fine grid analysis. For these two methods it is vital to refine the near-wall grid, as described in the next point. The third method averts the need for a temperature wall function, and so is the best choice for coarse grid analysis.

  Conventional wall-functions are inadequate for modelling natural convection boundary layer flow. The results are not independent of near-wall grid resolution, so the user should fine-tune the grid size. Our study suggest an optimum value for y^+ 30. More research is needed before improved wall-functions for natural/mixed convection can be widely adopted.

- Care should be taken to refine the computational grid in regions of locally steep gradients. Automatic grid generation (adaptive grid methods) makes this easier. Steady-state supply jets should be modelled using either the box method or the prescribed velocity method. This reduces the number of grid points needed to model the atrium.

  It is vital to account for heat transfer by surface-to-surface radiation exchange.

- It is important to model the thermal capacity of an atrium's building structure. This is most simply done by carrying out a steady-state CFD simulation of the worst-case (design) condition, using quasi

8.6 Summary

Computational fluid dynamics - CFD seems to be a very promising tool in the design process for atria and other large rooms. The continuous development of computer technology and software and the decreasing costs for computing, makes the CFD a must for future design.

CFD-simulations seem to have the most cost-effective use for verification of indoor environment and for trouble shooting. That is for the last stage in the design process before construction. At previous stages in the process, when information is more scarce, experience and simpler tools should be used to keep costs at a low level.

The objective of this chapter is on one hand to explain why CFD has been used in the Task 12 and in the other hand to bring attention to and to give guidance to the most essential factors in achieving a realistic and accurate solutions from CFD-simulations of atria.

Greek symbols

α	molecular thermal diffusivity ($= k/\rho c_p$) [m^2/s]
α_t	turbulent thermal diffusivity [m^2/s]
β	coefficient of thermal expansion [K^{-1}]
ε	turbulence energy dissipation rate
ν	molecular kinematic viscosity ($= \mu/\rho$) [m^2/s]
ν_t	turbulent kinematic viscosity (eddy kinematic viscosity) [m^2/s]
ρ	fluid density [kg/m^3]
ρ_0	reference fluid density [kg/m^3]
σ_t	turbulent Prandtl number, usually =0.9
$\sigma_{k,\varepsilon}$	empirical coefficients in the k-ε turbulence model

ISO 7730. Moderate thermal environments - Determination of the PMV and PPD indices and specification of the conditions for thermal comfort, 1990-10-05 (E)

Jones, P.J. & Whittle, G.E. 1992. Computational fluid dynamics for building air flow prediction : current status and capabilities. Building and Environment, Vol.27, No.3 (July), pp.321-338

Kondo, Y. & Niwa, H. 1992. Numerical study of an atrium by means of a macroscopic model and k-e turbulence model. Proc. Int. Symp. Room Air Convection and Vent. Effectiveness : Soc. Heat. Air Cond. Sanitary Engs. of Japan, Tokyo, Japan. 22-24 July 1992. pp.109-113

Launder, B.E. & Spalding, D.B. 1974. The numerical computation of turbulent flows. Computer methods in applied Mechanics and Engineering, Vol.3, pp.269-289

Li, Y.; Fuchs, L. & Holmberg, S. 1991. An evaluation of a computer code for predicting indoor airflow and heat transfer. Air Movement and Ventilation Control within Buildings, Proceedings of 12th AIVC Conference, Ottawa, AIC, Vol.3, p.123-136

Li, Y.; Fuchs, L. & Sandberg, M. 1993. Numerical prediction of airflow and heat-radiation interaction in a room with displacement ventilation. Energy & Buildings, Vol.20, pp.27-43

Melikow, A.K. 1988. Quantifying draught risk, Br& Kjær Technical Review No.2, Nærum, Denmark.

Moser, A. 1992. Numerical simulation of room thermal convection - review of IEA Annex 20 results. Proc. Int. Symp. Room Air Convection and Vent. Effectiveness, Tokyo : Soc. Heat. Air Cond. Sanitary Engs. of Japan. 22-24 July. pp.77-86

Murakami, S. & Kato, S. Numerical and experimental study on room airflow - 3D predictions using the k-e turbulence model. Building and Environment, 1989, vol.24, no.1, p.85-97

Murakami, S. 1992. Prediction, analysis and design for indoor climate in large enclosures. Roomvent'92. Proceedings of Third International Conference, Aalborg, Denmark : DANVAK. September 1992. Vol.1, pp.1-30

Nielsen, P.V. 1992. Description of supply openings in numerical models for room air distribution. ASHRAE Transactions, Vol.98, Part.1, pp.963-971

NS 3031. 1987. Calculation of buildings' energy consumption and demand, for heating and ventilation, (in Norwegian), Norwegian Standards Institution (NSF), 4th ed., May 1987.

Off, F.; Schälin, A. & Moser, A. 1994. Numerical simulation of air flow and temperature in large enclosures with surface radiation exchange.

9. Building energy simulation programs

9.1 Introduction

This chapter contains a short description of the Building Energy Simulation Programs used in this study. The simulation programs are DEROB (Sweden), Fres (Norway), TRNSYS (developed in USA, used by Switzerland) and *tsbi3* (Denmark).

A building energy simulation model is a simplified description of the building. The simplification has, in this case, been carried out from an energy and indoor climate point of view, so that the model only describes the aspects which are relevant in this connection. This means that it only includes data which form part of the various mathematical formulas and calculation algorithms, which together give an approximate description of the thermal and energy-dependent conditions regarding a building, its systems and operating conditions. As apparent from figure 9.1.1, the building model comprises the following:

- Data for the building's site, including climatic data.
- The building's form, i.e. its division into rooms or zones, delimiting surfaces for these zones, sub-surfaces consisting of windows and doors, as well as the materials used in them.
- Systems and loads in the building, including operating conditions and schedules.
- Data describing patterns of operation and use as well as other conditions for the individual zones.

The heat balance for the air in a zone does not make allowance for the heat capacity of the air which means that the air immediately adjusts itself to alterations in the surroundings. The following influences on the air's thermal condition are differentiated:

9.2 DEROB-LTH

DEROB-LTH which is an acronym for Dynamic Energy Response of Buildings, is a family of 6 modules calculating energy consumption for heating, cooling and ventilation.

DEROB is a flexible simulation tool using an RC (Resistance-Capacitance) network for thermal model design.

DEROB simulates buildings of arbitrary geometries and interprets the presence of shading devices.

The modules were originally developed at the Numerical Simulation Laboratory, School of Architecture, University of Texas, Austin. Since 1985, the DEROB modules are further developed to suit the local needs at the Department of Building Science at Lund Institute of Technology.

Below is a short description of properties of the DEROB modules.

9.2.1 Energy transmission

Walls are made of different materials with different thicknesses and thermal properties. DEROB divides the walls into a suitable number of layers and assigns internal nodes to the wall. A maximum of 7 nodes can be assigned inside each wall. Thermal properties for the walls are assigned by input or by material library. The inner and outer surfaces of the wall are each assigned one thermal node.

Windows are modelled with two surface nodes regardless of the number of panes. Thermal properties are assigned by input or by a windows library. The thermal resistance, including inner and outer film coefficients, is given as input.

Outer surfaces are coupled to other thermal nodes as follows:

- by conduction to the outermost of inner nodes in the wall
- by long wave radiation to the sky and ground
- by convection to the outdoor air

In the heat balance equation, loads from direct, diffuse and ground reflected solar radiation are taken into account.

Inner surfaces are coupled to other thermal nodes as follows:

- by conduction to the innermost of the inner nodes in the wall
- by long wave radiation to the inner surfaces in a volume
- by convection to the indoor air

9.2.4 Infiltration and air movements

Infiltration between a volume and the outdoor air can be modelled. The infiltration is specified by giving values for air change rate according to a time schedule.

If the building has more than one volume, air movements through advection connection between volumes are modelled. The air exchange is driven by the difference in temperature and static pressure.

9.2.5 Ventilation

Ventilation between volumes and to the outdoor air can be modelled. The direction of the forced ventilation can be defined. The sum of all air flows into a volume must be equal to the air flow out from the volume.

9.2.6 Internal gains

Internal loads including people, lighting and appliances can be specified in two ways. The simplest way is to specify a constant value that will be used during all days of the simulation period. The second alternative uses hourly values according to a time schedule. All values can be positive or negative.

9.2.7 Heating and cooling

Heating and cooling can be modelled. Two set points are specified according to an hourly schedule. The equipment used, can be assigned a maximum power. If not defined, the power is supposed to be unlimited, and the temperatures will then always satisfy the given set points.

9.2.8 Other systems

No modelling

9.2.9 Daylight

No modelling

9.2.10 Moisture

No modelling

FRES is developed at SINTEF Division of Applied Thermodynamics, Norway through the years 1988 to 1992. The description below corresponds to the version released in the spring of 1993.

FRES is based on a simplified description where a building is divided into elements. The elements are called thermal nodes as they represent average temperature within the boundary of one element. Several interconnected zones which are connected to a ventilation system can be studied. Equations for conservation of flow, heat and species are formulated for each node. The equations are solved by a finite difference method, hour by hour, for a predefined period. In northern climates it is normally not necessary to consider air stratification when calculating energy consumption (heating requirements and stratification occurs at different times). When the stratification is used as part of the heating strategy, it must be modelled.

A short description of the physical processes modeled is presented.

9.3.1 Energy Transmission

Walls are subdivided into layers to solve one dimensional heat transfer. There is no special modelling of thermal bridges or corner effects. Normally there are four thermal nodes, two at the inner and two at the outer surface. Thermal mass is modelled for the two nodes inside the wall. These nodes are normally 2.5 cm behind the surface. A thermal resistance is defined between the nodes in the wall. The inner surface node is connected to solar radiation from the window, to the other surfaces by long-wave radiation and to the air by convection. The outer surface is connected to the air by both convective and radiative coefficients. No solar radiation or long-wave radiation to other surfaces is modelled at the outer surfaces. All coefficients are constant.

Windows are modelled in a similar way with two thermal
absorbed part of the solar radiation is absorbed in the inner surface.

The method is acceptable for the design of most walls. With very low U-values, the relative importance of thermal bridges and corners increase, and these effects should be incorporated. The models do not provide the possibility to study movable insulation.

9.3.2 Solar Radiation and Distribution

The model performs hour by hour calculation of direct and diffuse solar radiation. The diffuse part is isotropic. Clouds are modelled with cloud-cover factors. The model works well, but the results seem to give a bit (about 5 %) too high solar radiation. As we get the new climate data

9.3.4 Infiltration, stratification and air movements.

The infiltration is calculated based on air change rate for zone air. It is strongly simplified, as there is no modelling of physical laws. Since no input data exists for better models this is still a good choice. Infiltration is often the largest source of uncertainty in the calculations. The available input data are uncertain.

For cases where stratification may occur, a linear temperature distribution model is provided. Average room temperatures from the stratification model are used when calculating heat transfer through the building elements. The stratification model option is favourable for rooms with large ceiling height.

Air flows may be modelled between zones, between ambience and the zones and as a part of the ventilation system. The flows are defined by their path, flow rate and schedule. The method works well for analysing most common problems. Data are easily available when forced ventilation is studied. For natural ventilation, air flow must be estimated in advance.

9.3.5 Ventilation and air conditioning

Ventilation plants with heat exchanger, heater, cooler, humidifier and fans are modelled. Each component has one thermal node. Both air temperature and humidity is calculated.

The method is good for detailed design when plant dynamics is not very important. It lacks the integrated analysis of the heating system and of a heat pump; Some precalculations and estimates must be per formed.

Ventilation through leakages and hatches caused by wind or chimney effect are not modelled. Precalculations must be performed and data given as infiltration with a schedule.

9.3.6 Internal gains

Internal heat gains like lighting, persons and equipment can be defined and scheduled. The user defines radiative and convective part.

9.3.7 Heating and cooling

Heating and cooling can be modelled in each zone locally and in the central air conditioning plant.

9.3.12 Limitations

The program can use up to 120 kb memory for variables and data.

9.3.13 Input and Output

Input is given in a pop down menu system. It provides help and standard data.

Output from the calculations are hourly air and operative temperatures and power consumption. Energy consumption values for heating, cooling, ventilation, humidifiers and equipment in the simulated period are given. Curves showing accumulated temperature values are available.

9.4 TRNSYS Type 56 (version 1.3) : Multi Zone Building

9.4.0 Introduction

This component of the TRNSYS program models the thermal behaviour of a building having up to 25 thermal zones. In each zone, the temperature of the air is assumed fully mixed.

There are two ways to model the equipment for heating, cooling, humidification and dehumidification :

1. Energy rate method
2. Temperature level method

With the energy rate method a simplified model of the air conditioning equipment is implemented within the type 56 component. The user specifies the set temperatures for heating and cooling, set point for humidity control, and maximum cooling and heating rates. These specifications can be different for each zone of the building. If the user desires a more detailed model of the heating and cooling equipment, a temperature level approach is required. In this case separate components are required to model the heating and the cooling equipment. The outputs from the type 56 zone can be used as inputs to the equipment model, which in turn produce heating and cooling inputs to the type 56 zones.

The zone temperature is free floating in the comfort region where the power is zero. If the temperature of a free floaching zone is within the heating or cooling region at the end of a timestep, power is applied throughout the timestep so that the first zone temperature just reaches Tset. If the power required is greater than the maximum specified, then the maximum power is applied throughout the timestep and the zone temperature is again free floating.

In order to determine either energy demands and floating zone temperatures it is necessary to evaluate the terms of equation 1).

9.4.1 Energy transmission

The walls are modelled according to the transfer function relationship of Mitalas and Arsenault. The wall are subdivided into layers to solve one dimensional heat transfer.

The long-wave radiation exchange between the surfaces within the zone and the convective heat flux from the inside surfaces to the zone air are calculated using a method called "star network" given by Seem - (1987). Area ratios are used to calculate view factors from one surface to the other so that this model is not a geometrical one.

Different types of walls can be defined :

External walls

Internal walls (play a role in solar and internal gains distribution and because of their thermal mass,- but there is no conduction losses associated with the wall because its both surfaces are in the same zone !)

Walls between zones

The temperature of the wall surfaces can be fixed.

A window is considered as a wall with no thermal mass, partially transparent to solar, but opaque to long-wave internal radiation and gains, and is always related with the outdoor conditions.

9.4.2 Solar radiation and distribution

The total solar gains to any zone i are :

For each wall separating zones of floating temperature or having a known boundary condition, it is possible to specify a convective coupling air movement from one zone to the other. This coupling is the mass flow rate that enters the zone across the wall. An equal quantity of air is assumed to leave the zone at zone temperature. Also this mass flow rate can be variable and calculated separately.

9.4.5 Ventilation and air conditioning

The ventilation rates are given in terms of air changes per hour for each zone.

The mass flow rate is the product of the zone air volume, air density, and air change rate. Infiltration occurs from outdoor conditions, while ventilation occurs from a specified (possibly variable as the rates) temperature. Equal amounts of air are assumed to leave the zone at the zone temperature.

The natural ventilation exchanges must be calculated separately and introduced in the zone as variable infiltrations or ventilation rates with ventilation temperature equals to the ambient (external) temperature.

9.4.6 Internal gains

The internal gains are defined as one convective part (in the air) and one radiative (longwave) part. Also these gains can be scheduled as variables. All surfaces are assumed to be black for radiation internal gains. These gains are distributed according to the area ratios.

9.4.7 Heating and cooling

This point has already been treated in the introduction, and we will not come back to it here.

Ratio = Multiplication factor generally in the range of 1 to 10. A moisture balance for any zone results in the following differential equation.

$$M_{eff,i} \cdot \frac{\delta \bar{\omega}_i}{dt} = \dot{m}_{inf,i}(\bar{\omega}_a - \bar{\omega}_i) + \dot{m}_{v,i}(\bar{\omega}_{v,i} - \bar{\omega}_i) + W_{g,i} + \sum_{walls,i-j} \dot{m}_{cplg,s}(\bar{\omega}_j - \bar{\omega}_i)$$

where

$\bar{\omega}_a$ = The ambient humidity ratio

$\bar{\omega}_i$ = The humidity ratio of the zone

$\bar{\omega}_{v,i}$ = The humidity ratio of the ventilation flowstream

$\bar{\omega}_{g,i}$ = Internal moisture gains

$\bar{\omega}_j$ = The humidity ratio of an adjacent zone j

In order to simplify the solution of the simultaneous set of differentiel equations, the values of at the end of the previous timestep are used in the above expression. If the average humidity ratio of the zone falls below or rises above a setpoint for humidification or dehumidification, then latent energy is added of removed to maintain the humidity ratio at the setpoint. It is assumed that the change in zone humidity ratio occurs instantly so that $\bar{\omega}_i = \bar{\omega}_{t,i}$. In this case

$$\dot{Q}_{lat,i} = h_v \left[\dot{m}_{inf,i}(\bar{\omega}_a - \bar{\omega}_{req,i}) + \dot{m}_{v,i}(\bar{\omega}_{v,i} - \bar{\omega}_{req,i}) + W_{g,i} + \sum \dot{m}_{cplg}(\bar{\omega}_{j,t-\Delta t} - \bar{\omega}_{i,t-\Delta t}) - \frac{M_{eff,i}(\bar{\omega}_{req,i} - \bar{\omega}_{i,t-\Delta t})}{\Delta t}\right]$$

where

$Q_{lat,i}$ = Latent energy removed (+ dehumidification, - humidification)

h_v = The heat of vaporization of water

$\bar{\omega}_{req,i}$ = The setpoint of humidification or dehumidification

Between the two setpoints, the humidity ratio is free floating.

- Gain due to convective coupling with adjacent zones
- Internal convective gains
- Change in internal sensible energy of zone air since beginning of simulation
- Humidity ratio of zone air
- Latent energy demand (-humidification, +dehumidification) (calculated only if heating/cooling defined)
- Net latent energy gains
- Total solar energy entering through windows
- Total radiation absorbed at all inside surfaces within zone, includes solar and gains, but not long-wave exchange with other surfaces
- Total radiation absorbed at all outside surfaces within zone, does not include long-wave from other walls

Other optional outputs are also available:

- Inside surface temperature
- Outside surface temperature
- Description
    - Inside surface temperature
    - Outside surface temperature
    - Energy from the inside surface including convection to the air and long-wave radiation to other surfaces
    - Energy to the outside surface including convection from the air and long-wave radiation
    - Total radiation absorbed at inside surface (except long-wave from other walls)
    - Total radiation absorbed at outside surface (except long-wave from other walls)

9.5.3 Infiltration

Infiltration is uncontrolled air penetrating through leakages in the building envelope. The model for infiltration defines the air-change rate for the current zone in three terms: a basic *air exchange* plus a term which is dependent on the difference between the inside and outside temperature plus a term dependent on the wind speed:

$$n_{venting} = n_0 + c_t \cdot (t_i - t_u)^p + c_v \cdot v \qquad (2)$$

where

n_o is the basic air change, h^{-1}

t_i, t_o are respectively the indoor and outdoor temperatures, °C

p is flow exponent on temperature difference (stack effect), often set to 0.5

c_t is a constant, which is especially dependent on the size of the room and openings as well as the height difference between the openings

is a constant, which is especially dependent on how leaky the building is, the building's geometry as well as the site in comparison with other buildings and the topography/roughness of the land

v is the wind speed, m/s

9.5.4 Solar distribution

The solar radiation transmitted through the windows in the zone, is distributed according to a "key" defined by the User in the following fractions:

- *Lost* is the part of the radiation which is reflected almost immediately back through the window or is in some way "lost" for the current zone.
- *To air* indicates the fraction of the total solar radiation to the zone that is assumed to be transferred to the air by convection. This fraction typically lies between 0.10 and 0.30.
- *To surfaces is* the part of the radiation distributed between the individual constructions in the floor, walls and ceiling. The total solar

much solar radiation can be accepted, but the shading can also be controlled according to the temperature in the zone. There are three possible control strategies for the shading device: *continuous*, *stepwise*, and *on/off* adjustment.

Buildings, trees etc., which at certain times of the year cast shadows on an actual building, can be described as "shadows". A shadow is defined by co-ordinates in connection with a window or a surface. On the basis of a description of the shadow in connection with the calculated solar path, it is possible to define how much the solar radiation through the window will be reduced.

In the connected schedule it is possible to describe variations in the shading coefficient over the day and over the year.

9.5.9 Simulation and results

Using hourly weather data (eg. TRY files) *tsbi3* simulates the multi-zone building and the dynamic interaction of the building fabric and the systems. During simulation day-values and hour-values of user-selected parameters are stored in files for later documentation and statistical analysis.

The recorded day-values are used to set up the complete energy balances for all of the zones and for the whole building. The energy balance can be documented in weeks, in months, or for the whole year.

During simulation *tsbi3* calculates up to several hundred parameters, and therefore the user has to determine which of these data should be recorded.

9.5.10 Documentation of data and results

The calculated hourly values of the *tsbi3* simulation can be documented in simple tables, treated statistically or presented graphically. The user decides which parameters should be presented in the tables or graphs, and he also determines the scaling of the axes.

$I_{t,s}$	=	The solar radiation incident upon the surface
A_s	=	Surface of the window
$M_{eff,i}$	=	Effective moisture capacitance of the zone
$M_{air,i}$	=	The mass of air in the zone
Ratio	=	Multiplication factor generally in the range of 1 to 10. A moisture balance for any zone results in the following differential equation.
$\bar{\omega}_i$	=	The humidity ratio of the zone
$\bar{\omega}_a$	=	The ambient humidity ratio
$\bar{\omega}_{v,i}$	=	The humidity ratio of the ventilation flowstream
$\bar{\omega}_{g,i}$	=	Internal moisture gains
$\bar{\omega}_j$	=	The humidity ratio of an adjacent zone j
$Q_{lat,i}$	=	Latent energy removed (+ dehumidification, -humidification)
h_v	=	The heat of vaporization of water
$\bar{\omega}_{req,i}$	=	The setpoint ofr humidification or dehumidification
n_0	=	the basic air change, h^{-1}
t_i , t_o	=	indoor and outdoor temperatures, respectively,°C
p	=	flow exponent on temperature difference (stack effect), often set to 0.5
c_t	=	a constant, which is especially dependent on the size of the room and openings as well as the height difference between the openings
c_v	=	a constant, which is especially dependent on how leaky the building is, the building's geometry as well as the site in comparison with other buildings and the topography/roughness of the land
v	=	the wind speed, m/s