5th Int. Conf. on Air Distributions in Rooms
ROOMVENT '96, July 17-19, Vol. 1, pp. 219-226, Yokohama, Japan, 1996.

A numerical comparison of three inlet approximations of the diffuser in case E1 Annex20

Peter Emvin[1] and Lars Davidson
Thermo and Fluid Dynamics
Chalmers University of Technology
S-412 96 Gothenburg, Sweden

ABSTRACT

A second order finite volume method with a locally refined structured grid was used. The method uses the QUICK scheme for the velocities and temperature and the HYBRID scheme for the turbulence. The turbulence model is a two-layer $k - \epsilon$ model.

We focused in the present work on investigating three different inlet conditions for a specific flow case, Annex 20 E1. The inlet in Annex 20 is a diffuser with 84 nozzles, which is difficult to handle geometrically. We therefore investigated how well the full diffuser can be represented by either a single jet or a decoupling of the mass and momentum boundary conditions. We performed calculations using these three inlet conditions and found they differ greatly. A discussion is presented of these boundary conditions and qualitative estimates of the related errors.

1 INTRODUCTION

In ventilation, an important parameter for study is the penetration length, meaning the distance X from the inlet at which a cold jet hits the floor. An uncomfortable situation may arise when the cold inlet jet hits a person in the occupied zone, for example. The person will feel a draft, since the temperature of this jet can differ several degrees from the surrounding temperature.

There are many parameters that affect the penetration length, such as the geometry, data of the inlet jet and other forces in the room. To study the fluid flow in a ventilated room, it is suitable to use a numerical method and simulate the flow. It is also important to have empirical rules to facilitate fast decisions and evaluate the numerical simulations. Finally, it is possible to make measurements on scale models or full-scale rooms, but it is expensive to construct an experimental rig, and sometimes also difficult to make accurate measurements. When simulating a ventilated enclosure, one often has a complex inlet consisting of several small jets. These interact for a short distance before they merge together into a large jet. It is costly to resolve all the inlet jets, and it is desirable to model the merging process. All the small inlet jets can then be represented by a simple boundary condition of a size similar to that of the diffuser. Adopting a mesh to the approximated inlet is then

[1]Peter Emvin changed name from Peter Johansson in September 1995

not difficult.

1 The first inlet model is a single jet in the center of the diffuser using an inlet area equal to the sum of the areas of the nozzles in the diffuser. The velocities are the same and therefore they give the same momentum and mass flux. This has earlier been referred to as the basic model (ANNEX20 1993).

2 The second inlet model is to resolve the diffuser and representing all 84 nozzles in the simulation.

3 The third inlet model is to decouple the boundary conditions of mass flux and momentum flux and apply it to an inlet with the same area as the total area of the diffuser. Since this area is larger than the sum of the areas of the nozzles the mass flux is not based on the nozzle velocity. This has earlier been referred to as the momentum model (ANNEX20 1993).

4 The fourth inlet model is to use the so-called box model (ANNEX20 1993), in which the inlet is replaced by a box around the inlet where data are prescribed around the box.

The box method of course offers the best possibilities for a good simulation with a if the measurements of the box data are better than than they can be simulated by the fully resolved method (2). Since we are dealing with elliptic equations, however, we must be careful in using such a method, as we must have the same feedback from the domain we are simulating, as we do from the one we have measured. This will place restrictions on the range of applicability and will also require accurate measurements around the box. Having noticed such lack of generality, we have not considered this boundary condition in our work.

2 CONTINUOUS MODEL

We used the two-layer $k-\epsilon$ model of Chen and Patel (Chen and Patel 1988). The equations read:

$$\partial_i(\rho U_i) = 0 \quad (1)$$

$$\partial_i(\rho U_i U_j - (\mu + \mu_t)\partial_i U_j + P\delta_{ij}) = F_j \quad (2)$$

$$F_j = [0, 0, -g(\rho - \rho_0)]^T \quad (3)$$

$$P_k = \mu_t \partial_i U_j(\partial_i U_j + \partial_j U_i) \quad (4)$$

$$G_k = -\mu_t g \beta \partial_z T/0.9 \quad (5)$$

$$\partial_i(\rho U_i k - (\mu + \mu_t)\partial_i k) = F_k \quad (6)$$

$$F_k = P_k + G_k - \rho\epsilon \quad (7)$$

$$\partial_i(\rho U_i \epsilon - (\mu + \mu_t/1.3)\partial_i \epsilon) = F_\epsilon \quad (8)$$

$$F_\epsilon = \epsilon(1.44(P_k + G_k - 1.92\rho\epsilon)/k \quad (9)$$

$$\partial_i(\rho U_i T - (\mu/0.72 + \mu_t/0.9)\partial_i T) = 0 \quad (10)$$

In the near-wall region, ϵ is prescribed via a mixing length approach.

3 NUMERICAL METHOD

We used the code PEC-FLTBB (Emvin 1996), which is based on our previous work. This is a collocated finite volume formulation using piecewise linear global polynomials approximating the solution of Equations (1)-(10). It uses boundary-fitted coordinates, and the second order QUICK scheme is used for the convective terms for the velocities and the temperature. For the turbulent quantities, we use the first order Hybrid-Upwind scheme for the convective terms. For the diffusive terms, we use a second order central difference stencil, and the source terms are evaluated by one point Gauss quadrature. A third order dissipation called PWIM or Rhie and Chow interpolation is added to the continuity equation to stabilize the pressure. This formulation has been shown to be approximately second order in terms of velocity and temperature, since the QUICK scheme and the PWIM provide sufficient stability and that the feedback from the first order turbulent scheme is small (Johansson and Davidson 1995).

The discretization is made on a locally refined mesh in order to resolve local features. This local mesh refinement is based on a flux discretization over the interfaces (Emvin and Davidson 1996). The solution procedure is a Full MultiGrid method using the SIMPLEC-TDMA smoother (Johansson and Davidson 1994).

4 E1 ANNEX 20

In (ANNEX20 1993), case E1 is specified and is briefly described below. It is a 3D-ventilated enclosure with a height $H = 2.5m$, a width $B = 3.6m$ and a length $L = 4.2m$. It has an inlet diffuser HESCO KS4W205K390 mounted 20 cm below the ceiling, symmetrically placed between the side walls. It consists of 84 nozzles $(D = 12mm)$ inclined 40 degrees towards the ceiling. The outlet ($H_{out} = 0.2m$ and $B_{out} = 0.3m$) is located on the same wall, 170 cm above the floor symmetrically between the side walls. On the opposite wall, a window ($H_{win} = 1.6m$ and $B_{win} = 2.0m$) is symmetrically located $0.7m$ above the floor.

The volume flux of the jet is $\dot{V} = 0.0158m^3/s$, the inlet temperature $T_{in} = 10^oC$ and the window temperature $T_{win} = 30^oC$. The walls have a constant temperature of 21^oC, except for the front wall, which has a temperature of 22^oC. This is a very difficult flow case to simulate (ANNEX20 1993), as the cold inlet jet should drop and hit the floor before it has reached the opposite wall. It has low physical stability, but the major challenge in this case is to handle the complex inlet geometry in an accurate and efficient way.

5 INLET MODELS

Simulating a diffuser is costly because it requires a fine resolution in the region near the inlet. We therefore used the local mesh refinement concept, and thus no unnecessary refinement will be made. Even so, the method results in large calculations. It is therefore desirable to replace a full resolution method with an inlet approximation which will account for the effects of the inlet, admitting the use of a coarser mesh in the inlet region.

The first approach would be the basic model, in which all jets are simply assembled and placed at the linear center of the diffuser plate. This model will have the severe restriction of giving a viscous area that is much too small and the jet will therefore penetrate too far into the room. An integral estimate of that is given in a later subsection. There are also variants of this method in which the shape of the modeled inlet is not scaled from the diffuser plate but is stretched for example in the horizontal plane. This can then account for some of the initial spreading of the jet as well as increase the entrainment. Numerical diffusion owing to poor resolution or a first order scheme could probably also give an increased diffusion and thereby compensate somewhat for the poor inlet model. To rely on such an assumption would be very dangerous,however, and it is questionable whether it is justified to allow numerical errors to compensate for errors in the boundary conditions.

The second simple approach is the momentum method. Here, all the inlet jets are replaced by decoupling the momentum and mass boundary conditions over the diffuser plate. This is also a very dangerous way of circumventing the resolution that is provided of the inlet diffuser because here the entrainment cannot be represented properly. The following example will show that this method is inconsistent in the limit of zero mesh size and will only be fruitful on very coarse meshes. On a finer mesh, the momentum method in practice degenerates into an inlet boundary condition in which the mass flux and momentum flux are not decoupled. The velocity is then the same as the scaled velocity used to specify the mass flux in the momentum method. Therefore, the major part of the U-momentum could be lost.

5.1 An example of the momentum method.

Assume an inlet diffuser with an inlet area A_1, a plate area A_2 and $U_{inlet} = U$ mounted on the $Y - Z$ wall in a room. Consider a control volume which extends one cell in the $X-$direction, just enclosing the inlet. Assume also $\rho = 1$ for simplicity. The momentum method then gives a modeled inlet mass flux of $\dot{m_{mi}} = UA_1$ and a modeled inlet velocity of $U_{mi} = U^2 A_1/(\dot{m}_{mi}) = U$. Using a finite volume formulation, we will require a local flux conservation of mass and momentum. Denote the velocity of the exit boundary of the volume U_f, the mean entrainment velocity U_e and the entrainment area A_e. Conservation of mass then gives:

$$-UA_1 + U_f A_2 - U_e A_e = 0 \qquad (11)$$

In principle a free jet has constant momentum which would require:

$$-U^2 A_1 + U_f^2 A_2 \simeq 0 \qquad (12)$$

If the mesh is compressed in the X direction in order to resolve the boundary layer along the vertical inlet wall, the thickness of the control volume will be small as compared with the diffuser plate, whereby $A_e/A_2 \sim 0.001$ and also typically $A_1/A_2 \sim 0.1$. That would give $U_e = 216U$, which is an unrealisic velocity, and the (numerical) diffusion will not allow that to occur. In practice, $U_e \sim U$ is more realistic, and then $U_f \simeq 0.1U$, whereby 90% of the momentum is lost.

Thus the momentum boundary condition on a compressed mesh is replaced in practice by an inlet velocity of $U_f = UA_1/A_2$ and a large inlet pressure gradient. The results will therefore be more or less the same as with the use of U_f as the inlet velocity instead. This was confirmed in practical calculations. However, as this is such a poor model of the inlet we have chosen not to report any of the results here Nevertheless, it should though be mentioned that it is of course possible to choose a sufficiently coarse mesh in the X direction and thereby obtain all the necessary entrainment in the first cell layer. Such a coarse mesh will result in large numerical errors and is not recommended. In the limit of the zero mesh size, the effect of the momentum inlet approximation will be the same as in an infinitely compressed mesh. Thus we conclude that the momentum method is inconsistent and can only be used to achieve rough estimates for coarse meshes.

5.2 Integral estimate of the basic inlet model

In this subsection, we will make an estimate of the difference in penetration length using the basic inlet model as compared with the fully resolved method. We must first estimate the total effect of all inlet nozzles. Since they are separated by three diameters, they will form by the action of diffusion a homogeneous flow within a short distance ($\sim 10 - 20$ diameters).

The action of the diffusion will require an entrainment due to conservation of momentum. We chose a volume enclosing the diffuser that was sufficiently thick to have a sufficiently uniform flow at the outlet. Conservation of momentum on this volume reads:

$$U_1^2 A_1 = U_2^2 A_2 \Rightarrow \frac{U_1}{U_2} = \sqrt{(\frac{A_2}{A_1})} \qquad (13)$$

We define the room temperature as reference and neglect kinetic and potential energy. Conservation of energy then reads:

$$\Delta T_1 U_1 A_1 = \Delta T_2 U_2 A_2 \Rightarrow \quad (14)$$

$$\frac{\Delta T_1}{\Delta T_2} = \frac{U_2}{U_1}\frac{A_2}{A_1} = \sqrt{(}\frac{A_2}{A_1}) \quad (15)$$

Now we have replaced the diffuser by a homogeneous inlet flow, U_2, A_2, T_2. We wish to estimate how much longer the penetration length will be with the use of the basic inlet model U_1, A_1, T_1 instead of the diffuser. We must here make an assumption in order to proceed further.

For a rigid body neglecting air resistance given $U, h, \Delta\rho$, the distance traveled, X, is

$$X = U\sqrt{(}h/2g\Delta\rho) \Rightarrow \quad (16)$$

$$X \sim U/\sqrt{(}\Delta\rho) \quad (17)$$

Linearizing the equation of state gives $\Delta\rho \sim \Delta T$.

Assuming the jet trajectory to scale similarly with respect to the buoyant force and noting that the entrainment temperature is approximately the same as the bulk temperature in the room, we get

$$X \sim U/\sqrt{(}\Delta T) \quad (18)$$

Comparing a single inlet jet with the diffuser gives an estimate of the attachment length as

$$\frac{X_1}{X_2} = \frac{U_1}{U_2}\sqrt{(}\frac{\Delta T_2}{\Delta T_1}) \Rightarrow \frac{X_1}{X_2} = \sqrt[4]{(}\frac{A_2}{A_1}) \quad (19)$$

In our case, $A_2/A_1 = 1/16$, which implies that we should expect $X_2/X_1 = 0.5$. Indeed, in our simulations, we arrive at $X_2/X_1 = 1.35/3.1 = 0.44$. This is of course much due to a happy coincidence under the assumptions given.

This case is stated in (Nielsen 1995) where the penetration length (as defined above) is split into one distance, X_s, in the place in which the jet is attached to the ceiling, owing to the Coanda effect, and another distance, ΔX_s, which is the distance the jet traveled after leaving the ceiling and reaching the floor. An estimate of X_s is given by (Grimitlin 1970).

$$X_s \sim \sqrt{(}A/Ar) \Rightarrow X_s \sim constant \quad (20)$$

since $Ar = g\beta\sqrt{(}A)\Delta T/U^2$. A rigorous analysis (Koestel 1955) gives an estimate of the jet path ΔX_s. It reads:

$$\Delta X_s \sim \sqrt[3]{(}1/(\sqrt{(}A) \times Ar))\sqrt{(}A) \Rightarrow \quad (21)$$

$$\Delta X_s \sim constant \quad (22)$$

This states that we should expect the same penetration depth using the different inlet models, and that is not verified by our calculations. We are of course well aware that our assumption resulting in $X \sim \sqrt[-4]{(}A)$ is crude, and the good agreement coincidential. On the other hand, the more sophisticated analysis given, does not perform as well. The difficulty may be that there is an elliptic situation with feedback from the room which is not accounted for. This feedback might give an effect in our case that is similar to the errors in our crude assumption. The uncertainty in the feedback and the assumptions in the global calculations are difficult to estimate.

Thus we turn to measurements to see whether our predictions are reasonable. Since it is X_s that differs most in our simulations, we focus on that. (Nielsen 1995) gives measurements of X_s. When performing a curve-fit we obtain $X_s/\sqrt{(}A) \sim Ar^{-3/8}$. Inserting this into our integral estimate would give us a shorter penetration length when using the basic inlet model in contradiction to our simulations. However, there is a inlet shape constant in the formulas of Grimtlin and Koestel, and it can easily vary by a factor of two. This may explain the poor bad agreement we achieved using the more rigorously developed formulas.

We thus conclude that such estimates as are given above must be used with care, which is the same conclusion as in (Nielsen 1995).

6 SIMULATIONS

The simulations were performed with the code PEC-FLTBB (Emvin 1996). We simulated case E1 Annex 20 using the basic inlet model, momentum inlet model and a fully resolved inlet diffuser. With the momentum method, it is possible to achieve more or less any kind of results by varying the first cell thickness, since this inlet model is inconsistent (see previous section). Therefore we do not show any results using the momentum method.

The basic model was used in the first calculations performed. Here we use a $58 \times 58 \times 58$ mesh expanded to resolve the boundary-layers down to $Y^+ = 0.5$. We also performed calculations on a twice as coarse mesh and, as the results did not differ significantly we concluded that we had sufficient resolution. In the basic inlet model, the inlet diffuser is represented by a scaled rectangular inlet ($H_{in} = 0.062m$ and $B_{in} = 0.18m$) in order to obtain the equivalent mass and momentum fluxes. The inlet boundary conditions of the turbulence kinetic energy and the dissipation of turbulent kinetic energy are set as $k_{in} = 0.01 \times (U^2+W^2)$ and $\epsilon_{in} = 0.00016 \times (U^2+W^2)^{1.5}/d_h$, where d_h is the hydraulic diameter of the inlet.

When simulating the case with all nozzles resolved, we use a locally refined grid. The base mesh is a $50 \times 58 \times 34$ mesh with three levels of refinement. The first refinement level includes both inlet and outlet, while the second and third refinement include only the inlet region. With this mesh, the inlet can be geometrically resolved, and all 84 jets can be implemented. The jets in this mesh are 2×2 cells wide, see Figure 2c. A calculation with each jet approximated by only one cell was found

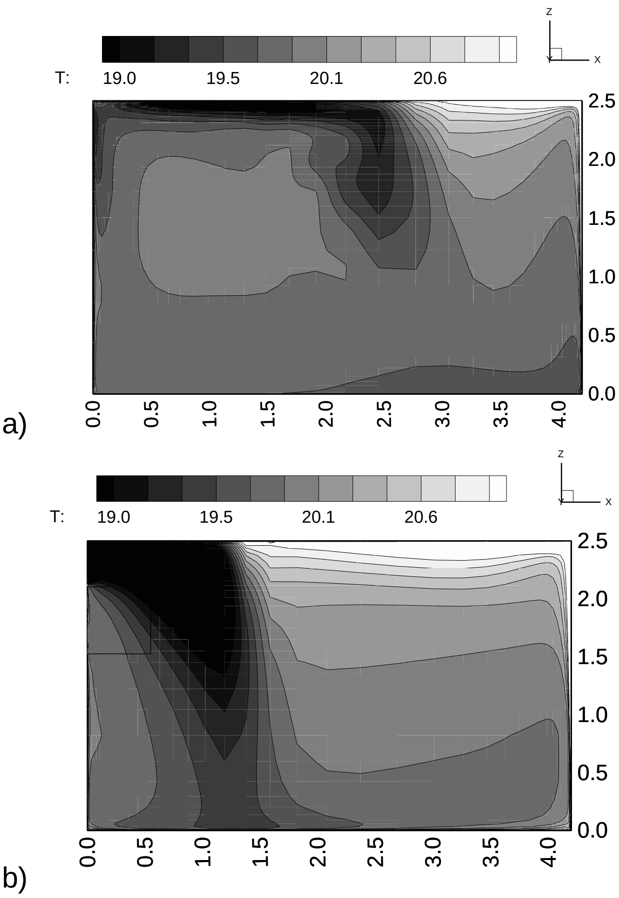

Figure 1: Contours of constant temperature. a) Basic model b) Full resolution method

to give similar results in the room except in a small region (O(d) d=jet diameter) close to the inlet. This indicates a sufficiently fine mesh in this region if the flow in the major part of the room is considered. The boundary conditions of k and ϵ are the same except that d is instead the nozzle diameter.

The calculations were found to differ considerably; for example, the penetration length using the basic model was twice as long as in the full diffuser simulation, see Figure 1. The flow is also symmetric in the case of full nozzle simulation, while it is not in the basic model. This indicates that the physics of many small jets cannot be modeled by a single large jet, since the information on the effect of the spreading of the inlet will not be correctly accounted for. We also would like to point out that measurements have been made on this flow case with the full diffuser as the inlet by (Blomkvist 1992; and Fossdal 1992). Unfortunately, their results differ as much as our two inlet models do, and thus no additional information is provided on these inlet models.

7 CONCLUSIONS

* The full representation of the inlet diffuser is accurate but expensive.

* The basic model of the inlet will give a too long penetration length and poor results for diffusers with small ratios of nozzle and diffuser area. Simulations on coarse meshes and poor numerical schemes can perhaps compensate via numerical diffusion the errors in the inlet model.

* The momentum method is singular in the limit of zero mesh size and will only perhaps work on coarse meshes, and then give only qualitative results.

* The box model is consistent and should perform as well as the full representation but will require measurements.

* Further work is needed in order to replace the full inlet by a simplified inlet model without performing any measurements of parts of the domain or sacrifying accuracy.

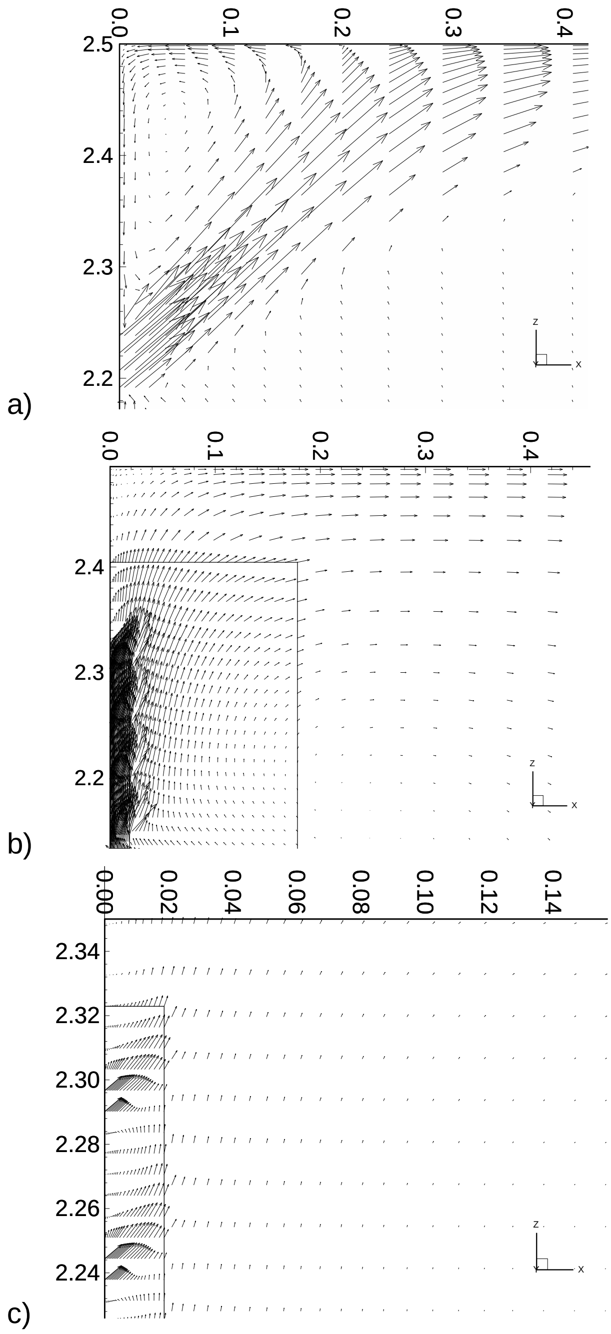

Figure 2: Vector plots of the inlet region. a) Basic model b,c) Full resolution method

ACKNOWLEDGMENTS

The support of the Swedish Research Council for Engineering Sciences (TFR) is gratefully acknowledged.

8 REFERENCES

ANNEX 20 1993, ANNEX 20, Room air and flow contaminant flow, evaluation of computational methods, Subtask 1 Summary Report, ed. A.D. Lemaire, TNO Building and Construction Research, Delft

Blomkvist C. 1991, Measurement of test case E (mixed convection, summer cooling), IEA Annex 20, Research Item 1.17, Tech. Rept. NSIBR, Gävle, Sweden

Chen H.C. and Patel V.C. 1988, Near-wall turbulence models for complex flows including separation, AIAA J., Vol.26, pp. 641-640

Emvin P., PEC-FLTBB, A CFD code for turbulent buoyant flow based on a local mesh refinement full multigrid strategy using boundary fitted coordinates, Rept. 96/7, Gothenburg, Sweden

Emvin P. and Davidson L. 1996, A local mesh refinement algorithm applied to turbulent flow, Submitted to Int. J. Num. Methods in Fluids

Fossdal S. 1990, Measurement of test case E (mixed convection, summer cooling), EA Annex 20, Research Item 1.17, Prel. Rept. NBRI, Oslo Norway

Grimtlin M., Zuluftvereilung in Räumen, Luft- und Kältetetechnik, Nr. 5

Johansson P. and Davidson L. 1994 A full multigrid method applied to turbulent flow using the SIMPLEC algorithm together with a collocated arrangement, Multigrid Methods IV, pp. 245-256, Birkhäuser Verlag

Johansson P. and Davidson L. 1995, Modified collocated SIMPLEC algorithm applied to buoyancy-affected turbulent flow using a multigrid solution procedure, Num. Heat Transfer, Part B, Vol.28, pp.39-57

Koestel A. 1955, Paths of horizontally projected heated and chilled air jets, ASHAE Transactions, pp 213-232

Nielsen P. 1995, Lecture notes on mixing ventilation.,Dept. of Building Techn. and Structural Engineering, Aalborg University, ISSN 0902-8005 U9513