#4127

ROOM -VENT '90

A1

Scale effect in room air movement modelling

H.B. Awbi
M.M. Nemri
Napier Polytechnic
Edinburgh, U.K.

SCALE EFFECT IN ROOM AIR MOVEMENT MODELLING

H B Awbi and M M Nemri
Napier Polytechnic
Edinburgh, U.K.

SUMMARY

In this paper, a numerical procedure is applied to solve the two-dimensional Navier-Stoke's equations describing the flow in an air conditioned room using the finite volume method. The effect of turbulence is described by the k-ε turbulence model. The range of influence of Archimedes and Reynolds numbers on the air velocity and temperature distribution in the room is investigated using the numerical solution. Comparison of the numerical prediction is made with experimental data. The predicted results can be used as a guide for the physical modelling of air movement under more complex thermal conditions.

INTRODUCTION

The expected performance of an air distribution system in some cases can be predicted from past experience and established design procedures (i.e.predictive method) . However, in other cases and particularly when non-conventional methods of air distribution are employed, physical modelling and or mathematical modelling must be used to evaluate the room environment. Where the size of the building precludes a full-scale physical model (eg. atria, indoor stadia, theatres etc), tests are carried out on a reduced scale model. In practice, although a small scale factor (a length ratio of building to model) is preferable, this may not be realised due to the high cost of constructing large models.

For the results from reduced scale model tests to be applicable to the prototype, geometric, kinematic and thermal similarity between model and prototype must be achieved (1 to 5). Geometric similarity is a pre-requisite for any modelling investigation. For isothermal flows, geometric and kinematic similarity must be present and these can usually be achieved without too much difficulty. However, for non-isothermal flows all three similarity requirements should theoretically be present before a complete simulation of the flow in the building can be achieved. In practice, it is not possible to provide a complete similarity for a non-isothermal flow and as a result difficulty in interpreting the model results may be experienced. Such problems are naturally irrelevent in mathematical modelling, however, most available mathematical models have recently been developed and require validation.

In this paper, the influence of kinematic and thermal similarities on the air velocity and temperature distribution in a room are investigated numerically using a Computational Fluid Dynamics (CFD) computer program. The predicted results are compared with measurements obtained in a full-scale test room.

PHYSICAL MODELLING

To use the test data from a scaled model of a room or an enclosure for the air distribution design of the prototype, similarity of the flow pattern, velocity distribution and temperature distribution should be produced in the model. Previous investigators (3, 5) have shown that such similarity can only be achieved if geometric, kinematic and thermal similarity between the model and prototype exist. With the help of dimensional analysis, it can be shown, eg. Rolloos (3), that complete similarity can only be attained if the Prandtl number, Pr, the Reynolds number, Re, and the Archimedes number, Ar, are equal for both model and protoype and, in addition, geometric similarity and similarity of the boundary conditions are present. For a predominantly convective heat transfer in the room and assuming that geometric and boundary similarities exist and the same fluid is used in the model and the prototype (air) ie., Pr is the same, then equality of Re and Ar must be obtained to achieve complete flow similarity. Where radiant heat transfer at the boundaries is significant, the parameter To^4/Uo (where To is a reference temperature, K, and Uo is a reference velocity) must also be equivalent in the model and prototype in addititon to the other conditions for convective heat transfer. Assuming convective heat transfer and applying this similarity criterion to the supply jet means that

(i) **<u>For Re equality</u>**

$$Re = \left(\frac{UL}{\upsilon}\right)_p = \left(\frac{UL}{\upsilon}\right)_m$$

(1)

with the same fluid (air) used in model and prototype:

$$\frac{U_m}{U_p} = \frac{L_p}{L_m} = S$$

(2)

where S is the scale factor, U is the supply velocity, L is a characteristic length and υ is the kinematic viscosity of air.

Equation 2 indicates that the velocity in the model must be higher than that in the prototype by the factor S.

(ii) **<u>for Ar equality</u>**

$$Ar = \left[\frac{g\,\beta\,L\,\Delta\theta o}{U^2}\right]_p = \left[\frac{g\,\beta\,L\,\Delta\theta o}{U^2}\right]_m$$

where β = cubic expansion coefficient, 1/K

$\Delta\theta o$ = air temperature difference between supply and room, K

g = gravitational acceleration.

Assuming similar thermal conditions between the model and the prototype and using the same fluid:

$$\frac{U_m}{U_p} = \sqrt{\frac{L_m}{L_p}} = \frac{1}{\sqrt{S}}$$

(3)

From equations (2) and (3), it is clear that the requirements for the equality of Re is quite different from the requirements for the equality of Ar and the two equalities can never be achieved concurrently in a model study. In the case of isothermal flows similarity can be achieved with constant Re. However, for non-isothermal flows complete similarity cannot be achieved in practice. In this case the ranges of Re and Ar over which the air distribution system in the prototype is required to operate can provide an insight into deciding whether equality of Re or Ar is most relevant for model investigation.

NUMERICAL SOLUTION

The general equations describing the steady incompressible flow in a room are solved in a finite difference form. The fluctuating velocities and temperature terms are represented by an equivalent time-average terms using the k-ε turbulence model. The effect of buoyancy on the vertical component of velocity and the kinetic energy of turbulence, k, and its dissipation rate, ε, is also included in the solution procedure, see Awbi (7, 9) for further details.

TEST ROOM

The room being investigated for this purpose has a square floor of length, L = 4.2m and ceiling height, H = 2.8m. The air is supplied from a continuous slot in the ceiling spanning the width of the room and at a distance 1.2m from the wall. The room load was produced by electrically heated tapes laid over the floor area to produce a uniform load distribution. The air velocities were measured with TSI 1610 low velocity anemometers which give the magnitude of the velocity at the measuring point. The thermocouples were provided with a radiation shield to reduce the effect of radiant temperature. The measurements were carried out using square grids of 1.0 and 0.5m at distances of 0.15, 0.6, 1.2 and 1.8m above the floor.

RESULTS AND DISCUSSION

Effect of Reynolds number (isothermal flow)

Figure 1a shows resultant velocity profiles in the occupied zone of the room for different air flow rates ie. different Re. Here Re and Ar are defined with respect to the height of the supply outlet h, i.e. $Re=U_0h/\upsilon$ & $Ar = g\beta h\Delta\theta_0/U_0^2$. U_0 is the supply velocity and $\Delta\theta_0$ is the difference between supply temperature and average temperature in the occupied zone. These profiles represent the ratio of the mean velocities in horizontal planes of the occupied zone to the supply velocity. As can be seen in Fig. 1a the predicted velocity profiles are close to the experimental values, except near the floor where the predicted values are higher in some cases. This can be attributed, especially in the low Reynolds number cases, to the directional sensitivity of the anemometers. The anemometer had a cylindrical sensor which measures the component of velocity normal to the cylinder axis. The sensor is normally set with its axis parallel to the floor of the room. The measuring plane at 150mm above the floor is within the boundary layer region of the reverse flow and unless the axis of the anemometer is perpendicular to the direction of the flow, a low velocity reading will be obtained at this height. At higher levels above the floor the anemometer will be measuring flow in the shear layer and the vortex regions where the flow has no defined direction. Indeed measurements at higher levels produced good correlation with predictions. Another cause of discrepancy at low levels may be attributed to the $k - \varepsilon$ model's failure in not describing the effect of wall proximity accurately and also the effect of low Re flows near the wall in which case the flow may be in a state of transition. The wall effect has been included in the present computation using Launder and Spalding's (6) wall function expression which does not allow for the effect of transition. However, Nagano and Hishida (10) have reported an improvement to the k - ε model by the inclusion of both the effect of low Re and wall proximity and have attained better results. The present authors anticipate the use of improved forms of the k - ε model in the near future.

As shown in Fig. 1a the effect of increasing Re on the velocity profile is most prominent near the floor. Figure 1b shows this effect extended to include higher Re and by increasing Re the maximum recirculating flow velocity near the floor also increases. The range of Re considered here covers most values used in model investigation.

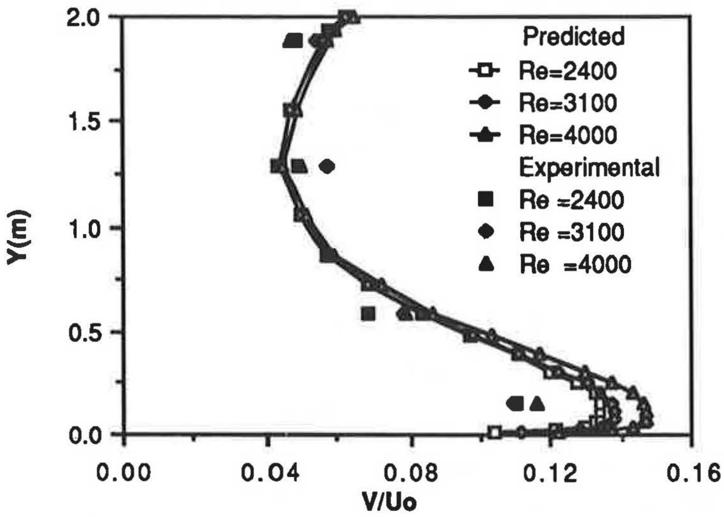


Fig.1a Computed and measured isothermal velocity profiles

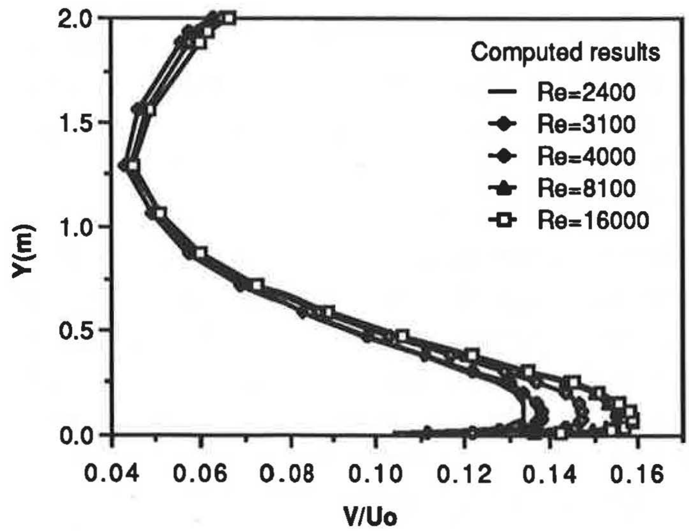


Fig 1b Computed isothermal velocity profiles

Effect of Archimeds number (non-isothermal flows)

Figure 2 shows the effect of Archimedes number on the mean velocity in the occupied zone. For cooling as Ar increases the room velocity increases as a result of the downward buoyancy acting on a cool jet. However, with heating a significant decrease in the mean room velocity occurs as Ar increases. The ratio Vr/Uo ranges between about 0.02 to 0.14 for a range of $-0.003 < Ar < 0.003$. This is due to the change in the room air movement pattern as will be discussed in the next section. This indicates that especially for higher Ar values, modelling at reduced scale should be based on the equality of Ar between model and prototype. The agrement between the predicted results and measurement is close for most practical purposes.

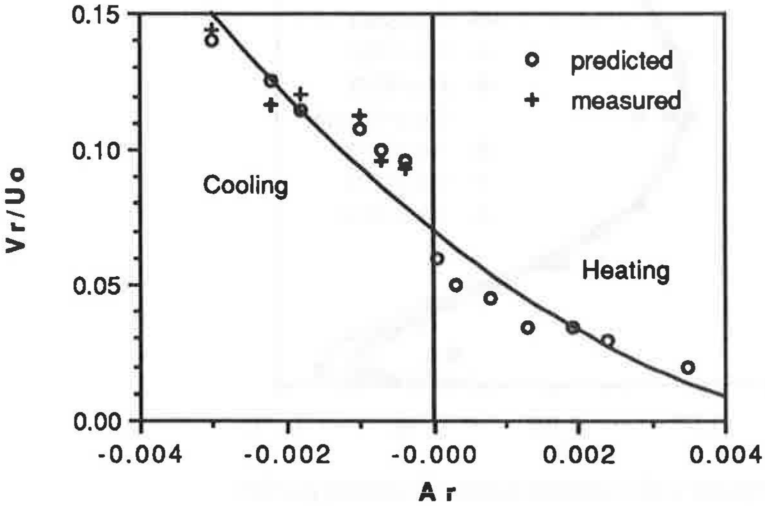


Fig.2 Effect of Archimedes number on the average velocity in occupied zone

Figure 3 shows the non-dimensional temperature distribution $\Delta\theta/\Delta\theta_0$ in the occupied zone for different values of Ar. $\Delta\theta$ represents the difference between the air supply temperature and the average temperature in a horizontal plane and $\Delta\theta_0$ is the difference between the supply temperature and the average temperature in the occupied zone . As can be seen from this figure the effect of Ar on the temperature gradient is very small which indicates a good mixing of the flow. Since the room load was situated on the floor, the temperature ratio increases towards the floor with $\Delta\theta/\Delta\theta_0 > 1$. Figure 4 shows a comparison of the same temperature ratio between cooling and heating using the predicted results. It can be seen that for the heating conditions the temperature gradient is steeper than it is for the cooling conditions. This is as a result of the poor mixing of the jet with room air caused by the upward acting buoyancy on the warm jet.

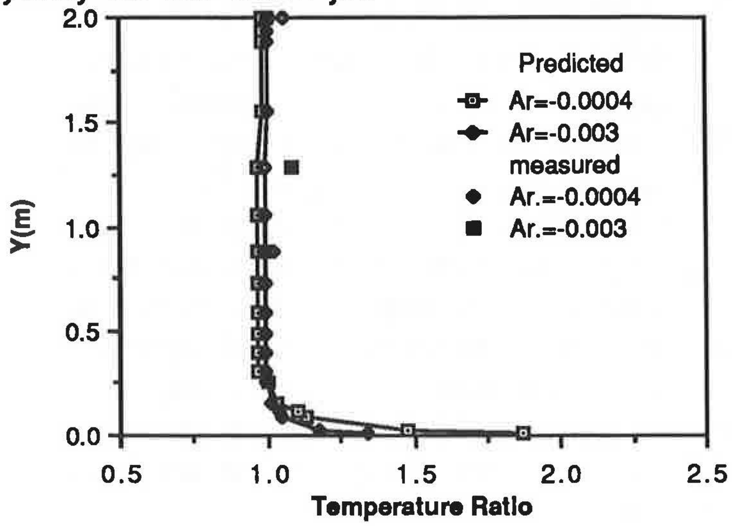


Fig.3 Temperature distribution in occupied zone for different Ar (cooling condition).

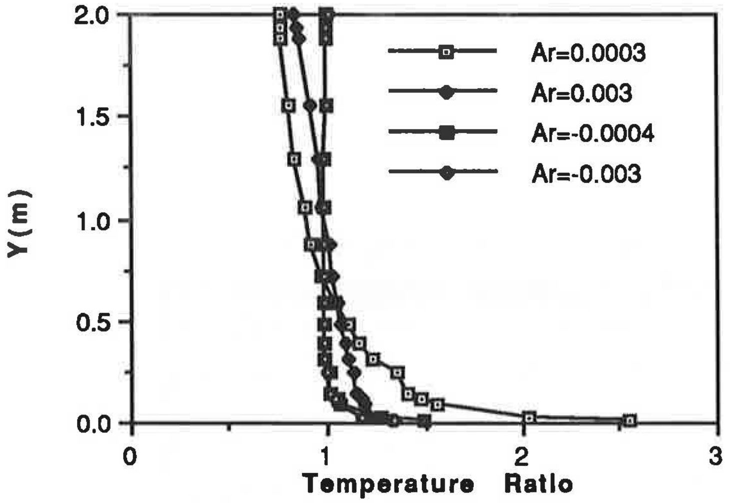
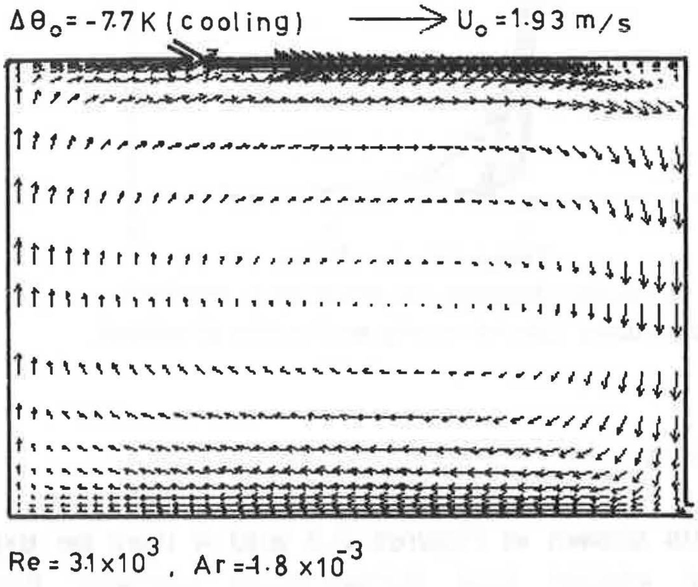


Fig.4 A comparison between the temperature distribution in the occupied zone for cooling and heating (Predicted).

Room Air Movement

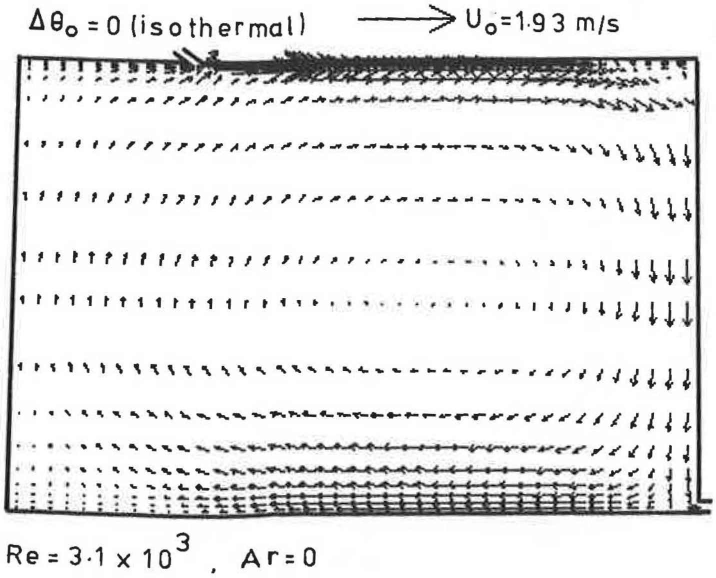
The results shown in Figures 2,3 and 4 may be explained using velocity vector and temperature contour plots for cooling, heating and isothermal conditions. Figure 5 (a,b and c) represent predicted velocity vector plots for cooling (Ar=-0.0018), isothermal and heating (Ar=0.0019) . The cooling and heating loads were uniformally distributed over the floor area and some heat transfer was also included between the walls and the room air. The total load for cooling is 132.4 W/m^2 of which 104.8 W/m^2 was over the floor. For heating, the total load is -130.8 W/m^2 of which -104.8 W/m^2 was also over the floor. The mixing of the jet with room air for cooling and isothermal flows is typical of a good high level air distribution system. However, the heating mode shows a totally inadequate mixing of the jet with room air whereby a stagnant zone is clearly present over most of the occupied zone. In addition, the effect of downward draught from the cold wall is also present over the floor. A vortex region at high level is responsible for poor distribution in the occupied zone which is created by the buoyancy acting on the warm jet and the lack of jet momentum.

The flow pattern described above is also reflected in the predicted temperature contours shown in Figures 6(a and b) for cooling and heating. Temperature stratisfication is clearly illustrated by Fig.6(b) for the heating mode condition but is not present in the cooling mode.

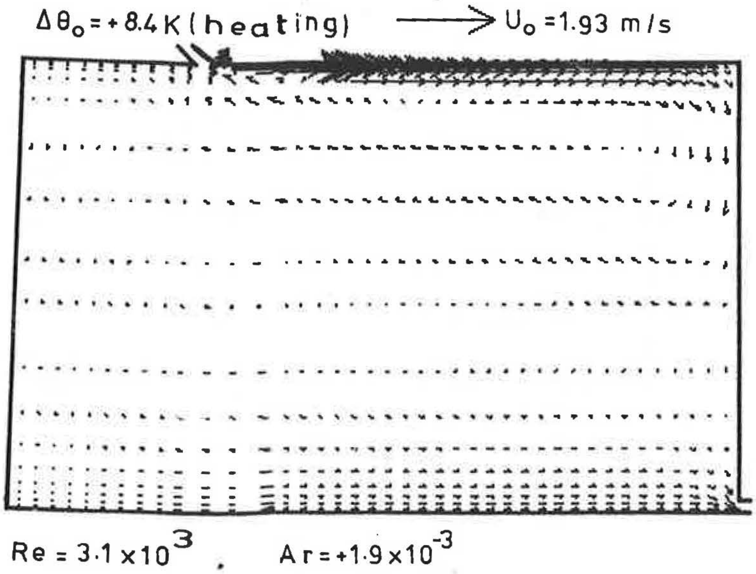


(a) cooling

Fig.5 Velocity vectors in the room

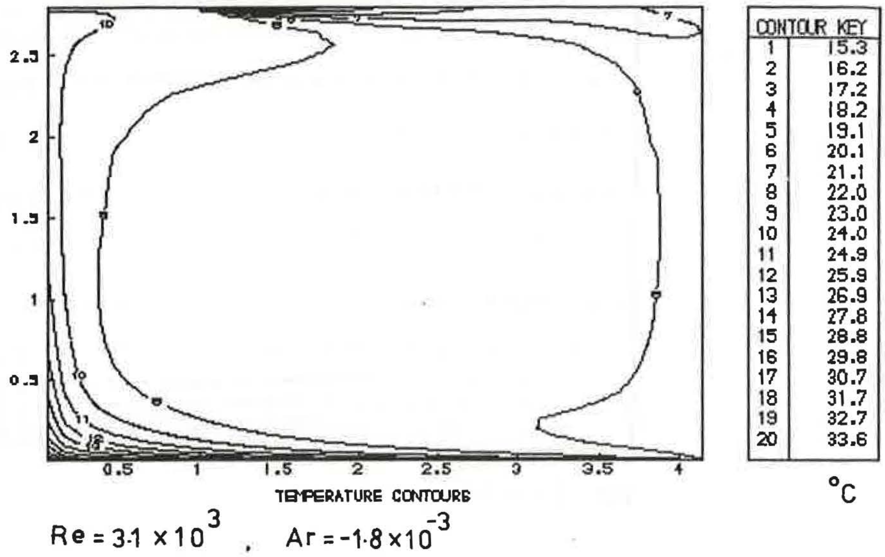


(b) isothermal

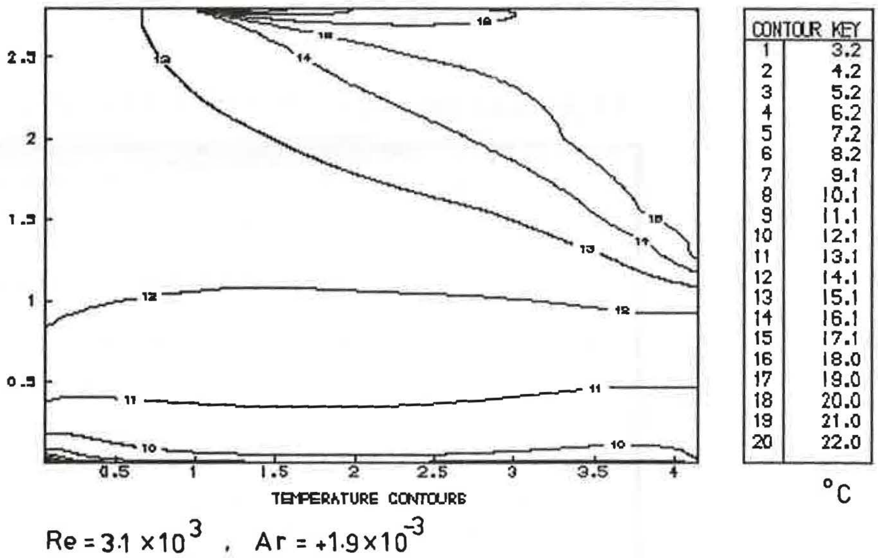


(c) heating

Fig _ 5 Velocity vectors in the room



(a) cooling



(b) heating

Fig. 6 Temperature contours in the room.

CONCLUSIONS

The results from this study show that when modelling isothermal flows in a room, it is important to perform the model test at the same Re as in the prototype, since Re was found to influence the velocity in the occupied zone. In case of a reduced-scale model this means a supply velocity equal to that in the prototype multiplied by the sale factor S, ie a higher velocity.

In the case of non-isothermal flows in which convection is predominant the velocity distribution in the occupied zone is affected by both Re and Ar, but it has been found that it is more important to perform the model test at the same value of Ar as in the prototype when $|Ar| > 1 \times 10^{-3}$ and to perform the model test at the same Re as in the prototype when $|Ar| < 1 \times 10^{-3}$. Theoretically, the effect of representing the correct boundary conditions in the physical model influences the room flow. Although this was not investigated in this paper it has been found from computer predictions that the distribution of thermal load on the room surfaces determines the air movement pattern in the room.

REFERENCES

(1). Parcewski, K.I., Renzi, P.N. Scale model studies of temperature distributions in internally heated enclosures. ASHRAE Journal 1963, 5, 7, 60-68, 84 and 89.

(2). Mullejans, H. The similarity between non-isothermal flow and heat transfer in mechanically ventilated rooms. Westdentscher Verlag, Koln and Opladen, 1966. (BSRIA English translation 202, Bracknell).

(3). Roloos, M. Possibilities and limitations for the prediction of air flow patterns, temperature and velocities in large halls using scale models. Meeting of Commission E1 of the International Institute of Refrigeration, Yugoslavia, 1977, 245-256.

(4). Linke, W. Eigenschaften der Strahlluftung. Kaltetechnik-

Klimatislerung, 1966, 18, 122-126.

(5). Moog, W. Room flow tests in a reduced scale. ASHRAE Trans., 1981, 87, 1162-1181.

(6). Launder, B.E. and Spalding, D.B. The Numercial computation of turbulent flows comput. methods appl. Mech. Eng. 1974 3, 269 - 289.

(7). Awbi, H.B. and Setrak, A.A. Numerical solution of ventilated air jet. Proc. 5th Int. Symposium on the Use of Computers for Environmental Engineering Related to Buildings, Bath, England 1986.

(8). Savin, S.J. Private communications. Martingale Technical Services Ltd., Penn, Buckinghamshire, England, 1986.

(9). Awbi, H.B. Application of computational fluid dynamics in room ventilation, Building and Environment, 1989, 24, 73-84.

(10). Nagano, Y. and Hishida, M. Improved form of the $K - \varepsilon$ model for wall turbulent shear flows, ASME Trans., J. Fluid Eng, 1987, 109, 156-160.