MULTI-GRID PREDICTION OF CONJUGATE HEAT TRANSFER AND AIR FLOW IN BUILDINGS

Yuguo Li*, Sture Holmberg, Laszlo Fuchs+**

**CSIRO, Division of Building, Construction and Engineering, Australia*

***Ventilation Division, National Institute of Occupational Health, S-171 84 Solna, Sweden*

#Department of Mechanics/Applied Computational Fluid Dynamics, Royal Institute of Technology, S-100 44 Stockholm, Sweden

ABSTRACT

The heat conduction through the walls changes the heat load and its distribution in a room, and thus affects the air flow pattern in a buoyancy-controlled ventilated room. This paper presents a methodology of how conjugate heat transfer and air flow in a room can be handled in an efficient computational fluid dynamics (CFD) code. The wall and indoor air regions are simulated simultaneously. The standard k-e model is used for modelling the turbulence in rooms. The transport equations take the same form in both fluid (air) and solid (wall) regions. The boundary conditions for the momentum and energy equations are specified at the outer surface of the walls. A multi-grid solver is applied to the problem of conjugate heat transfer and turbulent indoor air flow. The local grid-refinement technique is introduced to add local grid points in the regions where variable gradients are large. The multi-grid solver shows a much faster convergence rate than the single grid solver for the non-isothermal cases in this study. The developed approach is applied to a modelled 3D room and a room with displacement ventilation. This approach together with the previously developed general thermal boundary conditions, allows the study of interactions between indoor and outdoor environments.

INTRODUCTION

Air flow in buildings is characterised by flow recirculation, complex turbulence structure, coupling of buoyancy and turbulence, and coupling of indoor and outdoor environments. Over the last twenty years, prediction methods for indoor air flow have increasingly relied on computational fluid dynamics (CFD) based mostly on finite difference and finite element solutions of the Navier-Stokes equations in their time-averaged form with some turbulence models. The major challenges for application of CFD in indoor air flow include modelling the physics of the flow including turbulence, specifying realistic boundary conditions, representing the complex geometry of the room, and developing accurate and efficient numerical algorithms. All components may introduce significant errors in the prediction, see Fig. 1. This paper addresses the problems of how the conduction through walls can be included in a CFD code, and how the numerical efficiency in solving non-isothermal flows can be improved by a multi-grid method.

* P.O. Box 56, Highett, Vic. 3190, Australia
Tel: +61 3 252 6000 Fax: +61 3 252 6240
E-mail: yuguo@mel.dbce.csiro.au

The ventilation system is commonly used as an air distribution method for air-conditioning. In order to remove excess heat load or cooling load in a room, the supply air temperature is normally different from the room air temperature. Heat loss or heat gain by conduction through the walls is an important part of the load, but it is normally neglected in indoor air flow simulation. Li et al. (1991) have concluded that the generally used

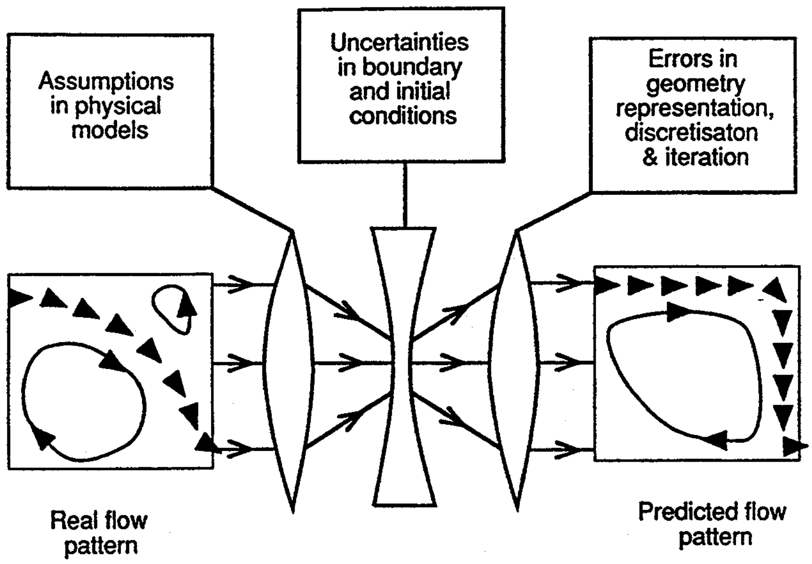

Fig. 1 Illustration of the basic error sources in a numerical simulation.

constant temperature or constant heat flux boundary conditions may not always be applicable. Experimental results presented by Li et al. (1993a) have shown that the temperature profiles in a room ventilated by displacement are influenced by the conduction through the walls.

At least two approaches are available to include the heat conduction through the walls in a CFD code. One is to use a general thermal boundary condition (Li and Holmberg 1993). The energy balance equation for a surface element is used iteratively to determine the surface temperature or heat flux during the calculation. Another approach is to enlarge the computational domain to include the wall regions. This presents a conjugate heat transfer problem in which both conduction in solid wall and convection in air are considered. The numerical treatment of this problem described previously by Patankar (1980) will be applied here.

The non-isothermal flow means a coupling of the momentum equations and the energy equation. In the IEA Annex 20* investigation, non-isothermal flow simulations need 5 to 10 times the computational time of the corresponding isothermal case when single-grid methods such as SIMPLE are used (Lemaire, 1991). This indicates a poorer performance of conventional single-grid methods for non-isothermal flows. The Multi-grid method has proved to be efficient in the calculation of elliptic PDEs and laminar and turbulent flows, see e.g. Peric et al. (1989), Bai and Fuchs (1992), Rubini et al. (1992). Application of the multi-grid method in both steady and unsteady heat transfer calculations can be found in Arnone and Sestini (1991). The multi-grid scheme can naturally be extended to include local grid refinement (Fuchs 1986). The application of the multi-grid method and local grid refinement in indoor isothermal flows has been reported previously by Li et al. (1991). This paper further develops the application of the multi-grid method together with a local grid refinement technique to three-dimensional non-isothermal flow with conjugate heat transfer.

* Annex 20 is an International Energy Agency (IEA) project on air flow patterns within buildings. Thirteen countries joined Annex 20.

MATHEMATICAL MODELS

Turbulent indoor air flow is modelled by the averaged Navier-Stokes equations with the Boussinesq approximation for buoyancy and the standard k-ε closure for turbulence. The turbulent heat transport is assumed to be proportional to the gradient of the mean field. The transport equations take the same form in both the fluid (air) and solid (wall) regions. The viscosity in the momentum equations is set equal to a large number (10^{10}) in the solid region to ensure zero velocity throughout the solid region. The boundary conditions for both the velocity and temperature fields are specified at the outer surface of the wall if required. The complete system of the partial differential equations and the boundary conditions can be found in Li et al. (1993b).

The wall function is used to consider the near-wall flows. Particular attention has been paid to implementing the wall functions for the multi-grid application. The usual approach in most single-grid methods is to calculate an "equivalent" wall slip-velocity, instead of using a non-slip condition. The "equivalent" viscosity, as first adopted by Agouzoul et al. (1989) is used here. Similarly, an "equivalent" diffusivity is used to replace the laminar diffusivity near the wall. The "equivalent" viscosity μ_{eq} and the "equivalent" diffusivity Γ_{eq} can be derived from the wall functions without further assumptions.

$$\mu_{eq} = \frac{\rho \kappa y_p U_\tau}{\ln(E y_{p+})} \quad (1)$$

$$\Gamma_{eq} = \frac{\rho y_p U_\tau}{Pr_t(U_{p+}+P)} \quad (2)$$

where ρ is air density, κ is the von Karman constant, U_τ is the friction velocity, E is a roughness parameter, y_p is the normal distance to the wall, y_{p+} is the dimensionless normal distance to the wall, Pr_t is the turbulent Prandtl number and P is an empirical function of (Pr/Pr_t).

The "equivalent" viscosity and "equivalent" diffusivity are stored in the same array as the effective viscosity and the effective diffusivity. The momentum equations and the energy equation are solved at the first grid point near the wall in the same manner as for other interior grid points.

MULTI-GRID METHOD AND LOCAL GRID REFINEMENT

The physical domain, including both air and wall regions, is described by different levels of global and locally refined grids. In the wall regions, different wall-cell-types are identified by an integer array, which might be a conductive cell, an adiabatic cell, or a temperature-given cell. The transport equations are discretized using finite differences on a staggered grid. The hybrid scheme is used, which employs second-order central differencing on both convective and diffusion terms but automatically modifies the convective diffusing procedure to an upwind formulation when the local cell Reynolds number exceeds two. A pseudo time derivative is used as an under-relaxation parameter. Distributive Gauss-Seidel (DGS) is used to treat the coupling between the momentum and continuity equations. The symmetric successive point relaxation is used for relaxation of all correction equations. The residuals of scalar equations and the scalar quantities are transferred from the fine to the coarser grid by volume averaging, while, due to the staggered grid, the restriction of each velocity component and the residuals of the momentum equations are done by area averaging. A tri-linear interpolation scheme is used for prolongation.

The Full Approximation Storage (FAS) algorithm (Brandt 1977), ideally suited for non-linear problems, is used. The multi-grid process used is a V-cycle algorithm. A five-level V-cycle algorithm with eight grids is depicted in Fig. 2. Each grid level is comprised of one or more sub-grids. The relaxation is performed sequentially on various sub-grids. For simplicity, the number of relaxation sweeps at each grid is fixed. The numbers may vary from level to level and grid to grid.

The boundary conditions (e.g. at inlets, outlets and walls) and the wall functions are applied only on the finest grids. The boundary conditions for coarse level grids are updated by the multi-grid restriction procedure. The interior boundary conditions of locally-refined grids are determined from the next coarser grid level using the current solution on that level. The same tri-linear interpolation as in the prolongation is used. As rectangular grids are used, the sum of mass flux on the finer grid equals the mass flux on the coarse grid.

Coarse grids play different roles in the multi-grid method for locally-refined grids than for only global grids. On the portion of any grid which has a finer grid above it, the coarse grid plays the role of a correctional grid for the fine one; while on the other portion without fine grids above them, the coarse grid plays the role of the finest grid. This portion of the grid should provide the solution to the flow problem. In our approach, this portion of the grid is located at levels equal to or higher than the finest global grid level.

It is not reasonable to terminate the calculation by monitoring only the magnitude reduction in the residual of all the equations on the finest grid, as is done in a multi-grid code with only global grids.

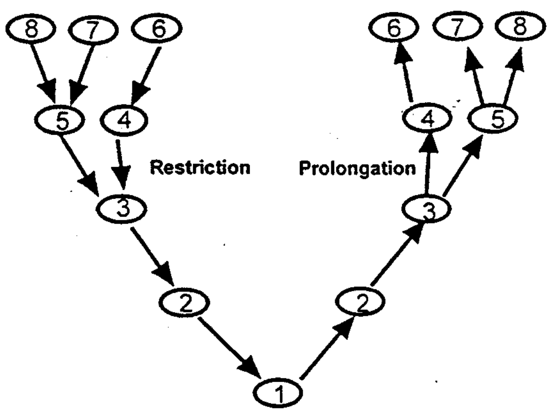

Fig. 2 A five-level V-cycle algorithm with eight grids.

The difference between a global grid and local grid in the solution is that the boundary conditions are all physical boundary conditions for the former and not for the latter. To guarantee that the solution on the local grid is the solution of the flow problem, not only does the residual need to be reduced below a certain tolerance, but the interior boundary conditions must also be correct. Therefore, the correction in updating these interior boundary conditions is also monitored. The calculation is terminated until both the correction and the residual are reduced to a certain tolerance (Srinivasan and Rubin 1992).

EXAMPLE CALCULATIONS

The first calculation is carried out on a modelled 3D room (5 x 5 x 3 m^3) with conductive ceiling, floor and internal body, see Fig. 3. Two side walls of the room are maintained at two different constant temperatures, 273.15K and 293.15K. The viscosity is adjusted to 1000 times the air viscosity in order to obtain a laminar case. The Rayleigh number based on the width of the room is 2. x 10^5. The flow and temperature fields at y= 1.25 and 3.75 m, and y= 2.5 m are shown in Fig. 4. The results indicate that the method is able to include the wall regions in the computation. This will make it possible to study the interaction between indoor and outdoor thermal environments. It should be mentioned that it is necessary to use the general flow boundary conditions in order to study the effect of infiltration, see Li and Holmberg (1993).

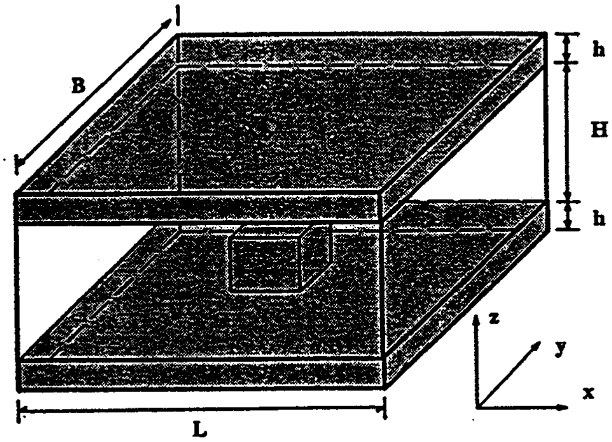

Fig. 3 A modelled 3D room with conductive ceiling, floor, and internal body.

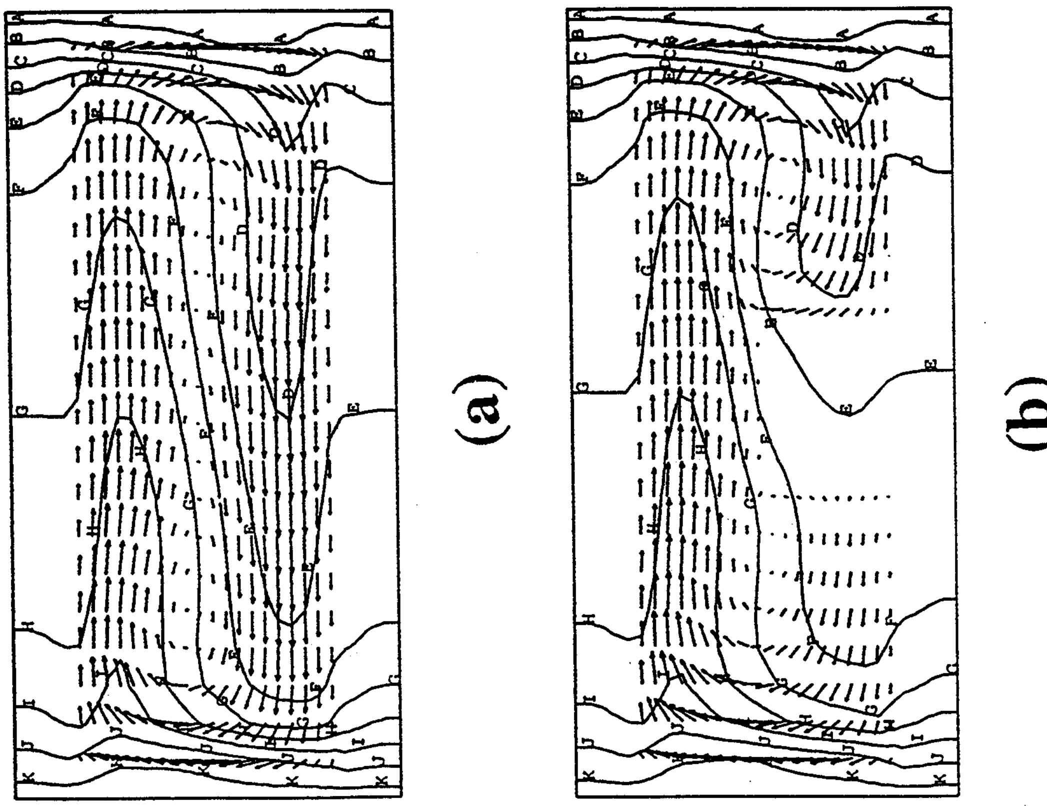

Fig. 4 Flow and temperature fields, (a) at $y = 1.25$ and 3.75 m; and (b) $y = 2.5$ m.

The next calculation is performed on a room ventilated by displacement, see Fig. 5. Displacement ventilation has been increasingly used to improve indoor air quality, especially in Scandinavia. The system is largely dependent on heat sources in the room providing the upward motion of the air, and uses the supply buoyancy flux to spread the "fresh" air at floor level. The test room used here has been used in our previous experimental study, see Li et al. (1993a). It was shown that the vertical temperature profiles are a result of the combined effects of conduction, convection and radiation. A numerical study using a SIMPLE method on a single grid has also been performed (Li et al, 1993b). As at this stage in our multi-grid code, the surface radiation calculation has not been developed, no thermal radiation will be included in this multi-grid study. The aim of the present calculation is two-fold: first to evaluate the effect of heat conduction through the walls on the indoor thermal flow, and second to study the efficiency of the multi-grid solver together with the local grid-refinement technique in solving non-isothermal flow problems.

Table 1 The grid system used for the room ventilated by displacement

Grid level number	Grid number	No. of Grid points	Grid description
1	1	7 x 7 x 7	global grid
2	2	12 x 12 x 12	global grid
3	3	22 x 22 x 22	global grid
4	4	10 x 22 x 16	local grid at inlet region
4	5	20 x 14 x 42	local grid at heat source region
5	6	28 x 14 x 60	local grid at heat source region

The grid system used for calculation is summarised in Table 1 and Fig. 6. The total number of grid points is 51519, which includes the boundary grid points at all level grids. If a non-uniform grid is used with a 3-level multi-grid algorithm to achieve the same resolution near the inlet and heat source, the number of grid points will increase more than 150%. This number will increase even further if a uniform grid is used. With this grid, each simulation case takes about 2 hours on a Silicon Graphics Indigo computer. A simulation of this case with 25520 grid points, running on an IBM RISC 6000 computer with the similar speed as Indigo using a single grid solver, took about 24 hours (Li et al, 1993b). As can be seen from Fig. 6, the grid resolution near the walls is not enough. This will be overcome in the future by a non-isotropic grid refinement. The grid will be refined only in one direction instead of three directions. The non-isotropic grid refinement will also allow the resolution of the thin walls compared to the size of the room.

The U-value of the wall is 0.36 W/m^2K. The supply air flow rate is 3 room volumes per hour and the supply air temperature is 291.15K. The temperature of the exterior wall surface is kept constant 296.15K. The predicted velocity fields are shown in Fig. 7 and Fig. 8. The flow from the supply inlet, which is governed by the buoyancy forces, spreads out on the floor. The plume flows are generated over the heat source. The locally refined grids are able to produce the details of the flow near the inlet in the gravity current and in the plume. The temperature fields are shown in Fig. 9. The temperature of indoor wall surfaces is not uniform. At the lower part of the vertical walls, the surface temperature is higher than the air layer close to it. This introduces upward flow along the room surface as shown in Fig. 7. If no heat conduction is considered and the walls are perfectly insulated, the exhaust air temperature will be 301.0K whereas the predicted result is 296.0K. As can be seen, heat loss from the walls also reduces the temperature gradient and overall air temperature in the room. Thus the conduction will influence the convective flow in the room and the thermal comfort of the occupants.

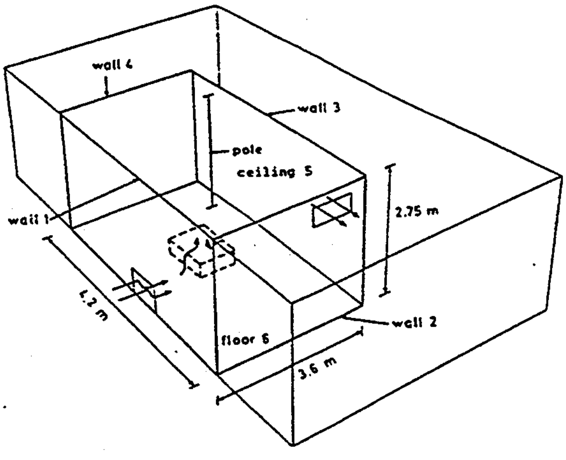

Fig. 5 Configuration of the room ventilated by displacement.

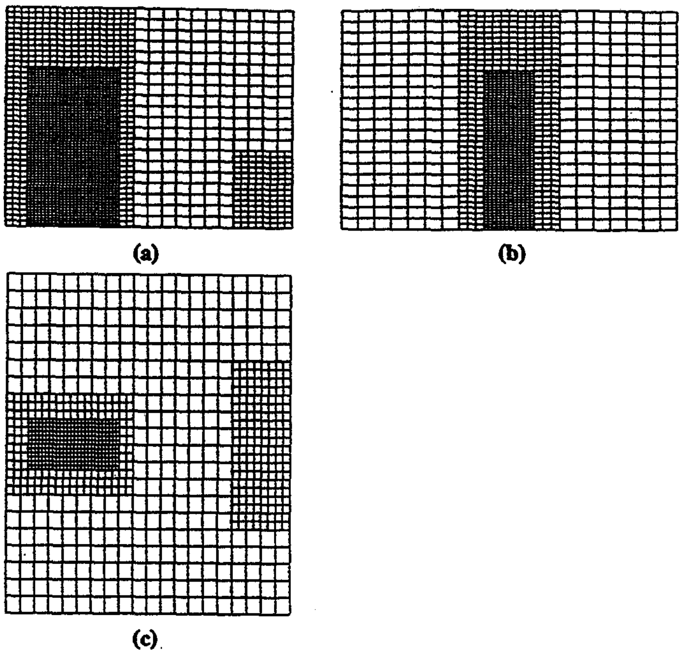

Fig. 6 The multi-grid system with locally-refined grids in the inlet region and heat source region for the test room; (a) plane y = 2.0 m, (b) plane x = 2.9 m, (c) plane z = 0.2 m.

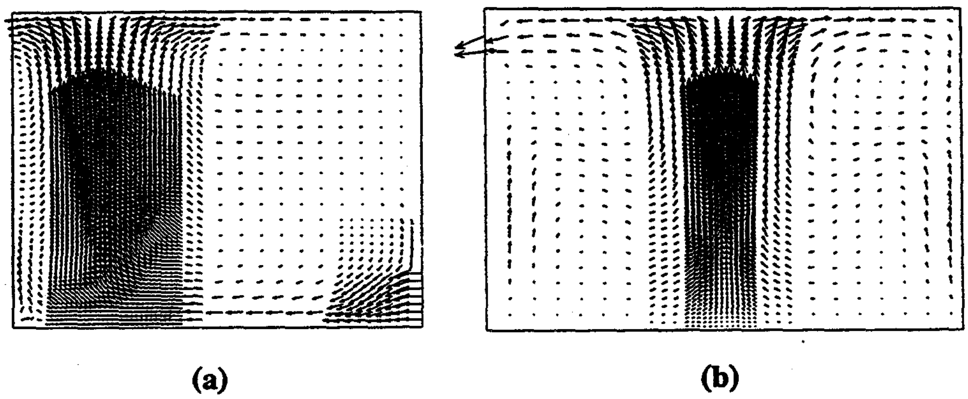

Fig. 7 Numerically predicted flow patterns; (a) at plane y = 2.0 m, (b) at plane x = 2.9 m.

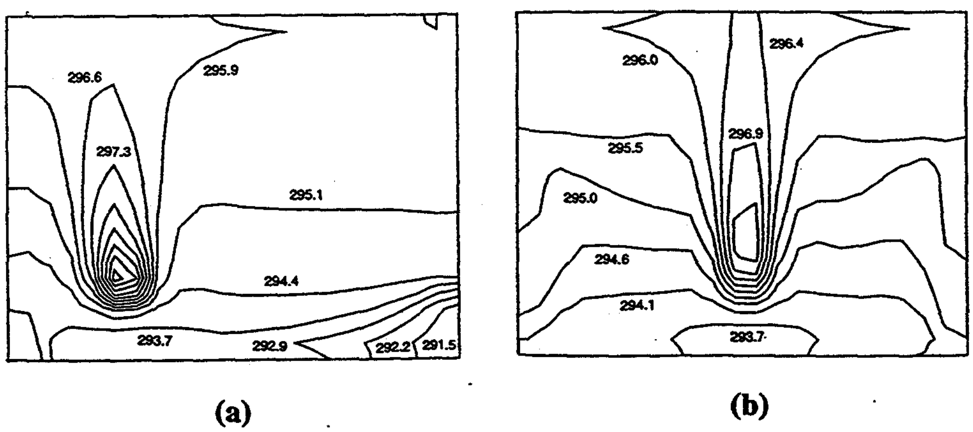

Fig. 9 Numerically predicted temperature fields; (a) at plane y = 2.0 m, (b) at plane x = 2.9 m

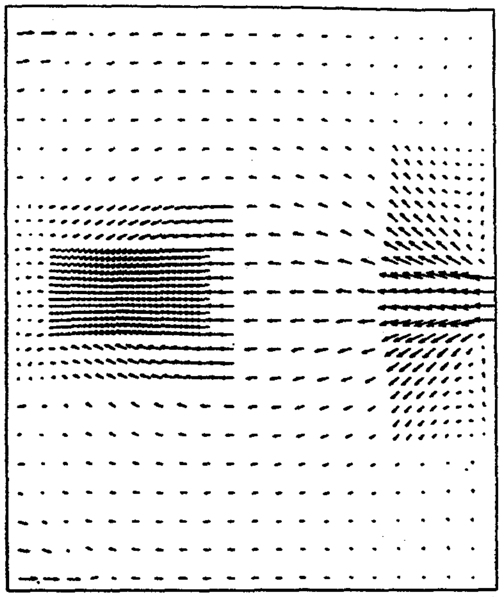

Fig. 8 Numerically predicted flow pattern at plane z = 0.2 m.

CONCLUSIONS

A multi-grid approach for predicting conjugate heat transfer and air flow in ventilated rooms has been developed. In this approach, the wall and indoor air region are treated simultaneously. The local grid-refinement technique has been used to resolve the inlet region and heat source region. The method is capable of achieving high resolution with significantly fewer grid points than a non-uniform grid method. The "equivalent" viscosity and "equivalent" diffusivity concepts are used to implement the wall functions. The multi-grid method, which incorporates the locally-refined grids, provides an accurate and efficient solver for non-isothermal flow in rooms. The effect of wall conduction cannot be neglected when studying the non-isothermal flow in a buoyancy-controlled ventilated room. The new approach will allow the specification of thermal boundary conditions at the exterior surface and thus allow the study of effects of outdoor environments on indoor climate.

REFERENCES

Agouzoul, M., Reggio, M., Camarero, R. 1989. "Simulation of three-dimensional internal turbulent flows." ASME Fluids Engineering Division FED, Vol. 69, *Flows in Non-Rotating Turbo-machinery Components.* The Winter Annual Meeting of the ASME Chicago, IL, Nov. 27 - Dec 2, 19-25.

Arnone, A., Sestini, A. 1991. "Multigrid heat transfer calculations using different iterative schemes." *Numerical Heat Transfer*, Pt. B, 19,1-11.

Bai, X.S., Fuchs, L. 1992. "Fast multigrid method for 3-D turbulent incompressible flows." *International Journal of Numerical Heat and Fluid Flow*, Vol. 2, 127-137.

Brandt, A. 1977. "Multi-level adaptive solutions to boundary-value problems." *Mathematics of Computation*, Vol. 31, 333-390.

Fuchs, L. 1986. "A local mesh-refinement technique for incompressible flows." *Computers & Fluids*, Vol. 14, No. 1, 69-81.

Lemaire, A.D. 1991. "Room air and contaminant flow - a summary report of IEA Annex 20, Sub task 1." AN20. 1-AFP-91-1.

Li, Y., Fuchs, L., Bai, X.S. 1991. "Accurate numerical simulation of air flows in ventilated multi-rooms." *Building Simulation'91*, Edited by J.A. Clarke et al., 85-91.

Li, Y. and Holmberg, S. 1993. "General flow- and thermal-boundary conditions in indoor air flow simulation." In: *Proceedings of 6th International Conference on Indoor Air Quality and Climate*, Indoor Air'93, July 4/8, Helsinki, Finland.

Li, Y., Sandberg, M., Fuchs, L. 1993a. "Effects of thermal radiation on air flow with displacement ventilation: an experimental investigation." To appear in *Energy and Buildings*.

Li, Y., Fuchs, L., Sandberg, M. 1993b. "A numerical method for predicting airflow-radiation interactions in a room with displacement ventilation." To appear in *Energy and Buildings*.

Patankar, S.K.: *Numerical heat transfer and fluid flow*, Hemisphere Publishing Corp., Washington, D.C., 1980.

Peric, M., Ruger, M. and Scheuerer, G. 1989. "A finite-volume multigrid method for calculating turbulent flows." *7th Symp. Turbulent Shear Flows*, Stanford University, Stanford, Calif., 7.3.1-7.3.6.

Rubini, P.A., Becker, H.A., Grandmaison, E.W., Pollard, A., Sobiesiak, A., Thurgood, C. 1992. "Multigrid acceleration of three-dimensional, turbulent, variable-density flows." *Numerical Heat Transfer*, Part B, vol. 22, pp. 163-177.

Srinivasan, K. and Rubin, S.G. 1992. "Segmented multigrid domain decomposition procedure for incompressible viscous flows." *International Journal for Numerical Methods in Fluids*, Vol. 15, 1333-1355.