#6175

COMPARATIVE STUDIES OF SELECTED DISCRETIZATION METHODS FOR THE NUMERICAL SOLUTION OF ROOM AIR FLOW PROBLEMS †

T. Skalický, G. Morgenstern, A. Auge, B. Hanel, M. Rösler
Dresden University of Technology
Dresden, Germany

SUMMARY

Three discretization methods and their implementation in computer codes are described:

- Finite Volume Methods with the variables velocity and pressure or vorticity and streamfunction, respectively, and
- a Finite Element Method employing the Galerkin/least–squares approach. This method embodies a straightforward extension of the Streamline–upwind Petrov–Galerkin (SUPG) method resulting in a very stable discretized scheme.

Statements concerning capability to predict air flow patterns within rooms are presented.

The same terms of the Reynolds–averaged Navier-Stokes equations and the standard k–ε turbulence model are used for all methods. Two simple two–dimensional test cases (turbulent room airflow with one inlet and one outlet device at opposite walls and laminar convection within a square room with different wall temperatures) are calculated to compare accuracy, convergency and dependence of grid spacing of the algorithms.

†The research was supported by the Bundesministerium für Forschung und Technologie under the contract 0329016D.

COMPARATIVE STUDIES OF SELECTED DISCRETIZATION METHODS FOR THE NUMERICAL SOLUTION OF ROOM AIR FLOW PROBLEMS

T. Skalický, G. Morgenstern, A. Auge, B. Hanel, M. Rösler
Dresden University of Technology
Dresden, Germany

INTRODUCTION

Three computer codes for prediction of room air flows have been developed and maintained at the Institute of Fluid Mechanics at the Dresden University of Technology. In two of them Finite Volume Methods are implemented (one with primitive variables and another one with the variables stream function and vorticity), the third is based on a Galerkin/least-squares Finite Element Method. Different solvers for the algebraic equation systems are implemented. Stimulated by ANNEX 20 activities, the codes have been tested for their capability to predict room air flows with natural and forced convection, that is the Nielsen test case 2D1 [9] and a closed cavity problem. In addition, accuracy, expence and dependence of grid spacing of the underlying methods are examined.

BASIC EQUATIONS

The basic equations are the Reynolds averaged Navier–Stokes equations and further transport equations, for instance the equation of energy transport. By using of the assumption of incompressibility, Boussinesq's approximation of buoyancy and a $k-\varepsilon$ turbulence model, the following system of differential equations can be presented:

$$\frac{\partial \overline{U}_i}{\partial t} + \frac{\partial \overline{U}_j \overline{U}_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\nu_{eff}\left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i}\right)\right) - \frac{\partial}{\partial x_i}\left(\frac{\overline{p}}{\rho}\right) - g_i\gamma(\overline{T} - T_0), \quad (1)$$

$$\frac{\partial \overline{U}_i}{\partial x_i} = 0, \quad (2)$$

$$\frac{\partial \overline{T}}{\partial t} + \frac{\partial \overline{U}_j \overline{T}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(a_{eff}\frac{\partial \overline{T}}{\partial x_j}\right) + \overline{\dot{q}}^V, \quad (3)$$

$$\frac{\partial k}{\partial t} + \frac{\partial \overline{U}_j k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\nu + \frac{\nu_t}{Pr_k}\right)\frac{\partial k}{\partial x_j}\right) + \nu_t \frac{\partial \overline{U}_i}{\partial x_j}\left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i}\right) - \varepsilon + g_j\gamma\frac{\nu_t}{Pr_t}\frac{\partial \overline{T}}{\partial x_j}, \quad (4)$$

$$\frac{\partial \varepsilon}{\partial t}+\frac{\partial \overline{U}_j\varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\nu+\frac{\nu_t}{Pr_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right)+C_1\nu_t\frac{\varepsilon}{k}\frac{\partial \overline{U}_i}{\partial x_j}\left(\frac{\partial \overline{U}_i}{\partial x_j}+\frac{\partial \overline{U}_j}{\partial x_i}\right) -C_2\frac{\varepsilon^2}{k}+C_1 g_j\gamma\frac{\nu_t}{Pr_t}\frac{\partial \overline{T}}{\partial x_j} \tag{5}$$

with

$$\nu_{eff} = \nu+\nu_t, \tag{6}$$

$$a_{eff} = \frac{\nu}{Pr}+\frac{\nu_t}{Pr_t}, \tag{7}$$

$$\nu_t = C_D k^2/\varepsilon \tag{8}$$

and appropriate initial and boundary conditions and moreover a set of constants, see table 1.

C_D	C_1	C_2	Pr_ε	Pr_k	Pr_t
0,09	1,44	1,92	1,3	1,0	0,77

Table 1: Set of constants for k–ε turbulence model

Equations (1), (2) and (3) with $\nu_{eff}=\nu$ are used for laminar flow problems.

NUMERICAL METHODS

Finite Volume Method with primitive variables

Discretization

This method is a time marching procedure. It is based on the algorithm of the MAC–method of Harlow and Welch [2]. The basic idea of the solution procedure can be presented as follows:

- Integration of equation (1) and (3) over a time step

$$\overline{U}_i^{m+1} = \overline{U}_i^m+\int_{t^m}^{t^{m+1}}\frac{\partial}{\partial x_j}\left(\nu_{eff}\left(\frac{\partial \overline{U}_i}{\partial x_j}+\frac{\partial \overline{U}_j}{\partial x_i}\right)\right)-\frac{\partial \overline{U}_j\overline{U}_i}{\partial x_j}-\frac{\partial \overline{P}}{\partial x_i} -g_i\gamma(\overline{T}-T_0)\,dt \tag{9, 10}$$

$$\overline{T}^{m+1} = \overline{T}^m+\int_{t^m}^{t^{m+1}}\frac{\partial}{\partial x_j}\left(a_{eff}\frac{\partial \overline{T}}{\partial x_j}\right)-\frac{\partial \overline{U}_j\overline{T}}{\partial x_j}+\overline{q}^V\,dt \tag{11}$$

- Approximation of the integrals with a simple explicit Euler scheme

$$\overline{U}_i^{m+1} = \overline{U}_i^m+\Delta t\left[\frac{\partial}{\partial x_j}\left(\nu_{eff}\left(\frac{\partial \overline{U}_i}{\partial x_j}+\frac{\partial \overline{U}_j}{\partial x_i}\right)\right)-\frac{\partial \overline{U}_j\overline{U}_i}{\partial x_j}-\frac{\partial \overline{P}}{\partial x_i} -g_i\gamma(\overline{T}-T_0)\right]^m \tag{12}$$

$$\overline{T}^{m+1} = \overline{T}^m+\Delta t\left[\frac{\partial}{\partial x_j}\left(a_{eff}\frac{\partial \overline{T}}{\partial x_j}\right)-\frac{\partial \overline{U}_j\overline{T}}{\partial x_j}+\overline{q}^V\right]^m \tag{13}$$

Treatment of equations (4) and (5) is analogously. (Another, more exact approximation one can find in [12].) Calculation of pressure $\overline{P}^m$ using equation of mass conservation (2) is based on:

$$\frac{\partial^2 \overline{P}^m}{\partial x_i^2} = \frac{1}{\Delta t}\frac{\partial \overline{U}_i^m}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\frac{\partial}{\partial x_j}\left(\nu_{eff}\left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i}\right)\right) - \frac{\partial \overline{U}_j \overline{U}_i}{\partial x_j} - g_i\gamma(\overline{T} - T_0)\right]^m \tag{14}$$

After the calculation of pressure, new velocity, temperature and further transport quantities are determined.

A Finite Volume Method on a staggered grid is used for the spatial discretization. By integration over each control volume, application of Gauß' theorem and local approximations one get a system of difference equations.

According to the topical problem, boundary conditions have to be discretized in the same manner. Additional grid points were introduced at the physical boundaries for Dirichlet conditions. Boundary conditions for the pressure are necessary to solve the Poisson equation. One can deduce boundary conditions from equation (12), see [11].

Solution of the algebraic equation system

Following the above mentioned algorithm, new velocity, temperature and further transport quantities are calculated by means of explicit equations. The Poisson equation for pressure is solved by a multigrid method at each time step. It consists of a so called V–cycle on 3 or 4 grid levels. Smoothing iteration is a Gauß–Seidel method, the equation system on the coarsest grid is solved by the Cholesky factorization.

Code

Currently the PASCAL code "ResCUE" runs under UNIX operating systems. It consits of about 6 000 lines source code. The amount of occupied memory is about 280 bytes per grid point.

Finite Volume Method with variables ω, Ψ

Differential equations, boundary conditions

For a steady flow described with the variables stream function Ψ and vorticity ω, defined by

$$U_1 = \frac{\partial \Psi}{\partial x_2}, U_2 = -\frac{\partial \Psi}{\partial x_1}, \omega = \frac{\partial U_2}{\partial x_1} - \frac{\partial U_1}{\partial x_2} \tag{15}$$

the differential equations 1 and 2 lead to

$$\frac{\partial}{\partial x_1}\left(\omega\frac{\partial \Psi}{\partial x_2}\right) - \frac{\partial}{\partial x_2}\left(\omega\frac{\partial \Psi}{\partial x_1}\right) = \frac{\partial}{\partial x_1}\left(\frac{\partial \nu_{eff}\omega}{\partial x_1}\right) + \frac{\partial}{\partial x_2}\left(\frac{\partial \nu_{eff}\omega}{\partial x_2}\right) + g\gamma\frac{\partial T}{\partial x_1}, \tag{16}$$

$$\frac{\partial^2\Psi}{\partial x_1^2} + \frac{\partial^2\Psi}{\partial x_2^2} = -\omega. \tag{17}$$

Problems of laminar flow can be solved also by equations (16), (17) and (3), if $\nu_{eff} = \nu$.

The relatively difficult boundary conditions for ω and Ψ are defined using relation (15), the non-slip-conditions on the walls and assumptions about velocity profiles in the vicinity of walls, given velocity profile on inlet, and the assumptions of a fully developed flow in the outlet, see [10].

Discretization

An orthogonal grid is used for the discretization. Each differential equation is integrated over rectangular control volumes, values of functions and their derivatives on the boundary of control volumes are interpolated and expressed using values of adjacent nodes.

For stabilizing the discrete equations, simple upwind differences are used for the convective terms.

Solution of the algebraic equation system

The system of discrete equations is solved successively. The connection of variables is given through an outer iteration loop.

The system of linear equations for each variable is solved by a modified block-Gauß-Seidel method (line by line). This method is stable on non-uniform grids if the aspect ratio of control volumes is $0.1 < \frac{\Delta y}{\Delta x} < 10$, cf. [13].

Code

The computer code "PSIOM2D" which consists of about 4 000 lines of FORTRAN 77 runs on workstations under UNIX operating systems. Approximately 172 bytes memory space are required per grid node.

Finite Element Method with primitive variables

Galerkin/least–squares formulation of the governing equations

The Galerkin/least–squares (GLS) formulation of a convection/diffusion (or transport) equation is a generalization of the Streamline diffusion or Streamline–Upwind–Petrov/Galerkin method (SDFEM or SUPG) which results in a very stable numerical scheme and allows arbitrary choosing of interpolation functions. This formulation is an extension of the Standard Galerkin method where the residuals are weighted additionally by the differential operator itself. Equ. (18), (19) are consistent in the sense that the exact solution still satisfies the stabilized problem. The above mentioned technique is applied as well to momentum and continuity equation as to energy transport equation.

$$\sum_{\mu=1}^{M} \int_{\Omega_\mu} (-\nu\Delta u + (u \circ \nabla)u + \nabla p) \circ \tilde{w}\, d\Omega = \sum_{\mu=1}^{M} \int_{\Omega_\mu} f \circ \tilde{w}\, d\Omega \tag{18}$$

$$\sum_{\mu=1}^{M} \int_{\Omega_\mu} (\nabla \circ u)\, \tilde{q}\, d\Omega = 0 \tag{19}$$

$$\begin{aligned} \tilde{w} &= w + \delta_1(-\nu\Delta w + (u \circ \nabla)w + \nabla q) \\ \hat{q} &= q + \delta_2(\nabla \circ w) \end{aligned}$$

It should be mentioned that only 2D laminar and steady flow problems are considered at this stage of research.

Discretization of boundary conditions

The boundary conditions are formulated in an integral form, i.e. nodewise given Dirichlet–boundary values are piecewise integrated over the boundary edges of the finite elements. As well prescribed Dirichlet conditions as homogeneous and non–homogeneous Neumann conditions can be employed.

Spatial discretization and choise of elementary shape functions

An arbitrary 2D domain can be discretized using an unstructured triangle based mesh. In case of a rectangular domain as in the examples presented here, only structured orthogonal triangulations are applied. For simplicity and computational convenience, we restricted the implementation to piecewise linear finite elements for velocity, pressure, and temperature approximation, respectively.

Solution procedure

The linear algebraic equation system resulting from the GLS–discretization of the successive linearized (with respect to all non–linearities) p.d.e. system (1)-(3) has a significiant block structure. Each block consists of banded sparse matrix of dimension n (n=amount of nodes). A particular block iteration procedure in which all unknowns of a single node are treated simultaneously is applied to this equation system. This very simple procedure has been successfully tested up to systems with some 7 000 nodes, i.e. some 30 000 equations. The successive approximation process was stopped after 200 cycles or when the relative difference of two following cycles became less than 10^{-7}.

Computer code

The program system is a research code consisting of several parts. These are two mesh generators, some interface programs and the main part, the program "NS". Programming languages are above all C and FORTRAN 77. The whole memory required is allocated dynamically at run time. The code consists of about 5 000 lines. Memory requirements are about 312 bytes per node, whereas the code is designed for flexibility and is not optimized for storage.

COMPUTATIONS

Laminar natural convection in a square cavity

Flow in a cavity is often used to validate numerical methods. Here the computational domain consists of a closed two–dimensional square (cf. fig. 1) with a hot (T_H) and a cold (T_C) vertical wall. The horizontal walls are isolated.

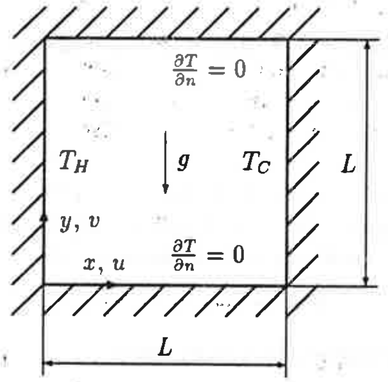

Figure 1: Geometry of cavity

Ra	10^5
Pr	0.71
g	10.0 m/s^2
γ	3.6 10^{-3} 1/K
ν	15.00 10^{-6} m^2/s
a	21.13 10^{-6} m^2/s
L	0.05605 m
ΔT	5.0 K
T_H	280.3 K
T_C	275.3 K

Table 2: Parameter of computations

Two dimensionless parameters are essential for flow and temperatur field — the Prandtl number $Pr = \nu/a$ and the Rayleigh number $Ra = (g\gamma L^3/\nu^2)\,\Delta T\,Pr$. Restriction to the laminar case ($Ra < 10^8$) allows investigations of some properties of discretization methods, e.g. stability and convergency, almost independent of turbulence modelling. Resulting from the same structure of the differential equations, the achieved piece of knowledge can be useful also for solving turbulent flow problems with natural convection.

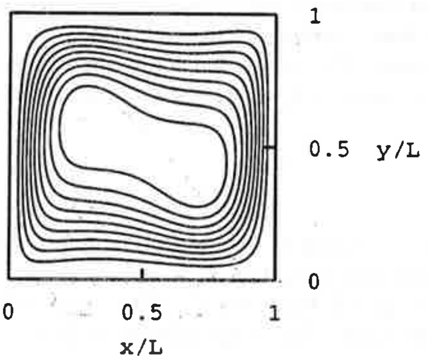

Figure 2: Streamlines

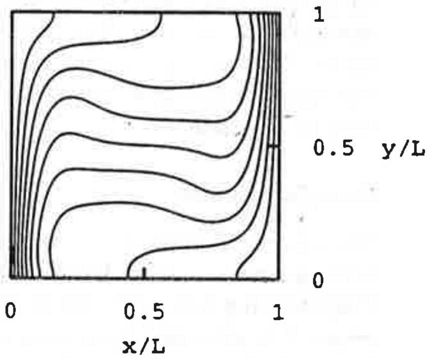

Figure 3: Isotherms

A lot of numerical results of laminar flow exists, cf. [4], [7], [15], [3]. Hortmann et al. [4] obtained very accurate results (estimated accuracy 0.01%) as solution on very fine grids (320x320 and 640x640 control volumes, respectively) for Rayleigh numbers 10^4, 10^5 and 10^6 and Prandtl number 0.71. They are used as a basis for comparison for all of the discretisation methods described in the previous section.

Computations of this problem have been carried out with parameters shown in tab. 2. The computational domain was discretized using a uniform grid (see tab. 3). Criterion of accuracy was specified by 10^{-5}.

The results – flow and temperatur pattern (cf. fig. 2, 3) – are compared by velocity and temperatur profiles along the horizontal and vertical midlines, cf. fig. 4. The coordinates x and y are normalized by the cavity lenght L, the velocity by the characteristic conduction velocity a/L. One can see that predicted velocity and temperature distribution are qualitatively fairly similar for all tree discretization methods.

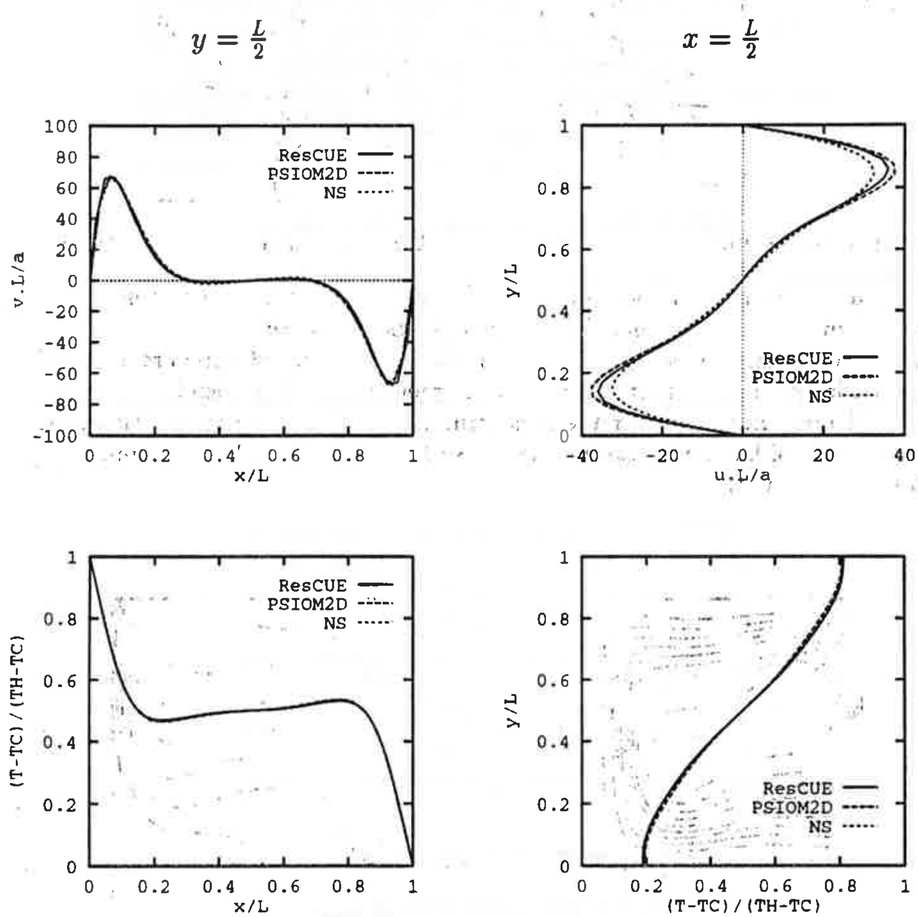

Figure 4: Profiles of velocity and temperature along the horizontal and vertical midlines

The value and position of maximal velocities on the midlines are summarized in tab. 3. This table contains also the comparison results of Hortmann et al. [4], "grid-independent' solution was yielded by extrapolation of the solutions on two grids with 320x320 and 160x160 control volumes. With this solution one can investigate

code	grid	$u_{max}L/a$	y_{max}/L	$v_{max}L/a$	x_{max}/L
ResCUE	16x16	36.4799	0.84374	66.0278	0.03124
	32x32	35.9944	0.85937	66.9722	0.07813
PSIOM2D	16x16	41.2881	0.87500	66.3700	0.06250
	32x32	37.5260	0.84374	67.7142	0.06250
	64x64	36.1235	0.85937	68.3207	0.06250
	128x128	35.5027	0.85937	68.6258	0.06250
NS	16x16	28.6578	0.81250	50.9485	0.06250
	32x32	32.4431	0.84375	66.4092	0.06250
	48x48	33.7588	0.85417	68.1954	0.06250
	64x64	34.1771	0.85938	68.5306	0.06250
cf. [4]	320x320	34.7414	0.85468	68.6187	0.06719
	indep.	34.7399	—	68.6396	—

Table 3: Maximal velocities on the midlines

the convergence. The estimated error for e.g. u_{max}, defined by

$$E(u_{max}) = \left| \frac{u_{max} - u^*_{max}}{u^*_{max}} \right|$$

where u^*_{max} is the 'grid-independent' value, is shown in figure 5 as a function of mesh parameter h, using logarithmic scale. The solution converges for all discretisation

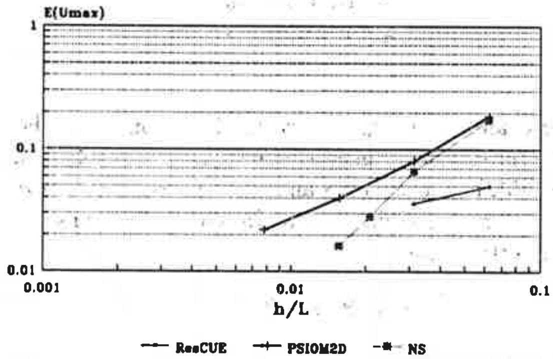

Figure 5: Estimated error of u_{max}

methods. The slope of the curves can be used to estimate the convergence order. The order of 1.15 obtained by PSIOM2D is slightly better then the theoretically estimated for first order approximation of convective terms. For NS the order is about 1.55 and agrees with a analysis [6] very well.

Room air flow

A simple two–dimensional test case specified by Nielsen [9] was choosen to validate the programs PSIOM2D and ResCUE for their ability to predict the isothermal two–dimensional turbulent flow in a ventilated room. The geometry of the testroom shown in figure 6 is $L/H = 3, h/H = 0.056$ and $t/H = 0.16$. The height of the room H is $3.0m$ in the present work. In addition, the expected flow pattern is shown in figure 6.

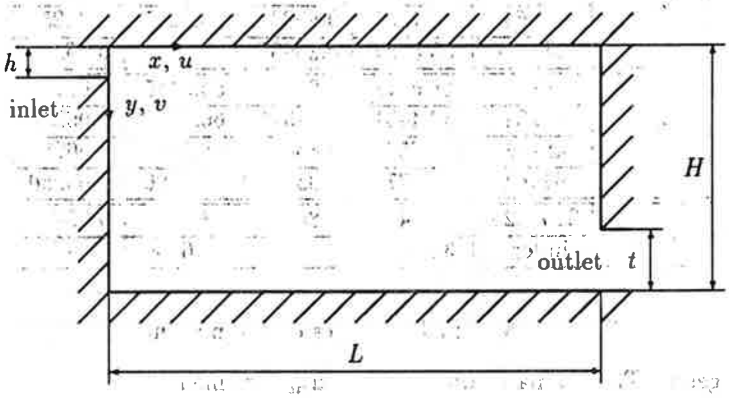

Figure 6: Geometry and expected flow pattern of the test room

In the inlet, following boundary conditions are given:

$$\begin{aligned} u_0 &= 0.455m/s \\ k_0 &= 1.5(0.04 \cdot U_0)^2 \\ \varepsilon_0 &= 0.09 \cdot k_0^{1.5}/(h/10). \end{aligned}$$

In each case a coarse and a fine grid were used to point out the effect of mesh refinement. The simulations have been carried out with a 64x32 and 128x64 grid for the program PSIOM2D and a 64x28 and 128x56 grid for the program ResCUE. In accordance with experimental data from Nielsen, predictions of velocity profiles are given at the two vertical lines

$$x = H,\; x = 2H$$

and at the two horizontal lines

$$y = h/2,\; y = H - h/2.$$

Figure 7 shows the profiles of the velocity in x–direction at $x = H$. The global prediction of all simulations is fairly good. Only the program ResCUE using the fine grid does not satisfactory predict the recirculation at the bottom of the room. That could be caused by a far small time step which was used to ensure stability of

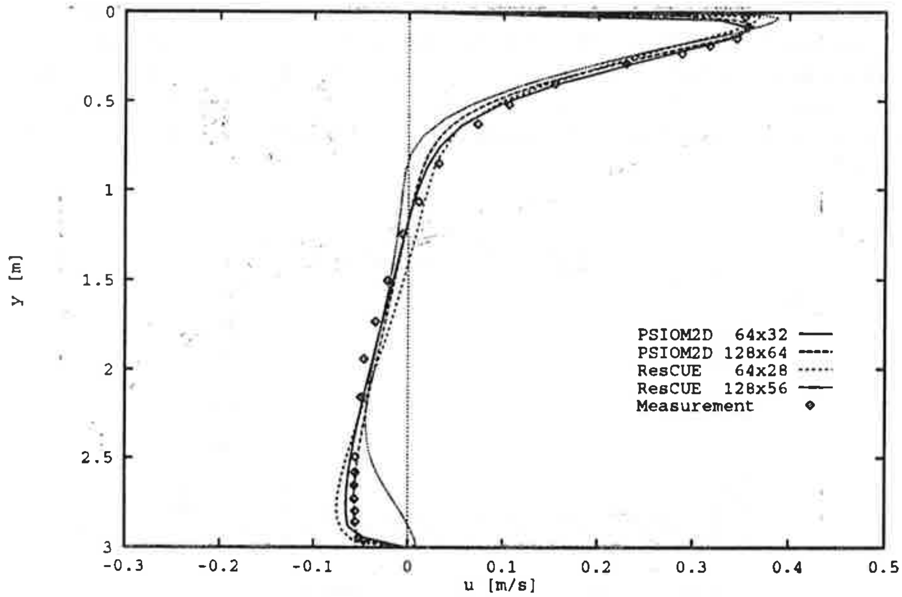

Figure 7: Velocity profiles at $x = H$

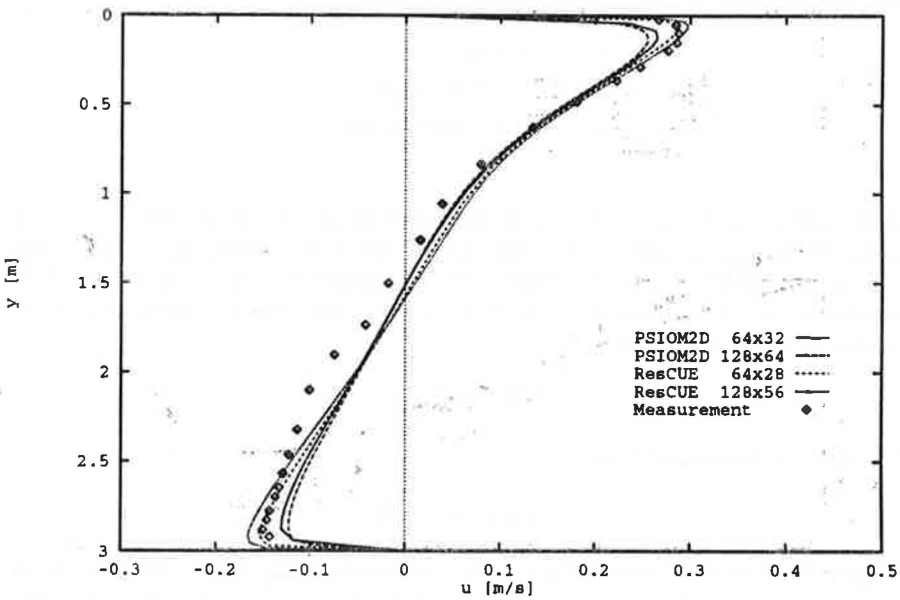

Figure 8: Velocity profiles at $x = 2H$

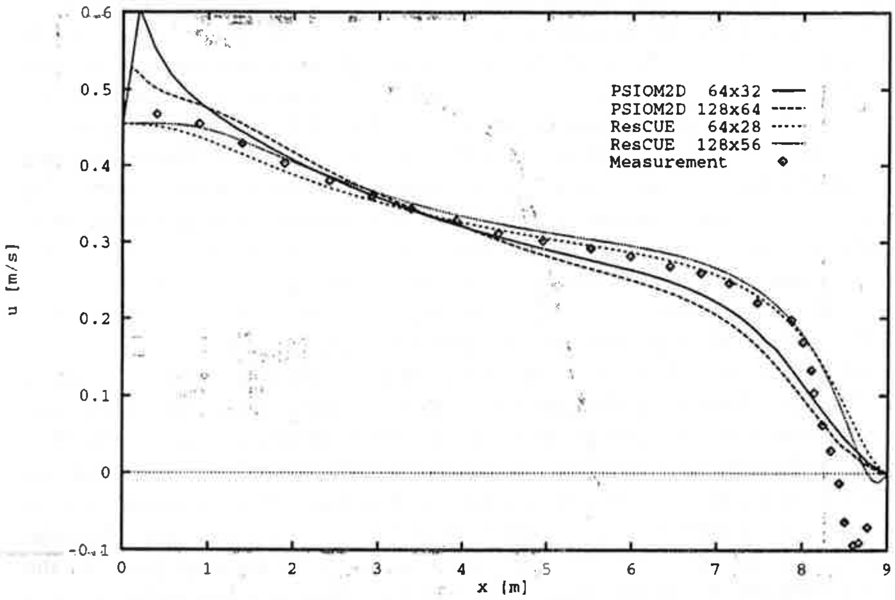

Figure 9: Velocity profiles at $y = h/2$

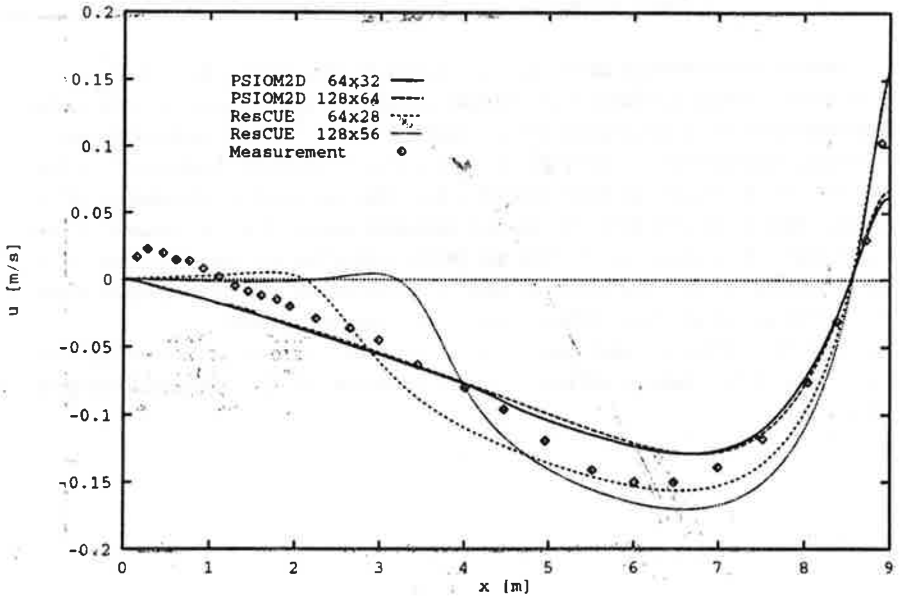

Figure 10: Velocity profiles at $y = H - h/2$

the explicit method. The criterion of convergence, the relative difference between two time steps ($< 10^{-5}$), has been achieved after about 12 000 time steps, but much more time steps should be proceeded. This reason will be amplified by the results shown in figure 8. The decay of the wall jet and the recirculating flow are well predicted by ResCUE on a coarse grid. The values of velocity in the wall jet and the recirculating flow are too high calculated with ResCUE using the fine grid.

Figure 9 shows the behaviour of the wall jet in the x-direction. There must be a maximum of u greater than u_0 that can be recognized from the measurement. The prediction of this velocity maximum by PSIOM2D using the coarse grid is too high. This value becomes more realistic by refining the grid. In contrast to most other computer codes [14], this velocity maximum behind the jet entrance is predicted only by PSIOM2D because of a special discretization for the vorticity at the corner. ResCUE predicts values greater than u_0 near the inlet, but they are too low and show only a tendency. Near the outlet, a region of recirculating flow exists which has not been predicted by PSIOM2D but by ResCUE with values that are too low. Figure 10 shows how the computer codes predict the existence of three specific flow regions near the bottom of the room. The vortex in the lower left corner is predicted by ResCUE with a too low intensity a a too big dimension. The maximum velocity of the recirculating flow is well predicted by ResCUE using the coarse grid. Solution by ResCUE on the finer grid suffers from the very small time step, too. At the outlet wall velocity is positive again. The fact that there is a low outlet velocity of PSIOM2D is caused by using Neumann boundary conditions for velocity in x-direction. In the program ResCUE Dirichlet boundary conditions are implemented for u.

CONCLUSION AND OUTLOOK

Three discretization methods have been described in this work. They have been applied to predict room air flows with natural and forced convection. In both cases the agreement of the computational results with experimental data and solutions of other authors, respectively, is fairly good. It must be considered, however, that for room air flow with regions of more complex flow like the wall jet development or recirculating flow in corners all codes predict different results. For this reason, more simple problems like a closed cavity flow are better suited for comparative studies of different computer codes. Numerical or analytical solutions of other authors allow a better estimation of accuracy without employing empirical data.

Next time the structure and properties of equation system, produced by discretization, should be examinated in detail. Implementation of a uniform Multigrid solver is planed.

REFERENCES

[1] Brooks, A.N.; Hughes, T.J.R.: Streamline upwind/Petrov–Galerkin formulations for convection dominated flows with particular emphasis on the incompressible Navier–Stokes equations. Computer Methods in Applied Mechanics and Engineering, 32 (1982), pp. 199–259.

[2] Harlow F. H.; Welch J. E.: Numerical calculation of time dependent viscous incompressible flow of fluid with free surface. Physics of Fluids, 8 (1965), pp. 2182-2189.

[3] Hatanaka K.; Kawahara M.: A fractional step finite element method for conductiv-convective heat transfer problems. International Journal of Numerical Methods for Heat and Fluid Flow, 1 (1991), pp. 77-94.

[4] Hortmann M.; Perič M.; Scheuerer G.: Finite volume multigrid prediction of laminar natural convection bench-mark solution. International Journal for Numerical Methods in Fluids, 11 (1990), pp. 189-207.

[5] Hughes, T.J.R.; Franca L.P.; Hulbert G.M.: A new finite element formulation for computational fluid dynamics: VIII. The Galerkin/least-squares method for advective-diffusive equations. Computer Methods in Applied Mechanics and Engineering, 73 (1989), pp. 173–189.

[6] Lube, G.; Auge, A.: Galerkin/least-squares approximations of incompressible flow problems. to appear in Notes on Numerical Fluid Mechanics, Vieweg Verlag.

[7] Mahdi H. S.; Kinney R. B.: Time-dependent natural convection in a square cavity: Application of a New Finite Volume Method. International Journal for Numerical Methods in Fluids, 11 (1990), pp. 57-86.

[8] Merker, G.P.; Küblbeck, K.; Straub, J.: Advanced numerical computation of two-dimensional ,time-dependent free convection in cavities. International Journal for Heat and Mass Transfer, 23 (1980), pp. 203–217.

[9] Nielsen, P.V.: Specification of a two-dimensional test case. ANNEX 20 - Internal report, Aalborg, 1990.

[10] Scholz R.; Hanel B.: Computergestützte Berechnung der Raumluftströmung. Verlag Technik, Berlin, 1988.

[11] Rösler M.: Mathematisch–numerische Untersuchungen zur Berechnung von dreidimensionalen, inkompressiblen Kanal– und Raumluftströmungen. Ph.D. Thesis, Dresden University of Technology, 1992.

[12] Rösler M.; Hanel B.: Numerical computations of flow and heat transfer in air–conditioned rooms by a special velocity–pressure iteration and a multigrid method. submitted to ROOMVENT 92 Conference, Aalborg, 1992

[13] Vogel P.: Untersuchung der Strömung und des Wärmeübergangs in turbulenten Wandstrahlen. Großer Beleg, Dresden University of Technology, 1991.

[14] Whittle G. E.: Evaluation of measured and computed test case, results from ANNEX 20, subtask 1, 12th AIVC Conference, Air Movement & Ventilation Control within Buildings, Ottawa, 1991.

[15] Zeng S.; Wesseling P.: A multigrid method combinated with defect correction for free convection problems at high Rayleigh numbers. Report of Delft University of Technology No. 89-27, Delft 1989.