6016

International Symposium on Room Air Convection and Ventilation Effectiveness
University of Tokyo, July 22 – 24, 1992

Numerical Simulation of Room Thermal Convection, - Review of IEA Annex-20 Results

ALFRED MOSER

Swiss Federal Institute of Technology, ETH,
Energy Systems Laboratory, Energietechnik ML,
ETH-Zentrum, CH-8092 Zurich, Switzerland

ABSTRACT

Computational fluid dynamics provide a powerful tool to model air motion in and around buildings. However, acceptance of numerical methods, routinely used in aerospace, propulsion, and meteorology, is slow in the building industry. Therefore, the International Energy Agency, IEA, has launched a project under its Energy Conservation in Buildings Program to evaluate numerical methods applied to prediction of building air flow. This Project, Annex 20, *Air Flow Patterns within Buildings*, is now completed.

The paper summarizes the goals and achievements of this project. It focuses on conclusions and recommendations resulting from the numerical simulation of five case studies that were supported by independent measurements by different countries in identical test chambers. The three-dimensional turbulent air flow patterns were simulated by nine different computer codes and compared with measurements. For a two-dimensional benchmark case, the throw of a cold jet was predicted as a function of Archimedes number to show how well different numerical algorithms agree with each other. The findings of the Annex-20 participants will be reported and the applicability and limitations of numerical field models in room air flow discussed. The impact Annex-20 work had on research at the ETH will be touched on, and approaches for the simulation of thermal convection surveyed.

KEYWORDS IEA Annex 20, Numerical Simulation, Computational Fluid Dynamics, Thermal Convection, Turbulence Model, Heat Transfer, Test Cases

INTRODUCTION

The performance of building envelopes has been improved during the last ten years, notably with regard to thermal insulation and air-tightness. While this progress helps to save considerable amounts of energy, more careful control of ventilation and air conditioning is required to guarantee optimal occupant comfort and indoor air quality and to prevent damages (e.g. by moisture) to the building structure. In addition, it has been recognized that provision of a healthy climate increases productivity of occupants in commercial buildings, and many so-called sick buildings had been under scrutiny by dissatisfied owners. These are reasons why accurate air flow pattern analyses pay off since they may prevent the installation of an inefficient ventilating system.

The International Energy Agency (IEA) project *Air Flow Patterns within Buildings*, Annex 20 of the IEA *Energy Conservation in Buildings and Community Systems* (ECB) Program, has examined the applicability of numerical methods to the design and analyses of building air flow (Moser 1991). One group of participants looked at the flow through entire buildings (multi-zone air flow), a second group at the flow within one room. In this single-zone air flow sub-task, numerical field models were evaluated by comparing calculations with full-scale measurements. The application of computational fluid dynamics (CFD) to indoor climate allows prediction of fields of velocity vectors, temperature, turbulent kinetic energy, and contaminant concentrations. When the flow field is known, it is easy to calculate derived quantities, such as local ventilation effectiveness in terms of *percentage dissatisfied persons* due to draft risk and odors.

One of the most delicate tasks is the modelling of forced and free thermal convection. Annex experts agreed that more work needs to be done to achieve reliable and accurate prediction of heat transfer to surfaces. It was shown that a successful method to assess ventilation performance asks for: (1) A turbulence model valid near walls, (2) an optimal grid, (3) an adequate description of the near-wall sublayer, (4) knowledge on heat transfer by radiation, (5) knowledge on the interactions of the air flow with thermal building dynamics, and (6), last but not least, an experienced specialist to apply the tools to actual buildings.

The understanding of natural and mixed convection is key to the development of computational methods for the simulation of large enclosed air spaces as found, for instance, in sports stadia, concert halls, or atria. The reason is that the air velocity in down-drafts grows with a fractional power of height, whereas the center-line velocity in a jet decreases with distance. Incidentally, the (square of the) ratio of these two characteristic velocities amounts to the Archimedes number, a parameter that expresses the importance of buoyancy relative to momentum forces. It has a high value for large air spaces with temperature differences. The Executive Committee of the IEA ECB program has recognized that the investigation of methods for the design, field evaluation, and retrofit of the ventilation of large enclosures deserves careful attention. Therefore it has initiated a new project, *Energy-efficient ventilation of large enclosures*.

The paper has two parts: Part 1 summarizes the work and results of IEA Annex 20 with focus on Subtask 1, *Room air and contaminant flow*; Part 2 concentrates on some of the unsolved problems reported by Annex-20 participants, mainly related to heat transfer, and discusses approaches under study at ETH and elsewhere.

PART 1: THE IEA ANNEX 20, AIR FLOW PATTERNS WITHIN BUILDINGS

The International Energy Agency (IEA) task-sharing project "Air Flow Patterns within Buildings" was initiated in May 1988 for a duration of 3 $^1/_2$ years. Thirteen nations contributed work and expertise and "shared the task" specified in the project's objectives. The project belongs to the Implementing Agreement on *Energy Conservation in Buildings and Community Systems*. As an "Attachment" to the Implementing Agreement, the project is called "Annex" 20.

The Annex 20 provided an opportunity to bring experts together, to compile information, and to undertake validation exercises. For single- and multi-zone air flow, experimental data sets were required as benchmark cases. Therefore, experiments had to be specified for the project. Soon the scope of the Annex was extended to include the evaluation and documentation of advanced measurement techniques for multi-zone air flow (Roulet and Vandaele 1991) and the development of new algorithms to model special flow mechanisms (Van der Maas 1992; Roulet et al. 1992).

The participants have learned how to calculate air movement in buildings and rooms. And the experienced user of air flow codes can now take advantage of the benefits of numerical simulation:

- Information is available in *all points* of the flow field (on computational grid).
- Any desired variable of the physical model can be output and plotted: Air velocity and its fluctuations (turbulence), temperature, concentrations of contaminants and humidity, "local age" (an indicator of the "freshness" of the air), and comfort parameters (thermal comfort and "risk of draft").
- Sensitivity tests and parameter variations are easy to do, and computed trends should be even more reliable than absolute values of variables.

The optimization of air flow patterns has a strong impact on energy conservation. The trend to air-tight buildings with improved thermal insulation asks for controlled air exchange as already documented by the work of IEA ECB Annex 18, *Demand Controlled Ventilating Systems*, Annex 14, *Condensation*, and Annex 9, *Minimum Ventilation Rates*.

The energy consumption by ventilation is of growing significance in relation to the heat loss through the envelope of well insulated buildings. The designer of new-generation buildings wants to know how the air flows before the house is built!

1.1 Annex 20 objectives

Formal participation in this task-sharing Annex is based on the legal text (IEA 1989) that defines project objectives, tasks, and responsibilities. The document states:

> "The objectives of this task are to evaluate the performance of single- and multi-zone air and contaminant flow simulation techniques and to establish their viability as design tools. The task is divided into two subtasks:
>
> (a) Subtask 1 - Room air and contaminant flow;
>
> (b) Subtask 2 - Multi-zone air and contaminant flow, including related measurement techniques."

Table 1 lists the participating countries along with their subtask commitments and cities where they organized meetings.

Country	Commitment: Subtask 1	2	Expert Meeting: City and date
Belgium		full	Lommel Nov. 89
Canada	full	c	Ottawa Sep. 91
Denmark	full		Aalborg May 89
Finland	full		
France	full	full	Nice Oct. 90
Germany	full		Aachen Apr. 91
Italy	full		
Netherlands	full STL	full	Lommel Nov. 89
Norway	full		Oslo June 90
Sweden	full	c	
Switzerland	full	full STL	Winterthur May 88
United Kingdom	c	full	Warwick Nov. 88
USA	c	full	
totals 13	10	6	8

Table 1 Participating countries and meeting sites. (**full** = full commitment, **c** = contribution, **STL** = subtask leader)

1.2 General Approach in Subtask 1: Room Air and Contaminant Flow

Objectives of the subtask:

- To evaluate the performance of three-dimensional complex and simplified air flow models in predicting flow patterns, energy transport, and indoor air quality,
- to show how to improve air flow models,
- to evaluate applicability as design tools,
- to produce guidelines for selection and use of models,
- to acquire experimental data for evaluation of models.

The basic approach was to solve *identical problems* in different participating countries by *different methods* and in different facilities. The results were collected, analyzed, and compared (Whittle 1991). This approach not only allowed each country to assess the performance of the employed method but provided a methodology and experimental data sets to evaluate simulation models of the future.

Special problems encountered during this evaluation process were studied in separate research items, e.g., modifications of turbulence models for low Reynolds numbers and thermal buoyancy (Moser 1988; Chen et al. 1990; Skovgaard and Nielsen 1991), simulation of air supply devices (Heikkinen 1991; Chen and Moser 1991), or the specification of temperature boundary conditions that account for radiation.

Simplified methods, that have a particular appeal to the design engineer, have also been evaluated, and in some cases even developed (Inard and Buty 1991; Nielsen 1991).

Complex and simplified simulation methods have been evaluated by applying them to four different benchmark cases (Whittle 1991), each representative of a particular basic air flow phenomenon, such as forced or natural convection. Measurements and simulations have been carried out simultaneously by different groups.

Available codes for air flow simulation are listed in (Liddament 1991). Some general air flow codes used in North America and elsewhere may be found in (Said 1988). And a critical review of computational fluid dynamics procedures was prepared for the ASHRAE by Baker and Kelso (1989).

Table 2 shows a partial list of air flow programs. The programs that were used by Annex-20 participants are indicated by country names. All programs listed use the finite-volume method (sometimes called finite-difference or finite-domain) with the exception of three, which employ the finite element method (FE). Subtask-1 workers have used nine of the listed algorithms. Denmark has its own three-dimensional research code. The Subtask-1 conclusions are restricted to the methods and programs actually used.

Name	*Origin of Code*		*Type*	*Annex-20 users*	*Remarks*
ARIA	Abacus,	UK	C		
ASTEC	Harwell	UK	C		
CALC-BFC	Chalmers	S	R	Sweden	own develop.
CHAMPION	TUD	NL	R		
EOL-3D	INRS	F	R	France	
EXACT3	NIST	USA	R	Canada	
FEAT		UK	?		FE
FIDAP	FDI	USA	C		FE
FIRE	AVL	A	C		
FLOTRAN	Compuflow		C		FE
FloVENT	FLOMERICS	UK	C		
FLOW-3D	Harwell	UK	C		
FLUENT	Creare	USA	C	Germany	
JASMINE	BRE-FRS	UK	R		fire, smoke
KAMELEON	SINTEF	N	R	Norway	
PHOENICS	CHAM	UK	C	Switzerland	
SIMULAR AIR	AVL	A	C	Germany	
STAR-CD	CD	UK	C		
TaskFLOW	Raithby	CDN	C		multigrid
TEAM	UMIST	UK	R	Denmark	2-D, ADI
TEMPEST	Battelle	USA	R		
WISH-3D	TNO	NL	R	NL, Finland	

Table 2 Computer codes for air flow simulation. Detailed information may be found in (Liddament 1991). (**R** = Research algorithms, **C** = commercial algorithms, **FE** = finite element method)

Technical Problems with Numerical Simulations

The major problems encountered during numerical simulation may roughly be grouped into five classes:

(1) Turbulence model at predominant Reynolds-number ranges and near walls.

(2) Natural and mixed convection at cold or warm surfaces.

(3) Simulation of air supply device.

(4) Problems with number and size of numerical control volumes (computational grid) and difficulty to reach grid independent solutions.

(5) Numerical procedure to reach solution of system of finite difference equations.

Attempts have been made to handle these problems, as documented in several Annex-20 technical reports. But many approaches still leave plenty of space for improvement. Some remarks to each of the above problems follow:

Problem (1): We have not tested turbulence models other than the widely-used k-epsilon closure. But experts have agreed that so-called low-Reynolds-number corrections are needed near walls and at low turbulence levels (Chen et al. 1990).

Problem (2): Three methods for dealing with heat transfer had been tried. The desirable approach is to prescribe wall temperature and have the program compute the heat flux. This works for forced convection (with wall functions) but is difficult for free convection because it requires a fine grid near the surface, low-Reynolds-number corrections in all equations, and careful setting of boundary conditions for computed variables. More work needs to be done on this method.

The second approach is to prescribe wall temperature and an empirical local heat transfer coefficient. This approach works well for window or radiator surfaces.

The third method is to estimate the local heat flux by empirical formulas and apply it in the simulation as a heat source (or sink) over the surface. If the correct heat flux (in W/m^2) is available, the method is reliable and does not require very fine grid, but the surface temperature is not automatically calculated. May also be used for internal heat sources such as computers.

Problem (3): A number of approaches were tested and reported (Chen and Moser 1991; Ewert et al. 1991; Heikkinen 1991; Skovgaard and Nielsen 1991). It would be helpful if the manufacturers of air diffusers would publish some near-field data (e.g., profiles in front of the device) with their technical specifications.

Problem (4): All computations were done with cartesian grids, where grid lines run through the entire flow domain ("tensor grid"). This mesh system has the disadvantage that grid refinements, also, extend from wall to wall, and into regions, where a fine resolution is not needed and cells with undesirable large aspect ratios may appear. Computational meshes with local grid refinement (Liddament 1991) would be preferable but were not tested. We found that grid independence was not reached.

Problem (5): Convergence of computations of flow fields with buoyancy effects in general is slow or inexistent. It is a good practice to look at changes of the flow *pattern* during iterations (Heikkinen and Piira 1991). Adjustments of relaxation factors during the solution process are normally required.

Experimental and numerical results suggest unsteady air motion under certain conditions at high Rayleigh number. However, this must still be verified. Investigations with spectral methods (Le Quéré 1992) indicate that characteristic frequencies of turbulent fluctuations in a heated cavity decrease as the Rayleigh number increases. These slower fluctuations may be interpreted as unsteady air flow. As this large-scale, slow "turbulence" cannot properly be represented by a Reynolds-averaged turbulence model, time-dependent simulation or large-eddy simulation (LES) would be appropriate.

Simulations of one test case (case **e**: mixed convection, summer cooling) have shown asymmetric flow fields at symmetric boundary conditions and geometry. Two stable solutions (with the jet turning left or right, respectively) and one unstable but symmetric solution were observed. The latter is obtained by just computing one half of the flow field and enforcing symmetry-plane conditions. The asymmetric flow pattern is confirmed by measurements (Blomqvist C. 1991).

An interesting extension of single-zone numerical flow field simulation was demonstrated by Schaelin (Schaelin, van der Maas, Moser 1992): The air exchange through an open door to the atmosphere. The motion of warm and cold air was modelled on a finite-volume mesh that extended from the room to outdoors.

Conclusions from evaluation of simulation methods

A detailed description of the benchmark exercises and quantitative comparisons of measurements and simulations with a critical evaluation are presented by Whittle (1991). Chen and Jiang (1992) give additional answers to significant questions regarding prediction of room air motion.

What can we say now about the performance of room air flow simulation techniques and about their applicability as design tools?

It is not a question of codes but of *methods* for flow simulation. The available computer codes are only one component of a method. Examples for some of the components or *techniques* of a method follow:

Turbulence model	k-ε two-equation model
Computational grid	non-uniform grid with 30x30x30 cells, finer near walls
Boundary conditions	for velocity and turbulence at supply device, for temperature or heat flux at surfaces, for concentrations at contaminant sources, etc.
Wall functions	appropriate for forced or free convection
Difference scheme	upwind, hybrid, PL, QUICK, etc.
relaxation technique	(techniques to accelerate convergence)
Computer code	(see table 2 for examples)

[**PL** = power law, **QUICK** = quadratic upstream interpolation for convective kinematics (Vogl and Renz 1991)].

The "best" method would be the one that combines the "best" techniques. The evaluation (Whittle 1991) and summary (Lemaire 1992) reports contain specific discussions of each technique.

To assess performance and viability of methods, performance *criteria* should first be specified. But each potential user has different requirements. Three groups of users may be distinguished:

- Designers and consultants
- Specialists of a CFD service-organization
- Scientists and students at research labs

Each of the three groups has different criteria as listed in table 3.

User: / *Criterion:*	**Designer**	**CFD Service**	**Research**
Cost of resources: staff, training, hard- & software	important		
Cost per case: labor, CPU-time	important		
Speed: response time	crucial	important	
Reliability & expediency: accuracy, detail, questions answered?	important	crucial	crucial
User-friendliness: ease of input, presentation of results	important	important	important
Interaction with other programs: building dynamics, radiation, CAD, etc.	important	important	

Table 3 Criteria for performance of simulation methods by different user groups. (**CPU** = central processor unit, refers to machine-time and fee for using computer, **CAD** = computer-aided design)

The entries in table 3 reflect the author's opinion as based on discussions during Annex-20 expert meetings. The weight "crucial" means that this user would *not* undertake a numerical simulation if that criterion were not met.

"Response time" refers to the span of time elapsed between formulation of task and delivery of results. "Expediency" expresses whether the computed output answers the questions of the user or client. For instance, have velocities and temperatures been calculated and output for the zones of interest?

The work of Subtask 1 (Lemaire 1992) and the evaluation of benchmark exercises (Whittle 1991) lead to these conclusions:

- CFD-simulations are useful when values of difficult-to-measure variables are needed in all points of the flow field.
- CFD-simulations are useful to study the sensitivity of flow patterns to small changes of conditions (trends).
- CFD-simulations are useful to predict air flow patterns for critical projects, i.e., when neither similar experience nor measured data exist (large spaces, unconventional ventilating systems, strong buoyancy effects).
- Simplified methods are useful to estimate the throw of supply air jet, the maximum velocity in the occupied zone, or the thermal plume in a radiator-window configuration.

In his evaluation report, Whittle (1991) concludes that CFD codes can predict room air movement with sufficient realism to be of use to design practice. Skill and experience are still required to use these codes. Many problems have been identified during this project, and he mentions three areas where further work is needed: Modelling of supply jet, modelling of turbulence, and thermal wall functions.

The Subtask-1 work demonstrates that CFD methods and several simplified approaches are now ready to be used as design tools. Initial use will be by specialists, but further developments of methods and improvements of the user-interface should soon lead to a wide acceptance.

PART 2: SIMULATION OF ROOM AIR THERMAL CONVECTION, PROBLEMS AND APPROACHES

Conclusions from IEA Annex 20 caused a group at ETH (the Swiss Federal Institute of Technology, Zurich) to study methods for the numerical simulation of free convection. Some existing methods and the state of the art were surveyed and some new approaches tried out. Three basic approaches to model the heat transfer on solid boundaries have been listed in Part 1, above:

(1) The numerical program computes the local heat flux,

(2) An empirical heat transfer coefficient, h, is input for each surface, and heat fluxes, q, are found from computed temperature differences according to $q = h\ \Delta T$.

(3) The local heat flux, q, itself is input. It may be estimated from a preliminary heat balance by using empirical correlations for h as in (2).

Part 2 of the present paper concentrates on approach (1) only, the boundary condition of prescribed temperature. To be universally useful, the CFD codes should be able to predict heat transfer for given boundary temperatures. Four topics are discussed below to illustrate the type of problems encountered when heat transfer by natural or mixed convection is modelled.

Numerical computation of thermal boundary layers requires fine numerical grid near walls, say, perhaps 10 grid lines within the zone of high gradients. A convenient variable grid is proposed in section 2.1. Other devices in combination with the k/ε- turbulence model concern the low-Reynolds-number corrections, variation of the turbulent Prandtl number, and the buoyancy term in the laminar sublayer, as discussed in 2.2-2.4. Table 4 shows how these improvements are introduced to forced and free convection.

forced convection	*free convection*
k/ε-model with low-Reynolds correction *Section 2.1*	→ add buoyancy term in viscous sublayer (Hachmann 1991) *2.2*
↓ add variation of turbulent Prandtl number (Yuan et al. 1992) *2.3*	↓ → *2.4*

Table 4 Desirable improvements to the numerical models of forced- and free-convection boundary layers. Approaches are discussed in the indicated sections.

2.1 Variable grid for simulation of cavity flow

Investigations by IEA Annex 20 have shown that accurate simulation of room air flow with forced convection is possible with the well-known k/ε-model for turbulence but requires low-Reynolds-number corrections and a variable grid with very fine resolution near the walls (Lemaire 1992). This state of the art is represented by square 'one' of table 4. In this section, a new grid function developed by Hachmann (Schaelin, Hachmann, Moser 1992) is proposed.

This grid function is designed to generate computational meshes that are very fine near two opposing walls but coarse in the bulk flow in between. The total number of control volumes should be optimized for a good trade-off between computer cost and accuracy. For a smooth grid distributions, a single analytic function is sought. Finite volume methods work best if the ratio of the widths of two adjacent cells does not exceed a certain value, a (for example, $a = 1.5$):

$$\Delta x_{j+1} / \Delta x_j \leq a \tag{1}$$

For the purpose of introducing the function, we assume that the physical length across the room (from wall to wall) runs from x = -1 (line j=-n) to x = +1 (line j=+n), i.e.,

$$x_j = j/n , \quad j = -n, \ldots n. \tag{2}$$

The ratio of adjacent cells is expressed by

$$\Delta x_{j+1/2} / \Delta x_{j-1/2} = f(x) \tag{3}$$

The function (3) must be symmetric with respect to both walls and should therefore satisfy $f(x) = 1/f(-x)$. Further, it should fulfill (1). The obvious choice is

$$f(x) = a^{-x} \tag{4}$$

and is shown in Figure 1 for different values of a.

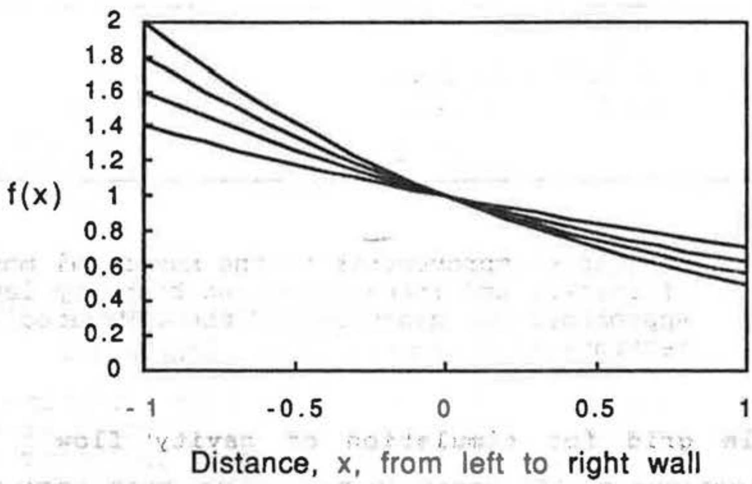

Figure 1 Ratio $f(x)$ of widths of adjacent numerical control volumes with extrema a and $1/a$ at the walls, equation (4), for a = 1.4, 1.6, 1.8, 2.0.

The grid spacing that satisfies equation (3) and (4) is

$$\Delta x_j = b \exp(-(rj/n)^2). \tag{5}$$

with

$$a = \exp(2r^2/n). \tag{6}$$

The parameter r is found from (6) for given a. It may be verified by differentiation that the Error Function (erf) defines the locations, x_j, of the grid lines that obey (5):

$$x_j = \frac{\mathrm{erf}(rj/n)}{\mathrm{erf}(r)} \tag{7}$$

The value of the constant b follows from differentiating (7) with respect to j. Two typical grids that use equation (7) with $r = 3$ are shown in figure 2.

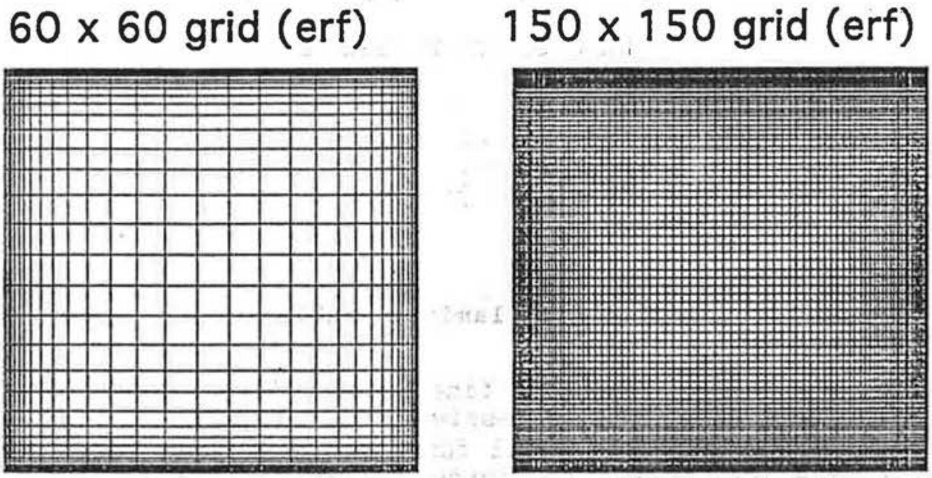

Figure 2 Two examples of grids generated by the Error Function, equation (7), with $r = 3$.

This grid has a high concentration of lines near the walls but almost uniform distribution in the core of the square. The relationship (6) between the maximum stretching factor, a, and the range, r, within which the Error Function is evaluated, is shown in figure 3 for different line numbers n.

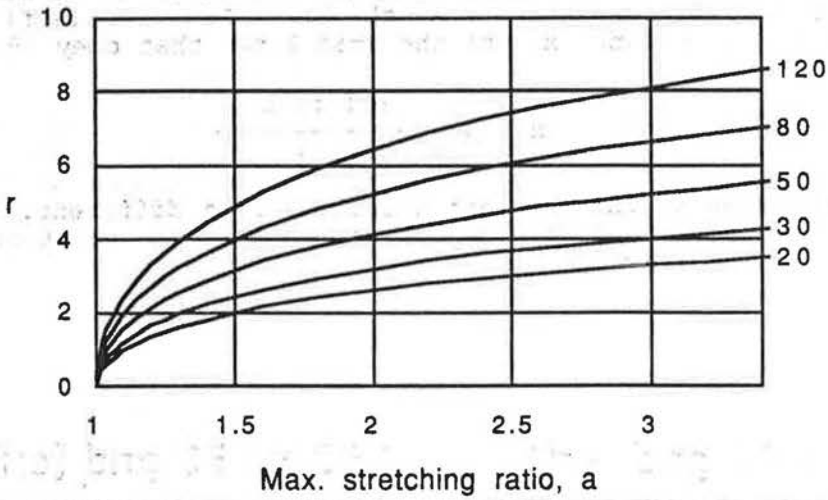

Figure 3 The parameter *r* of equation (7) as a function of the maximum ratio, *a*, of adjacent cells for different *n*. The grid has *2n* cells in one direction.

2.2 Asymptotic profile in laminar sublayer in free convection

In practical applications fine boundary layer grids in three dimensions result in excessively high element numbers, e.g $N \times N \times N = 100^3$. Therefore, wall functions have been introduced into the numerical models for forced convection. It would be desirable to have valid wall functions also for free convection flow. On heated (cooled) vertical surfaces, these boundary layers have an inner and outer region with different scaling laws (George and Capp 1979). A careful analysis of these layers was presented by Henkes (1990). Further work on scaling and similarity of profiles was reported by Cheesewright and Mirzai (1988), Tsuji and Nagano (1989), and Webb (1990).

At ETH, we looked at the laminar sublayer of the flow along a heated vertical plate (field 2.2 of table 4). Hachmann (1991) found that a buoyancy term should be added to the otherwise linear velocity profile. This modification seems important in an approach where the maximum of the velocity profile is resolved by the numerical grid, but the laminar sublayer modelled by an "inner" wall function.

The boundary layer equations for momentum and energy have been applied to the vertical flat plate. The profiles of temperature and velocity in the immediate vicinity of the surface (subscript w) were derived after further neglecting the convection terms. In the familiar u^+-y^+ notation, these are:

$$T^+ = T_w^+ - Pr\ y^+ \tag{8}$$

$$u^+ = y^+ - H\left(\frac{1}{2}T_w^+\ y^{+^2} - \frac{1}{6}Pr\ y^{+^3}\right) \tag{9}$$

The linear velocity profile in (9) is modified by second- and third-order terms in y^+ that account for buoyancy. In the sublayer, the conventional scaling with the skin friction velocity, u_τ, may be used. Temperature and heat flux are scaled as follows:

$$T^+ = \frac{\rho c_p u_\tau}{q_w}(T - T_\infty) \quad ; \quad q^+ = q / q_w \tag{10}$$

A non-dimensional parameter, H, in the profile (9) expresses the effect of buoyancy:

$$H = \frac{\nu\ \beta\ g}{\rho\ c_p u_\tau^4}\ q_w \tag{11}$$

H contains the product of the coefficient of thermal expansion, β, with the gravitational constant, *g*, as also found in the Grashof number, *Gr*. For the construction of wall functions only *local* variables are available, therefore, the combination *H* (instead of *Gr*) had to be introduced to avoid the use of characteristic lengths and temperature differences. T_∞ is the temperature outside of the boundary layer.

In figure 4, the velocity profile (9) is compared with an exact solution of the laminar boundary layer equations.

2.3 Variation of turbulent Prandtl number within the boundary layer

In most simulations carried out within the Annex-20 project, the turbulent Prandtl number, Pr_t, was kept constant at a value around 0.9. However, measurements indicate that Pr_t may assume different values in forced-convection boundary layers (e.g., Kays and Moffat 1988). With direct numerical simulation (DNS), turbulence quantities in the near-wall region may be determined more accurately than by experiments (Antonia and Kim 1991; Gilbert and Kleiser 1991). DNS studies also show variations of Pr_t.

To improve prediction of heat transfer, we looked into ways of accounting for variable Pr_t in the formulation of wall functions (Yuan et al. 1992). In a first attempt we selected existing formulas for the variation of the turbulent Prandtl number (Cebeci and Khattab 1975) in combination with the Van-Driest damping function for the viscous sublayer (field 2.3 of table 4).

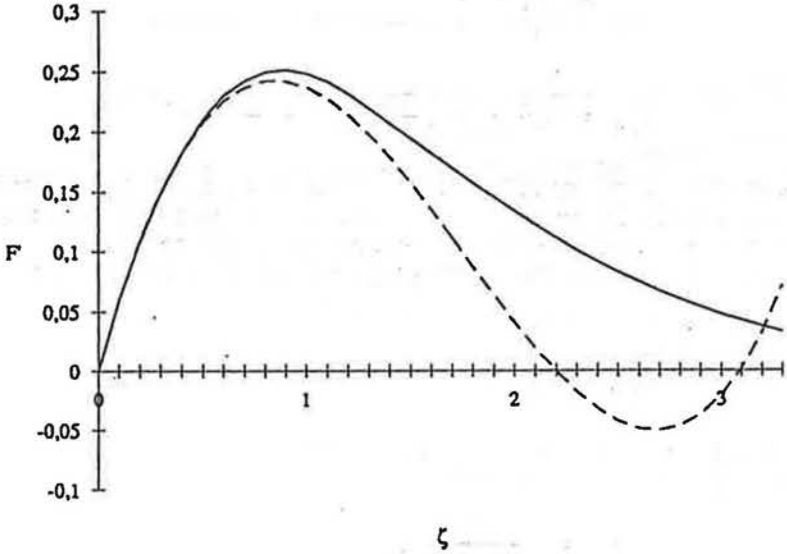

Figure 4 The laminar sublayer velocity profile (dashed line) that includes a buoyancy term, equation (9), is compared with the exact similarity solution of the laminar boundary layer equations (solid line). The new function is only applicable in a thin layer inside the velocity maximum.

Cebeci's expression is

$$Pr_t = \frac{K\,(1 - \exp(-y^+/A^+))}{K'(1 - \exp(-y^+/B^+))} \tag{12}$$

with $K/K'=0.9$, $A^+= 26$, and $B^+= 37$.

The turbulent boundary layer equations with Prandtl's mixing-length and Boussinesq's eddy-diffusivity concepts have been simplified by dropping all the derivatives with respect to the streamwise coordinate x. Cebeci's and Van Driest's modifications were applied. The resulting ordinary differential equations for $u^+(y^+)$ and $T^+(y^+)$ were numerically integrated. The numerical curve was fitted with four segments of the form

$$f = a \ln y^+ + b \tag{13}$$

where f is u^+ or T^+, respectively. The first segment of each function is $u^+ = y^+$ or $T^+ = Pr\, y^+$, respectively. These new wall functions are shown in figure 5 and compared with the traditional curves. Strictly speaking, the new functions are only applicable to forced convection. Yuan et al. (1992) show good agreement with the measurements by Moffat and Kays (1984).

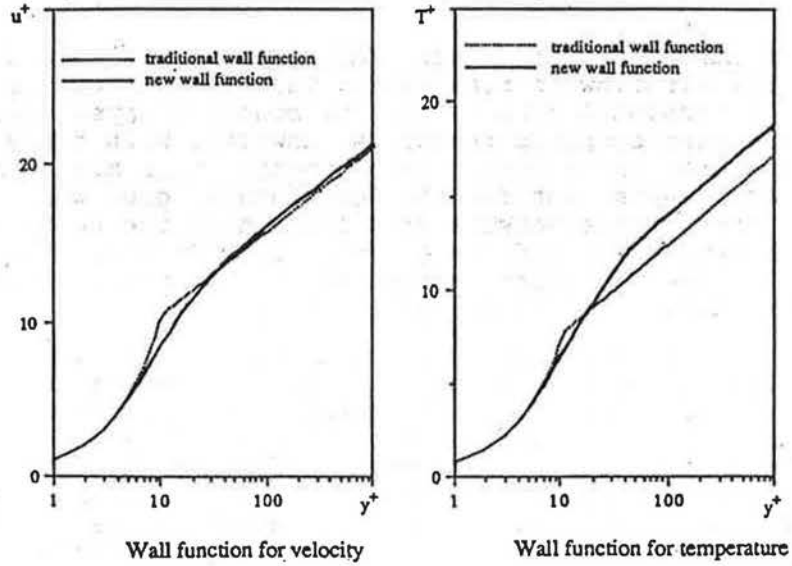

Figure 5 Comparison of the new wall function with variable turbulent Prandtl number with the traditional one.

2.4 Wall functions for boundary layers with velocity maxima

The last field of table 4 is still blank. Today, forced-convection wall functions are sometimes applied to surfaces with natural or mixed convection, but the predicted heat fluxes are not reliable and numerical results often are grid-dependent. The first steps of how we at ETH approach the problem of finding valid wall functions for boundary layers with maxima have been reported above. Further research is needed to find universal functions with appropriate scales for inner and outer regions.

RESULTS AND CONCLUSIONS

The work of IEA Annex 20 has shown the state of the art in building air flow simulation. Specialists in the field had an opportunity to try out their methods at common test cases and to make improvements of their models.

It has been found that correct prediction of heat transfer is essential in room aerodynamics for proper assessment of thermal convection. In the occupied zone of a room, air velocities should not exceed, say, 0.25 m/s, at the same time, thermal driving forces (caused by warm or cool surfaces or heat sources) may be quite high and play an important role in affecting the "slow" motion of the air. If dimensions of a room become bigger, the influence of buoyancy effects grows in relation to forced-flow effects. With increasing height, the momentum of a plume builds up (Schaelin and Kofoed 1992), but the velocity in an isothermal jet decays with distance. So the prediction of thermal convection becomes more important in large enclosures.

Direct numerical simulation (DNS) is not available for some time to model air flow in real rooms. The approach with a very fine three-dimensional mesh to resolve boundary layers still requires too much computer resources. However, with today's progress in hard- and software developments, this might change in a couple of years. But for the near future, good wall functions, that save expensive grid lines near the surfaces, will make numerical methods more attractive. If only 30 elements are needed in one direction instead of 60, the total number of three-dimensional elements is reduced to 1/8.

Buildings are ventilated or conditioned to provide a pleasant climate for the inhabitants. Humans feel or are affected by thermal sensation, by chemical or particulate contamination, and relative humidity. The air velocity has no direct effect. The thermal comfort is influenced by radiative and convective heat transfer on the skin. Thus, human comfort depends on indoor air quality, relative humidity, radiation, and thermal convection. Therefore it is of prime importance to understand thermal convection.

ACKNOWLEDGEMENTS

This work was partially supported by the Swiss Federal Office of Energy, BEW. I like to thank Peter Hachmann, Arthur Huber, Alois Schälin, and Xiaoxiong Yuan for their creative ideas, technical contributions and fruitful discussions.

REFERENCES

Antonia R. A.; J. Kim 1991. Turbulent Prandtl number in the near-wall region of a turbulent channel flow, **Int. J. Heat Mass Transfer**, Vol. 34, No. 7, pp.1905-1908.

Baker A. J., Kelso R. M. 1989. Computational fluid dynamics procedures applied to prediction of room air motion, Contract Technical Report, **ASHRAE TR** on 464-RP, Report CFDL/89-2, Dept. Engineering Science and Mechanics, University of Tennessee, Knoxville, June.

Blomqvist C. 1991. Measurements of test case e (mixed convection, summer cooling), **IEA Annex 20**, Subtask 1, Research Item 1.17, Working Report, The National Swedish Institute for Building Research, Gävle, Sweden, March.

Cheesewright R.; M. H. Mirzai 1988. The correlation of experimental velocity and temperature data for a turbulent natural convection boundary layer, Paper C140/88, Proceedings 2nd UK National **Conference on Heat Transfer**, pp. 79-89, Glasgow.

Chen Q. and Moser A. 1991. Simulation of a multiple-nozzle diffuser, **IEA Annex 20**, Subtask 1, Research Item 1.20. 12th AIVC Conference, Ottawa, Canada, September 24-27.

Chen Q., Moser A., Huber A. 1990. Prediction of buoyant, turbulent flow by a low-Reynolds-number k-epsilon model, **ASHRAE Transactions**, Vol. 96, Part 1, 1990. Paper AT-90-2-2(3366) of ASHRAE Atlanta Winter Meeting, Feb.

Chen Q.; Z. Jiang 1992. Significant Questions in Predicting Room Air Motion. **ASHRAE Transactions**, Vol. 98, Part 1, 1992. Paper AN-92-9-1 of ASHRAE Anaheim Winter Meeting.

Ewert M.; U. Renz; N. Vogl; M. Zeller 1991. Definition of the flow parameters at the room inlet device, - measurements and calculations. **IEA Annex 20**, Subtask 1, Research Item 1.43. 12th AIVC Conference, Ottawa, Canada, September 24-27.

George W. K.; S. P. Capp 1979. A Theory for Natural Convection Turbulent Boundary Layers Next to Heated Vertical Surfaces. **Int. J. Heat and Mass Transfer**, Vol.22, No.6 (June), pp 813-826.

Gilbert N.; L. Kleiser 1991. Turbulence model testing with the aid of direct numerical simulation, Paper 26-1, **Eighth Symposium on Turbulent Shear Flows**, Technical University of Munich, September 9-11.

Hachmann P. 1991. Freie Auftriebsströmungen in rechteckigen Behältern und an der vertikalen Platte. **Diplomarbeit, ETH** Zürich, Lab. für Energiesysteme, March 30, 1991 (in German).

Heikkinen J,; K. Piira 1991. Simulation of simple (two-dimensional) test cases. **IEA Annex 20**, Subtask 1, Research Item 1.46SF, Annex Report, Technical Research Center of Finland (VTT), Espoo, Finland.

Heikkinen J. 1991. Modelling of a supply air terminal for room air flow simulation, **IEA Annex 20**, Subtask 1, Research Item 1.24. 12th AIVC Conference, Ottawa, Canada, September 24-27.

Henkes R. A. W. M. 1990. Natural-Convection Boundary Layers. Ph.D. Thesis, **TU Delft**, The Netherlands.

Inard C., Buty D. 1991. Simulation of thermal coupling between a radiator and a room with zonal models, **IEA Annex 20**, Subtask 1, Research Item 1.26. 12th AIVC Conference, Ottawa, Canada, September 24-27.

INTERNATIONAL ENERGY AGENCY, IEA, 1989. Air flow patterns within buildings, - Annex XX of the IEA Implementing Agreement on Energy Conservation in Buildings and Community Systems. Legal text with description of project objectives, means, and responsibilities, etc., drafted by the **IEA R&D** staff, reviewed by the IEA Office of the Legal Counsel, and adopted by the Executive Committee. No.0633R/13.6.89.

Kays W. M.; R. J. Moffat 1988. The near-wall region of the turbulent boundary layer: Some results from heat transfer measurements. **Near-Wall Turbulence**. S.J. Kline and N. H. Afgan (eds.), Hemisphere.

Le Quéré 1992. Proceedings of the **EUROTHERM/ERCOFTAC** Workshop on *Benchmark computation and experiment for turbulent natural convection in a square cavity*, March 1992, Delft, NL. Berlin, New York: Springer.

Lemaire A. D. 1992 (editor). Room air and contaminant flow, - evaluation of computational methods, **IEA Annex 20**, Subtask 1, Annex Report, TNO, Delft, Netherlands, May.

Liddament M. W. 1991. A review of building air flow simulation, IEA, **Air Infiltration and Ventilation Center**, Technical Note AIVC TN 33, March.

Moffat R. J.; W. M. Kays 1984. A review of turbulent-boundary-layer heat transfer research at Stanford, 1958-1983. **Advances in Heat Transfer**, Vol. 16, Hartnett and Irvine (eds.).

Moser A. 1988. Low Reynolds number effects in single room air flow, **IEA Annex 20**, Subtask 1, Research Item 1.1, Working Report, ETH, Zurich, Switzerland, November.

Moser A. 1991. The Message of Annex 20: Air Flow Patterns within Buildings. IEA Annex 20, **12th AIVC Conference**, Ottawa, Canada, September 24-27.

Nielsen P. V. 1991. Models for the prediction of room air distribution, **IEA Annex 20**, Subtask 1, Research Item 1.5. 12th AIVC Conference, Ottawa, Canada, September 24-27.

Roulet C.-A., Cretton P., Fritsch R., Scartezzini J.-L. 1992. Stochastic model of inhabitant behavior in regard to ventilation, **IEA Annex 20**, Subtask 2, Research Item 2.2, Annex Report, LESO-EPFL, Lausanne, Switzerland, April.

Roulet C.-A.; Vandaele L. 1991. Air flow patterns within buildings, measurement techniques, Joint IEA Annex-20 / **AIVC Report**, Air Infiltration and Ventilation Center, Technical Note TN 34, September.

Said M. N. 1988. Two- and three-dimensional computer codes developed for applications other than room air flows, **IEA Annex 20**, Subtask 1, Research Item 1.10, Working Report, IRC-NRC, Ottawa, Canada, November.

Schaelin A.; J. van der Maas; A. Moser 1992. Simulation of Air Flow through Large Openings in Buildings. **ASHRAE Transactions**, CFD-Symposium Paper at ASHRAE 92 Annual Meeting Baltimore.

Schaelin A.; P. Hachmann,; A. Moser 1992. Flow Field Calculations of Cavities with Low Re Number Models. Presentation at the **Eurotherm/ERCOFTAC Workshop** *Benchmark Computation and Experiment for Turbulent Natural Convection in a Square Cavity*, Delft, The Netherlands. Proceedings, Berlin, New York: Springer.

Schaelin A.; P. Kofoed 1992. Numerical simulation of thermal plumes in rooms with temperature stratification, **ROOMVENT'92 Conference** Proceedings, Aalborg, Denmark.

Skovgaard M.; P.V Nielsen 1991. Modelling complex inlet geometries in CFD, - applied to air flow in ventilated rooms, **IEA Annex 20**, Subtask 1, Research Item 1.24. 12th AIVC Conference, Ottawa, Canada, September 24-27.

Skovgaard M.; P.V. Nielsen 1991. Numerical investigation of low Reynolds number effects, **IEA Annex 20**, Subtask 1, Research Item 1.1. 12th AIVC Conference, Ottawa, Canada, September 24-27.

Tsuji T.; Y. Nagano 1989. Velocity and temperature measurements in a natural convection boundary layer along a vertical flat plate, **Experimental Thermal and Fluid Science**, Vol. 2, pp 208-215.

Van der Maas J. (Editor) 1992. Air flow through large openings in buildings, **IEA Annex 20**, Subtask 2, Research Item 2.1, Annex Report, LESO-EPFL, Lausanne, Switzerland, April.

Vogl N.; U. Renz. 1991. Simulation of simple test cases b1, b2, b3, **IEA Annex 20**, Subtask 1, Research Item 1.19, Working Report, RWTH Aachen, Germany, March.

Webb S. W. 1990. A local similarity model for turbulent natural convection along a vertical surface, Fundamentals of Natural Convection, **AIAA/ASME** Thermophysics and Heat Transfer Conference, June, pp. 105-112, Washington.

Whittle G. E. 1991. Evaluation of measured and computed test case results from Annex 20, Subtask 1, IEA Annex 20, Subtask 1, Research Item 1.35. **12th AIVC Conference**, Ottawa, Canada, September 24-27.

Yuan, X.; A. Huber; A. Schaelin; P. Hachmann; A. Moser 1992. New Wall Functions for the Numerical Simulation of Air Flow Patterns in Rooms. **ROOMVENT'92 Conference** Proceedings, Aalborg, Denmark.