Proceedings of the International Conference on Air Distribution in Ventilated Spaces, ROOMVENT-87, Vol.4a, Stockholm, pp.1-14, 10-12 June, 1987.

EXPERIMENTS AND 2D APPROXIMATED COMPUTATIONS OF 3D AIR MOVEMENT, HEAT AND CONCENTRATION TRANSFER IN A ROOM

BY

CHEN QINGYAN AND J. VAN DER KOOI

Section of Indoor Climate Technology
Delft University of Technology
Mekelweg 2, 2628 CD Delft
The Netherlands

ABSTRACT

A few approximated methods were used to predict 3D mixed air movement of natural and forced convection with concentration transfer in a room by a 2D computer code CHAMPION SGE. The computation method involves the solution, in finite-domain form, of two dimensional equations for the conservation of mass momentum, energy, concentration, turbulence energy and dissipation rate, with wall function expressions for solid boundary conditions. Corresponding measurements of temperature distributions and flow patterns were used for comparison.

The following cases were investigated:
- natural convection in the room;
- the room with solar radiation through the window and with cold air supply, also with a CO_2 gas source.

The approximated methods were:
- treating the heat transfer through the walls in the Z-direction (see Fig.2) as thermal sources or sinks for the finite-domain equation;
- concentrating the heat transfer through the walls in X- and Y-direction; and
- using equivalent inlet temperatures.

Mention comparisons with test chamber results

NOMENCLATURE

A	grid surface area	m^2
a	coefficient of the finite-domain equation	-
b_1, b_2, b_3, b_4, b_5	coefficients in the heat exchange coefficient equation	
C	concentration	$Kmol/m^3$
C_p	constant-pressure specific heat	W/kgK
C_1, C_2, C_3, C_D, C_μ	coefficients in approximated turbulent transport	
E	function of wall roughness (E=9.0)	-
g	gravity	m/s^2
H	specific enthalpy	J/kg
H	height of the room	m

h	heat exchange coefficient	W/m^2K
h	height of the cooling inlet units	m
L	length of the room	m
l	characteristic length scale	m
p	pressure	Pa
Q	heat flux	W/m^2
S	source term of general fluid property	-
T	temperature	K
u	velocity component in X-direction	m/s
V	flow velocity	m/s
v	velocity component in Y-direction	m/s
W	half of the room width	m
w	width of the cooling inlet units	m
x	general coordinate	m
x,y	two directions	m
y	distance to walls	m

GREEK SYMBOLS

Γ	diffusive coefficient	$N.s/m^2$
ε	dissipation rate of turbulence energy	m^2/s^2
κ	von Karman's constant (κ=0.4)	-
μ	fluid viscosity	$N.s/m^2$
ν	fluid kinetic viscosity	m^2/s
ρ	fluid density	kg/m^3
σ	Schmidt or Prandtl number	-
τ	Reynold's stress	N/m^2
ϕ	general fluid property	-

SUBSCRIPTS

eff	effective coefficient
H	specific enthalpy
i,j	subscripts denoting coordinate directions
in	inlet
k	kinetic turbulence energy
o	reference point for pressure
out	outlet
mass	mass flow rate
P,N,S,E,W	subscripts denoting directions
l	laminar
t	turbulent
w	wall surface
ε	dissipation rate of turbulence energy

SUPERSCRIPTS

$\rightarrow$	vector

1. INTRODUCTION

In addition to human comfort conditions, detailed information on local temperatures and velocity components and concentration of harmful gas is required. We should be able to deal with them effectively. This ability can result from an understanding of the nature of the process and from methods which can predict them quantitatively. The calculation method for heating and cooling load has been changed from traditional steady, one dimensional to transient, three dimentional. But most of the cooling load programs are based on the one-air-point-model, that is, the values of the whole temperature field in the room are taken as uniform. This is not true in preactice. Mostly there is a gradient of temperature in a room. It is the gradient which is generated from the influence of buoyancy and plays an important role in the heat transfer.

Armed with these expertises, the designer of an air-conditioning plant equipment can ensure the desired performance - the designer is able to choose the optimum design from among a number of alternative possibilities. The power of prediction enables us to operate existing equipment more efficiently.

Since 1967, many computer codes have been invented for calculating the fluid flow, heat and concentration transfer, such as TEACH [1], GENMIX [2], CHAMPION 2/E/FIX [3] and PHOENICS [4]. A lot of mumerical studies of buoyancy affected flow have been reported in the last decade, such as Nielson et al [5], Jedrzejewska-Sibak et al [6], Hjertager et al [7] and Schmitz et al [8]. Chen Qingyan et al [9] gave comparisons between computation and measurement of two and three dimensional problems under steady and transient boundary conditions. But the study has shown that 3D computations were very expensive and two dimensinal computations were limited in their applications, because in practice pure 2D cases are rather exeptional. In the present study, a few approximated computations of 3D air movement, heat and concentration transfer in a room will be introduced by a 2D computer code CHAMPION SGE in order to reveal the nature of mixed convection in more economical ways.

2. CHAMPION SGE FORMULATION AND STRUCTURE

The differential equations of which CHAMPION SGE solves the discreted versions are those which express the physical laws of 'conservation' of mass momentum, concentration and energy [3][10]. All these equations are expressed in a single form:

$$\mathrm{div}(\rho\vec{V}\phi-\Gamma_\phi \,\mathrm{grad}\ \phi)=S_\phi+S_{Buoyancy} \qquad (1)$$

where ϕ, Γ_ϕ, S_ϕ and $S_{Buoyancy}$ are given in Table 1.

The equations are derived, as mentioned above, by integrating the equations over a finite set of sub-domains. The resulting equations normally connect each grid point $\phi(P)$ with four neighbouring ϕ's, namely those at the north (N), east (E), south (S) and west (W). In algebraic form, the relations are:

$$a_P\phi_P= a_N\phi_N+a_S\phi_S+a_E\phi_E+a_W\phi_W+S_{sources} \qquad (2)$$

Table 1. Values of ϕ, Γ_ϕ, S_ϕ and $S_{Buoyancy}$ terms

ϕ	Γ_ϕ	S_ϕ	$S_{Buoyancy}$
1	0	0 (continuity)	0
V	μ_{eff}	$-\partial p/\partial x_i+\partial(\mu_{eff}(\partial V_j/\partial x_i+\partial V_i/\partial x_j))/\partial x_i$	$\rho\beta g_i\theta$
H	μ_{eff}/σ_H	0	0
k	μ_{eff}/σ_k	$G-\rho\varepsilon$	G_B
ε	$\mu_{eff}/\sigma_\varepsilon$	$\varepsilon(C_1G-C_2\rho\varepsilon)/k$	$C_3\varepsilon G_B$
C	μ_{eff}/σ_c	0	0

V – u and v; velocity components in x and y direction respectively.

$\mu_{eff}=\rho(\nu+\nu_t)$; ν and ν_t are laminar and turbulent viscosity respectively

$\mu_t=C_Dk^2/\varepsilon$

$\Theta=T-T_0$; excess temperature, where T_0 is a reference value.

$G=\mu_t(\partial V_i/\partial x_j+\partial V_j/\partial x_i)\partial V_j/\partial x_i$

$G_B=\rho\beta g_i\dfrac{\nu_t}{\sigma_H}\dfrac{\partial\theta}{\partial x_i}$

$C_1=1.44$, $C_2=1.92$, $C_D=0.09$, $C_3=1.44$, $\sigma_k=1.0$, $\sigma_\varepsilon=1.3$, $\sigma_H=0.9$, $\sigma_C=1.14$.

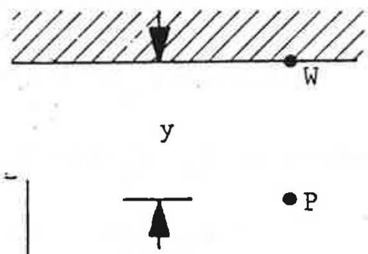
where a's are positive coefficients, obeying:

$$a_P \geq a_N+a_S+a_E+a_W \tag{3}$$

The finite-domain equations are solved by an algorithm which is line by line 'SIMPLE', in 'NEAT' arrangement [3].

The boundary conditions are treated as wall functions [3][11]:

$$\left.\begin{aligned}\tau_w&=\mu_l\frac{du}{dy}\\ Q&=\frac{(\tau_w/\rho)^{1/2}\rho\, C_p(T_p-T_w)}{\sigma_l\, y+}\end{aligned}\right|\ \text{when } y+ < 11.5 \tag{4}$$



and

$$\left.\begin{aligned}\frac{V}{(\tau_w/\rho)}\, C_\mu^{1/4}\, k_p^{1/2}&=\frac{1}{\kappa}\ln\left(E\, y_p\frac{(C_\mu^{1/2}k_p)^{1/2}}{\nu}\right)\\ Q&=\frac{(\tau_w/\rho)^{1/2}\rho\, C_p(T_p-T_w)}{\sigma_l 11.5+(y^+-11.5)\sigma_t\frac{\mu_l}{\mu_t}}\end{aligned}\right|\ \text{when } y+ \geq 11.5 \tag{5}$$

Where the velocity variation near the wall is based on the value of y+. Here y+ is calculated from:

$$y+ = C_{\mu}^{1/4}\rho k^{1/2}y/\mu_1 \qquad (6)$$

And τ_w is assumed to be uniform from the wall to the adjacent grid line. τ_w is a boundary condition for the u and v equation and enters the generation term for the near wall k:

$$S_k=\tau_w \frac{\partial u}{\partial y} - \frac{C_D\rho^2k^2}{\tau_w}\frac{\partial u}{\partial y} \qquad (7)$$

For the dissipation rate, the length scale is assumed to be proportional to the distance from the walls, consequently:

$$\varepsilon_P=C_D^{3/4}k_P^{3/2}/(\kappa y) \qquad (8)$$

The computer code CHAMPION SGE (CHAMPION stands for Concentration, Heat And Momentum Program Instruction Outfit, the final N being added for euphony, and SGE for Satellite, Ground-station and Earth structure) is of the capacity to solve the two dimensional concentration, heat and momentum transfer of turbulent flows. It has been developed from CHAMPION 2/E/FIX [3] in following ways:

- Extend the original code 2/E/FIX for non isothermal computations. For density of fluid, boussinesq approximation is also available;

- Develop the code for concentration computations;

- Introduce a universal formula for all kinds of boundary condition descriptions which are often met in practice. That is:

$$S_{\phi,Boundary}=[C_{\phi}+C_{mass}(V_{mass}-p)][V_{\phi}-\phi] \qquad (9)$$

- Provide some new advices for getting convergent computational results and obtaining the results in a time as short as possible (such as false time step);

- Construct a graphics computer code GROCS (which stands for Graphical Representation Of CHAMPION SGE) which provides display facilities for the numerical predictions obtained from the CHAMPION SGE code;

- Reorganize the original code CHAMPION 2/E/FIX into a user-friendly data input, general purpose code, which can easily be adapted for the solution of particular problems, in such a way that users can communicate with the code in very limited subroutines; and

- Derive a very detailed description of the governing equations, boundary conditions, solving procedure and code structure [12]. A general instruction manual is also available. The code can be installed in a micro-computer.

The CHAMPION SGE code is constructed from a central 'Earth' code, which is the main solver and a 'Satellite', which provides data input, defining specific problem. For special purposes, there are possibilities to provide additional coding which interacts with 'Earth' during operations. This function is performed by the so-called 'Ground-station' subroutine. The structure of the CHAMPION SGE code is shown in Fig.1 and is of very similar form as PHOENICS (1983 version). Another advantage is that the complete listing of the CHAMPION SGE code is available.

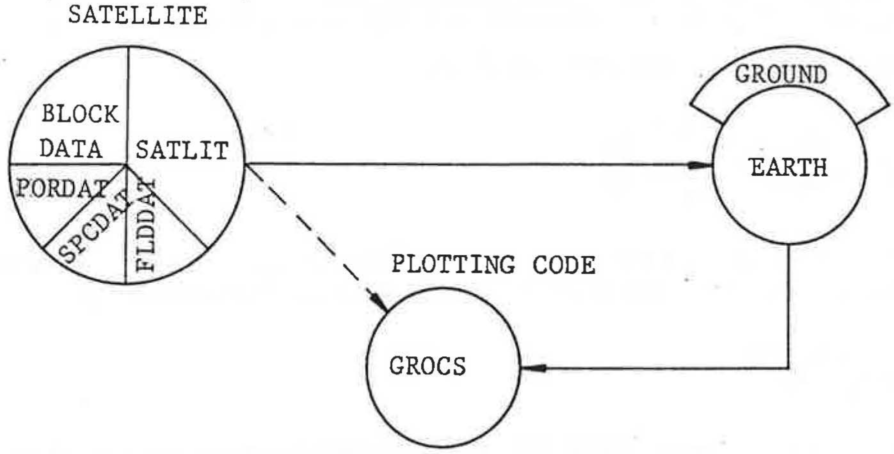


Fig.1 CHAMPION SGE and GROCS structure

3. EXPERIMENTS

The experiment were done in a rectangular climate room 5.6m long, 3.2m wide and 3.0m high and was shown in Fig.2. There were two kinds of inlet units, one on the floor near the window and the other on the rear wall near the ceiling. The outlets were located on the rear wall near the floor. The experiments dealt with the measurements of the temperature fields, the heat exchange coefficients and velocity fields. The temperature fields were measured through thermo-couples by means of a data logging system and converted into printed results by a micro-computer. The

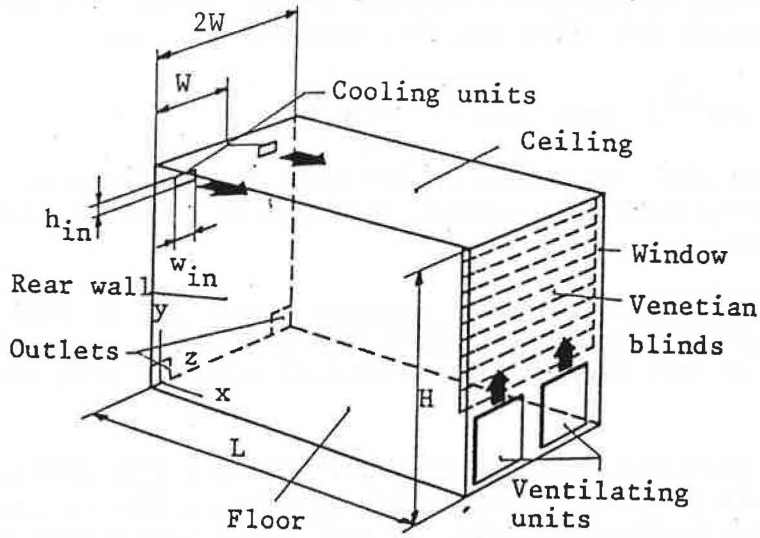


Fig.2 The geometry of the room and notation

heat exchange coefficients were measured via heat flux meters and the temperature differences between walls and the air points 10 cm near the correspondent walls. The velocity pattern was observed by means of smoke.

4. A FEW APPROXIMATED METHODS FOR 3D COMPUTATIONS

4.1 THERMAL SOURCES/SINKS FOR HEAT TRANSFER IN THE Z-DIRECTION METHOD

Generally speaking, a 2D computer code can only be used in the practical situations which are more or less two dimensional. For example, in a room with natural convection, the cold window and hot radiator under it are of the same width as the room. The heat transfer through the side walls (Z-direction) can be ignored if its adjacent rooms are of more or less the same conditions. But in many cases, for instance if the side walls are external ones, the errors caused by the approximation will be big and unacceptable. In order to solve the problem above, an approximated method is to consider the heat transfer through the side walls as additional thermal sources or sinks.

From the governing equation (1), $S_H=0$ in normal conditions. The additional thermal sources (if the temperatures on the side walls are higher than those in the corresponding grid points) or sinks (if those are lower) can be put into the term S_H via following equation:

$$S_H=h\ (T_W-T_P)\ A_P \tag{10}$$

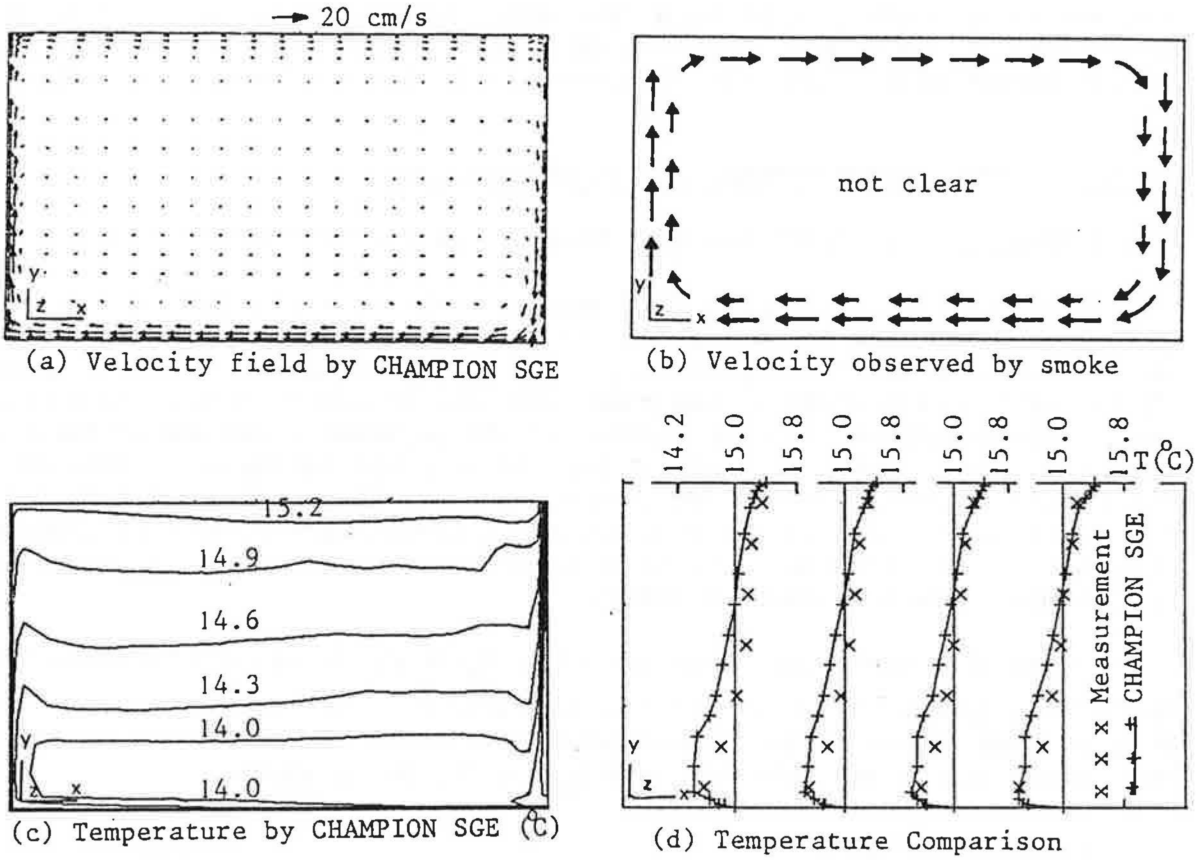
where T_P is the temperature values in cell P and T_W is the temperature on the side walls. A_P is cell area and h is of the physical meaning of the convective heat exchange coefficient.

Because the buoyancy driven force is the most important one in the heat transfer through the side walls (Z-direction), the coefficients, h's may be computed via [13]:

$$h=\{[b_1(\Delta T/L)^{b_3}]^{b_5}+[b_2(\Delta T)^{b_4}]^{b_5}\}^{1/b_5} \tag{11}$$

and $\Delta T=|T_W-T_P|$. Where L is the characteristic length scale, which is the height of the room (b_1=1.5, b_2=1.23, b_3=1/4, b_4=1/3 and b_5=6).

The physical model of the case studied here is shown in Fig.2 above. In this case all ventilating units and cooling units did not function. The room temperature was 15.0 °C and the temperature on the window surface was 6.0°C. The temperature on the outside of the side walls was 22.0 °C which resulted in a 2.0 °C higher temperature on the inside of the side wall surfaces than that in the room. The corresponding Rayleigh number in this case was $2.2*10^{10}$. The measured and predicted results by CHAMPION SGE are shown in Fig.3(a-d). The agreement between computation and experiments is good.



(a) Velocity field by CHAMPION SGE

(b) Velocity observed by smoke

(c) Temperature by CHAMPION SGE (°C)

(d) Temperature Comparison

Fig.3 Velocity and temperature distributions of a natural convection system by the first computed method

The second example is forced convection. The two ventilating units of the cooling system (100cm*0.8cm each) are located near the window as shown in Fig.2. The inlet mass flow for the room was 0.075 m^3/s (that is ventilation rate 5 times/h) which corresponded with a Reynolds number 4800. A concentrated heat (950W) was put on the venetian blinds. The outlet temperature was controlled at 23.0 °C. The inside surface temperature of the side walls was 21.0 °C. The two horizontal cooling units on the rear wall were closed in this case. A supply of CO_2 gas was put concentrated in the line (x=1.1m, y=1.2m and z=0.0-3.0m). The measured and computed results by the CHAMPION SGE code and PHOENICS (3D) code are shown in Fig.4(a-f). The agreement between the 2D, 3D computations and experiments are rather good. The measured results for CO_2 distribution are not available at this moment, but the computational results seem resonable (see Fig.4(f)). The 2D CHAMPION SGE and 3D PHOENICS give very similar results. However, the 3D PHOENICS computation is very expensive, it needs about 50 minutes CPU time in IBM 3083-JX1 computer for the grid number 9*18*27 [9]. And only 2 minutes CPU time is required for CHAMPION SGE for the grid number 18*27. For two dimensional computations, it is easy to get convergent results.

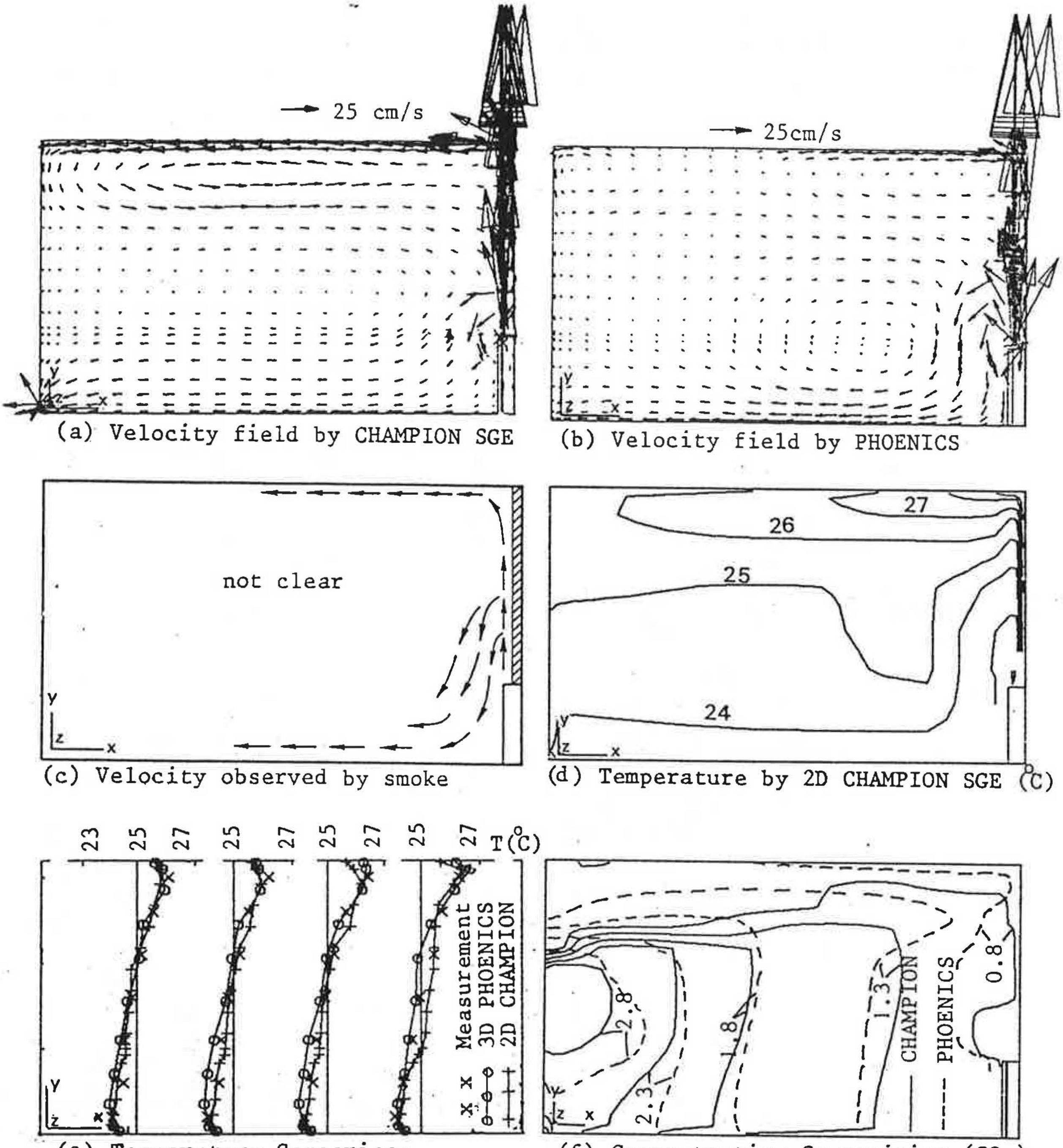


Fig.4 Velocity, temperature and concentration distributions of a forced convection system by the first computed method

4.2 CONCENTRATED METHOD

When the width of the ventilating units is much smaller than the width of the room, like the cooling units on the rear wall shown in Fig.2, normal 2D computation will cause very big error. If we just consider the shaded section in Fig.5(a) for half of the room (because of the symmetry), all heat transfer through the ceiling, floor, window, rear wall and parapet are put together into the corresponding shaded section. The heat transfer through the side walls may also be set via thermal sources or sinks as treated in the section 4.1. By this method, we can update the heat transfer through the shaded section by increasing the heat exchange coefficients W/w_{in} times or by calculating the total heat flux values over the whole width of the room and setting this as thermal sources or sinks on the boundary cells.

In the case described in Fig.5(a), the inlet mass flow through the cooling units in the rear wall (30.0cm*2.0cm each) was 0.075m³/s for the whole room. The corresponding Reynolds number was 15000. A concentrated heat (950W) was put on the venetian blinds. The outlet temperature was controlled at 23.0 °C. The ventilating units near the window were closed for this case.

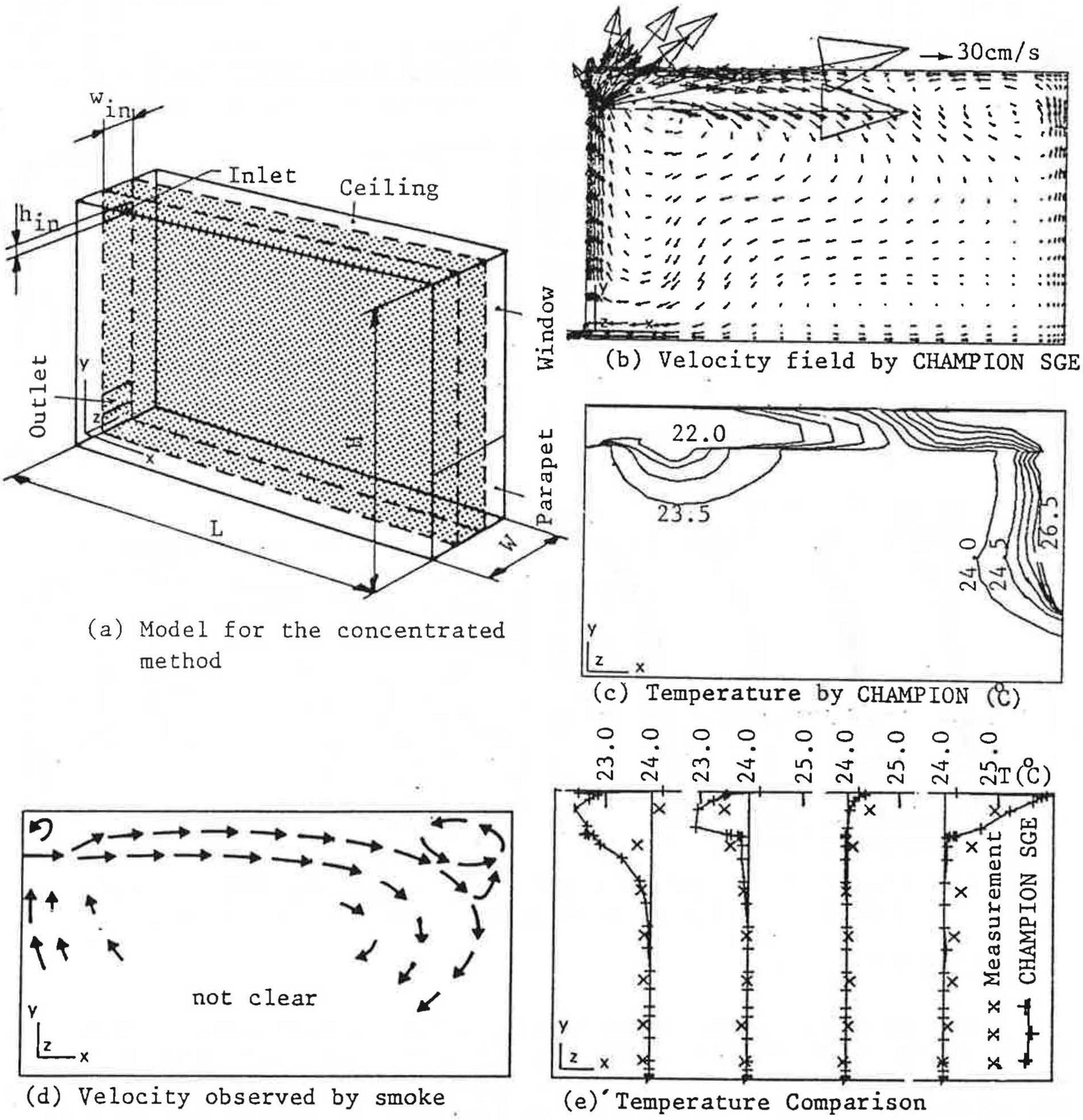


Fig.5. Velocity and temperature distributions of a forced convection system by the second computed method

The computations by CHAMPION SGE and the measurements of the velocity and temperature distributions in the shaded section are shown in Fig.5(b-e). The computational results, comparing with the measurements, are quite reasonable but there are some discrepancies. These differences may have been caused by the approximated method. In fact, the method increases the mass flow in the room and limits the mass transfer in the Z-direction. The average temperature and the gradients in the room are good in agreement with measurements. This is enough for testing and for cooling load computations which consider the temperature gradients in the room [14]. The computions by PHOENICS did not show better results.

4.3 EQUIVALENT INLET TEMPERATURE METHOD

Another method for getting approximated numerical predictions on temperature and velocity distributions is to keep the inlet opening height and inlet velocity constant and to adjust the inlet temperature to satisfy energy balance in the room. This, in fact, assumes the inlet openings to be extended over the whole width of the room. For example, in the model shown in Fig.5(a), the inlet size is $w_{in}* h_{in}$, inlet velocity v_{in}, inlet temperature T_{in}. In order to predict this case by the 2D computer code CHAMPION SGE, the inlet height and velocity for the input data remain as h_{in} and v_{in}, while the adjusted inlet temperature $T_{d,in}$ may be calculated from the following formula:

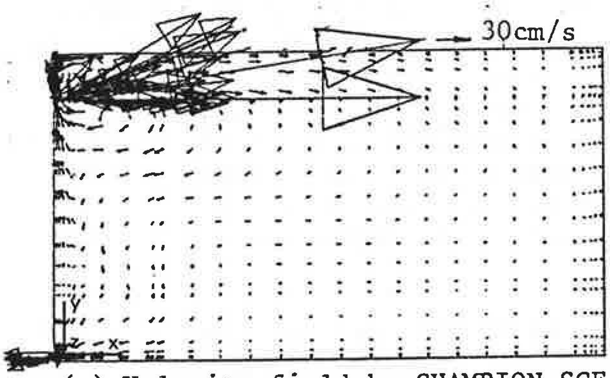
$$(T_{d,in}-T_{out})*v_{in}*W*h_{in}=(T_{in}-T_{out})*v_{in}*w_{in}*h_{in} \qquad (12)$$

that is

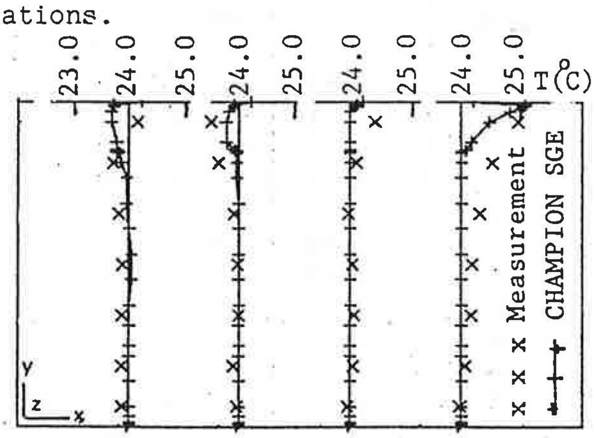
$$T_{d,in}=T_{out}+(T_{in}-T_{out})*w_{in}/W \qquad (13)$$

When $w_{in}=W$, $T_{d,in}=T_{in}$, this becomes a real 2D problem. This method can be explained as that the mixing in the zone near the inlet is so well that the air temperature increases rapidly.

The calculated results of the case shown in Fig.5(a) are given in Fig.6(a-b). The discrepancies are not too big in this case. The results are acceptable in many practical situations.



(a) Velocity field by CHAMPION SGE



(b) Temperature Comparison

Fig.6. Velocity and remperature distributions of a forced convection system by the third computational method

Another way is to use an inlet model method in which the velocity, temperature and concentation distributions in the region near the inlet can be calculated by jet formulae and put as a kind of boundary condition for numerical boundary setting in the near inlet area.

5. DISCUSSION

The CHAMPION SGE code has been developed from the original code CHAMPION 2/E/FIX in many ways. Now it is a general user program. Because the essential non-linearisties of the equations of fluid dynamics demand interations in order to get convergent results, besides under relaxation factors, false time steps have been used in CHAMPION SGE. This device slows down other parts of the calculations, particularly those which change the coefficients in the linear equation components. For example, the lower false time step values can be set to H or v for which the flow driven by buoyancy are particularly prone to divergent behaviour, resulting from the interactions between the enthalpy and velocity component v.

Because the values of dependent variables may wander during their approach to a converged solution, the CHAMPION SGE is also equipped with certain limiting-value facilities: specific values can be prescribed to upper and lower limits of u, v, k, ε, H, C, etc, so that the relevant values cannot even transitorily overstep the bounds of physical realism. The final solution will not be affected, but the danger of divergence resulting from a poor initial guess (supposed) will be greatly reduced.

The wall functions for enthalpy H, which are given in Eq.(4) and (5) give too small values of heat exchange coefficient (normally 1-3 W/m^2K). They are too low by comparing with measurements (normally 3-5W/m^2K). But the wall functions for enthalpy used in PHOENICS [15] gave much smaller values than those from CHAMPION SGE. In all computations above, the heat exchange coefficient were set by measured ones instead of computed ones.

For turbulent wall functions the near wall cell distance is determined in such a way that it must lie in the turbulent part of the boundary layer (y+ between 30 and 100) [16].

The first approximated method, which treats the heat transfer through the side wall in the Z-direction as thermal sources or sinks for the finite-domain equations, gives very good predictions on temperature and velocity distributions of the room. As mentioned in [9], 3D PHOENICS computations are very expensive, 40 minutes CPU time on an IBM 3083 computer is needed for a 3D steady computation with a grid number 9*18*27. The 2D CHAMPION SGE code combined with the methods above only costs 2 minutes CPU time for a grid number 18*27. 2D computations are much cheaper.

The second and the third approximated methods are very useful to get general information, such as velocity patterns, average temperature and its gradients. If only a limited area is considered, such as the variable values near the inlet area, the numerical predictions will be not satisfactory. According to authors' experience, the proportion of W/w_{in}

bigger than 5.0 will cause too big discrepancies between computations and measurements. In this case, 3D computations are required.

When there is a temperature gradient in a room, such as in industrial halls and theatres, normal one-air-point cooling load code will not give good predictions on cooling/heating load. If flow programs are combined with those cooling load programs, better agreements will be obtained. The combination can be done in following steps:

(1) Calculate wall temperatures and cooling/heating load by a cooling load code;
(2) Use the results from step (1) as boundary conditions for flow codes to calculate the convective heat exchange coefficients and temperature distributions and air movements in the room;
(3) Re-update the wall temperatures and cooling/heating load via the cooling load code by using the convective heat exchange coefficients and temperature distributions/air patterns from step (2);
(4) Go to step (2) for next time step computations, etc.

A computer code combined flow codes CHAMPION SGE/PHOENICS with cooling load code has been built at the moment by the authors. Primary computations on the computer code show much better agreement between measurements and computations [14].

6. REFERENCES

[1] Gosman A.D. and Pun W.M., The TEACH Program, HTS course notes, Imperial College, London, 1974
[2] Spalding D.B., GENMIX - A general computer program for two-dimensional parabolic phenomena, Pergamon Press, 1977
[3] Pun W.M. and Spalding D.B., A general computer program for two-dimensional elliptic flows, HTS/76/2, Imperial College, London, 1976
[4] Rosten, H.I. and Spalding, D.B., The mathematical basis of the PHOENICS EARTH computer code, CHAM TR/58b/, 1981
[5] Nielson, P.V., Restivo, A. and Whitelaw, J.H., Buoyancy-affected flows in ventilated rooms, Numerical Heat Transfer, vol 2, 115-127, 1979
[6] Jedrzejewska-Scibak, T. and Lipinski, D.M., Numerical prediction of air flow in industrial halls, CLIMA 2000, vol 4, pp377-381, Copenhagen, 1985
[7] Hjertager, B.H. and Magnussen, B.F., Numerical prediction of three dimensional turbulent flow in a ventilated room, In: Spalding, D.B. and Afgan, N., Heat transfer and turbulent buoyant convection, Washington, Hemisphere publ. cop. 1977
[8] Schmitz, R.M. and Renz, U., Berechnung turbulenter Raumluftstromungen bei gekoppeltem Impuls-, Warme- und Stoffaustausch, CLIMA 2000, vol.4, pp371-376, Copenhagen, 1985
[9] Chen Qingyan and Kooi, J. van der, Measurements and computations on air movement and temperature distribution in a climate room, XVIIth International Congress of Refrigeration, Vienna, 1987
[10] Launder, B.E. and Spalding, D.B., The numerical computation of turbulent flows, Computer methods in apllied mechanics and energy, 3, 269-289, 1974

[11] Leur, P.H.E van de, Numerieke berekeningen van de turbulente gestratificeerde luchtstroming in een gang, Afstudeerverslag, Delft University of Technology, 1983
[12] Chen Qingyan, The mathematical foundation of the CHAMPION SGE computer code, Report No. K-118, Delft University of Technology, 1986
[13] Alamdari F.G. and Hammond G.P., Improved data correlatons for buoyancy-driven convection in rooms, Building Services Energy R & D, vol.4, No.3, pp106-112, 1983
[14] Kooi, J. van der and Chen Qingyan, Improvement of cooling load programs by combination with an air flow program, XVIIth International Congress of Refrigeration, Vienna, 1987
[15] Gunton M.C., Rsten H.I., Spalding D.B and Tatchell D.B., PHOENICS: An instruction manual, CHAM TR/75/, London, 1983
[16] Aldham, C., Private Communication, CHAM, London,June 1986