International Energy Agency

Energy Conservation in Buildings
and Community Systems Programme

Annex 20 Air Flow Patterns within Buildings

Room Air and Contaminant Flow, Evaluation of Computational Methods

Subtask-1 Summary Report

TNO Building and Construction Research
P.O. Box 29, 2600 AA Delft, The Netherlands

Room Air and Contaminant Flow, Evaluation of Computational Methods

Subtask-1 Summary Report

Editor:
A.D. Lemaire

Contributors:
Q. Chen
M. Ewert
J. Heikkinen
C. Inard
A. Moser
P.V. Nielsen
G. Whittle

December 1993

This report documents results of cooperative work performed under the IEA Programme for Energy Conservation in Buildings and Community Systems, Annex 20: Air Flow Patterns within Buildings.

3.2.1 Two-dimensional test cases

The two-dimensional test case is only simulated, but its results can be compared with available experimental data. The test case represents both isothermal flow at a Reynolds number of 5,000 (2D1), and summer cooling at a range of Archimedes number (2D2). The configuration is shown in figure 3.1. The room is specified by ratios of L/H= 3, h/H= 0.056, t/H= 0.16, where 'L' is the room length, 'h' the inlet slot height, 't' the exhaust height and H the room height (3.0m).

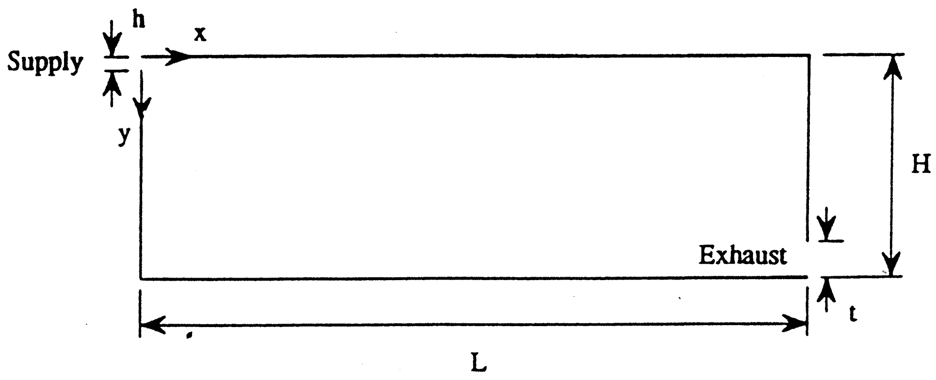

FIGURE 3.1 2D Test case configuration

Test case 2D1: isothermal

At the inlet the Reynolds number is 5,000 and the turbulence intensity 4%. For a real room with height 3.0m this corresponds with inlet velocity u_o = 0.455m/s and inlet temperature T_o = 20°C. The test case is extended with transport of contaminants with a uniform mass flux of contaminants along the floor. Experimental data for Reynolds number 5,000 has been reported previously [69]. The aim is to compare the simulated results with this data. In particular profiles on two vertical lines x = H and x = 2H and on two horizontal lines y = h/2 and y = H - h/2.

Test case 2D2: non-isothermal

The aim of this test case is to predict flow with a strong buoyant effect. A constant heat flux is added along the floor. The critical factor is the influence of the Archimedes number on jet penetration. The simulations are repeated for increasing Archimedes number (= increasing heat flux) until the CFD code predicts a flow with a reduced penetration depth x_s (see section 2.3.1).

The Archimedes number is defined as:

$$Ar = g\,\theta\,h / T\,u_o^2 \qquad (3.1)$$

where h = inlet slot height (m), g = gravitational acceleration (m/s²), u_o = inlet velocity (m/s), T = temperature level (K), θ = temperature difference between exhaust and inlet (°C).

The penetration depth x_s depends in some cases on the initial conditions. Different values of x_s can be obtained by increasing or decreasing the Archimedes number until the same experimental conditions are reached. Each participant should predict the penetration depth as a function of Archimedes number. The maximum velocity u_{rm} in the occupied zone can also be given as a function of the Archimedes number. The reduction of x_{Te} is expected to occur for Ar between 0.2 and 0.12. For room height H = 3.0m the Archimedes number Ar = 0.02 corresponds to θ = 0.74°C for u_o = 0.455m/s and T = 20°C.

3.3.2 *Two-dimensional test cases*

Very detailed computations are possible for this particular test case, and useful data has been generated. A survey of performed simulations is given in table 3.3

TABLE 3.3 Test case references: two-dimensional cases 2D1 and 2D2

Ref.	Case	Investigator	Code	Diff. scheme	Grid XxYxZ	Additional profiles	High or low Re	Full or half room
[79] C	D1D2	Said	EXACT3	HDS	37x34x15	-	high	half
[80] CH	D1D2	Chen	PHOENIX-84	UDS		temp.	low	-
[81] D2	D1D2	Vogl et al.	FLUENT	PLDS/ QUICK	56x62x1	-	high	-
[77] DK	D1	Skovgaard et al.	TEAM	PLDS	38x78x1	-	low	-
[83] NL	D1D2	Lemaire	WISH3D	UDS	36x30x1	conc.	high	-
[84] SF	D1D2	Heikkinen et al.	FLUENT/ WISH3D	PLDS/ QUICK	45x26x1	conc.	high	-

Test case 2D1: isothermal

Vogl et al.[81] Figure 3.3 shows predicted velocity field u/u_0 and distribution of turbulent intensity $\sqrt{u'^2}/u_0$ which agree well with others. A comparison is shown with predictions from Skovgaard et al.(fig.3.4) with a low-Reynolds-model. Figure 3.5 shows comparisons at section X/H = 1.0 of power law and QUICK differencing, with simulations by Chen and with measurements. The general trends of velocity and turbulent intensity are represented reasonably well by all simulation approaches but some discrepancies exist in certain areas. In general, the simulations by Vogl and Renz, along with most others, do not predict recirculation in the corners, and under-predict turbulence levels particularly near the floor.

Heikkinen et al. [84]. Results with WISH3D and FLUENT with PLDS-scheme show that the flow pattern is well predicted apart from the lack of recirculation in the upper corner at the wall containing the exhaust. A good correspondence of velocity decay and velocity fluctuation is obtained up to X/H = 2.0, beyond which the predicted velocity decay is more rapid. Figure 3.6 shows a comparisons of velocity profile at X/H = 2.0 between WISH, FLUENT (PLDS and QUICK) and measurement. The maximum velocity near the floor occurs at the same position in x-direction as measurements indicated, but the value is 8% lower. The velocity fluctuation is less well predicted near the floor.

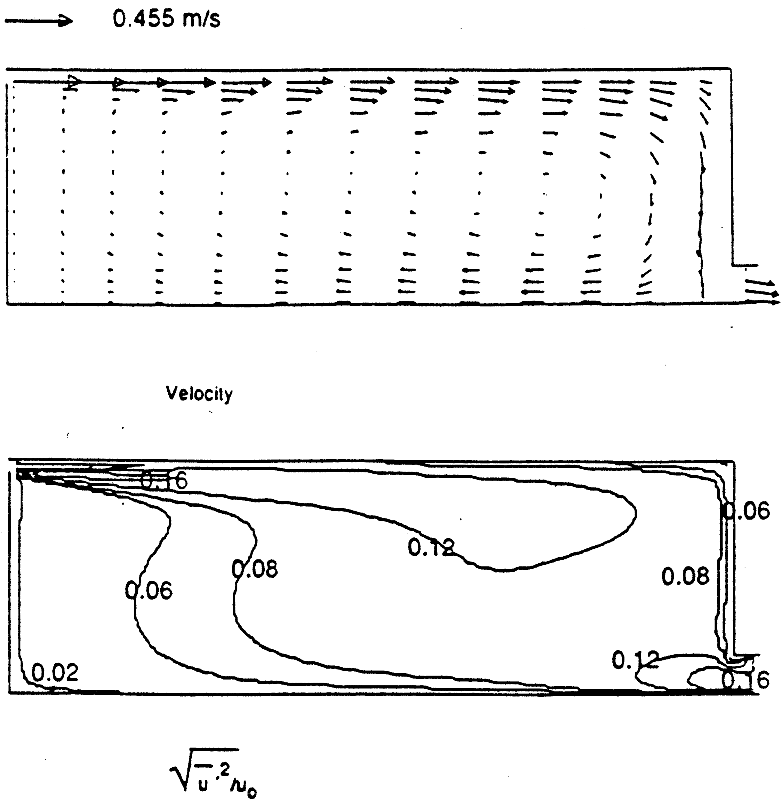

FIGURE 3.3 Test case 2D1: Vogl et al. (high Reynolds model)
Velocity field u/u_o and turbulent intensity $\sqrt{u'^2}/u_o$ (from $\sqrt{k} = 1.1\ \sqrt{u'^2}$)

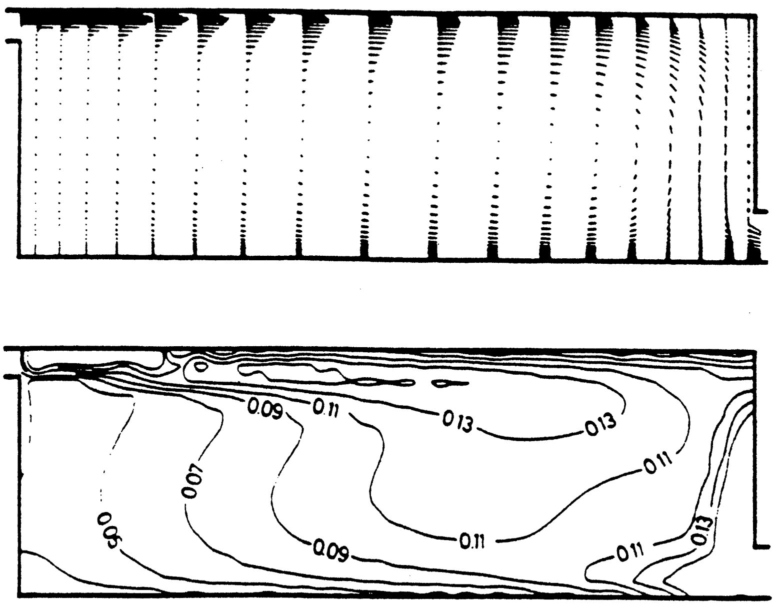

FIGURE 3.4 Test case 2D1: Skovgaard et al. (low Reynolds model)
Velocity field u/u_o and turbulent intensity $\sqrt{-u'^2}/u_o$ (from $\sqrt{k} = 1.1\ \sqrt{u'^2}$)

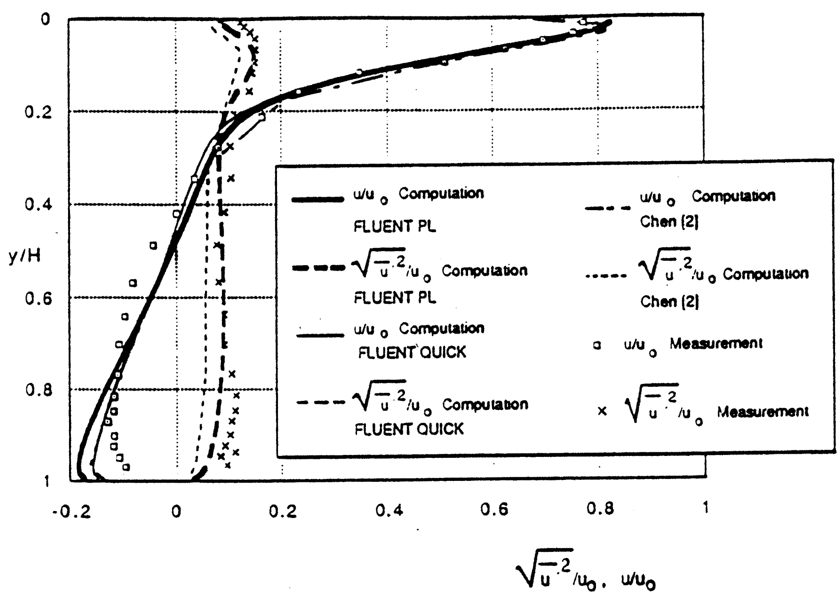

FIGURE 3.5
Test case 2D1: Vogl et al. Comparison between the computed (FLUENT QUICK, FLUENT PLDS, Chen) and measured (Nielsen) mean velocity and turbulent intensity in section x/H = 1.0

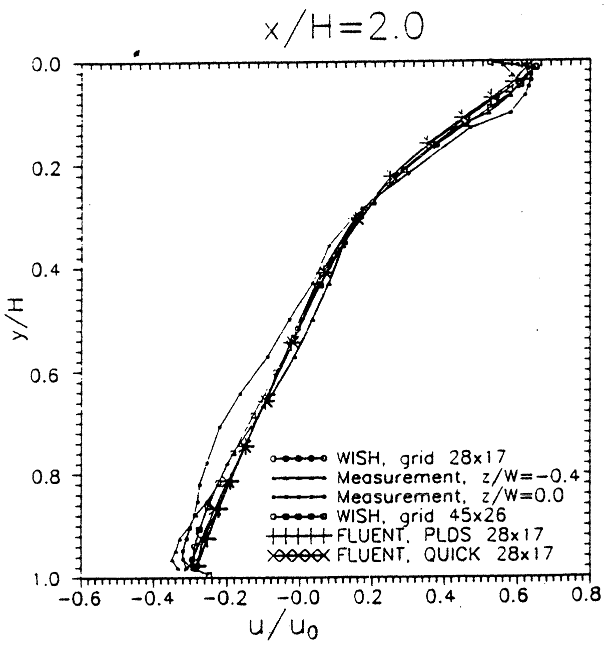

FIGURE 3.6
Test case 2D1: Heikkinen et al.
Comparison between computed and measured mean velocity in section x/H = 2.0

The best result for velocity near the floor level was found using QUICK on a coarse grid where the velocity was within 6% of measurement. Finer grids resulted in an over-estimate of wall friction due to the wall function used.

Lemaire [83]. The prediction of velocity decay corresponded quite well with measurements except that the measured recirculation in the corners was not predicted (fig. 3.7). The turbulent fluctuation near the floor was, as with Heikkinen et al., under-predicted. The comparison of predicted and measured concentration in the isothermal flow was good (fig. 3.9).

Skovgaard et al. [77] used a low Reynolds number k-ε turbulence model (fig. 3.8). It is stated that the low Reynolds number model demanded a fine grid be used in the inlet because of its location directly beneath the ceiling. Comparison of velocity and turbulence quantities are made with LDA measurements obtained in a scale model and with other simulations. At sections X/H = 1.0 and 2.0 the agreement with the measured velocity and turbulence levels is good. Generally, the velocity decay in the jet is slightly faster than the measurements suggest and hence the growth in the jet width is over-predicted. An important observation is that a small

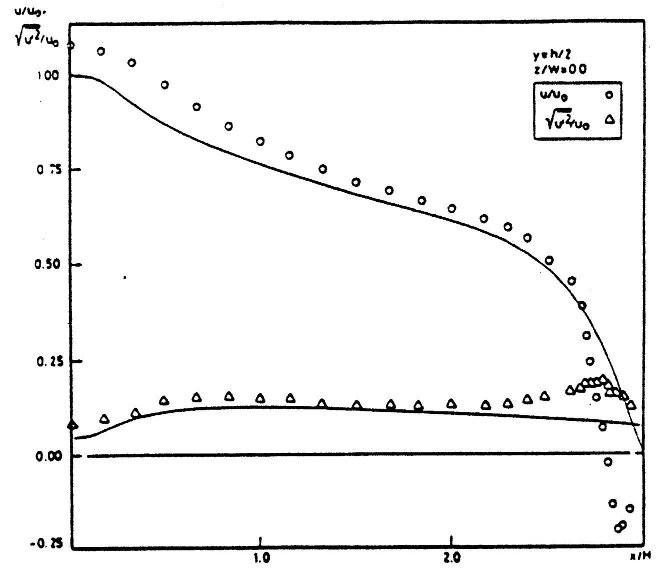

FIGURE 3.7
Test case 2D1: Lemaire
Comparison between computed and measured mean velocity and turbulent intensity in section y = h/2

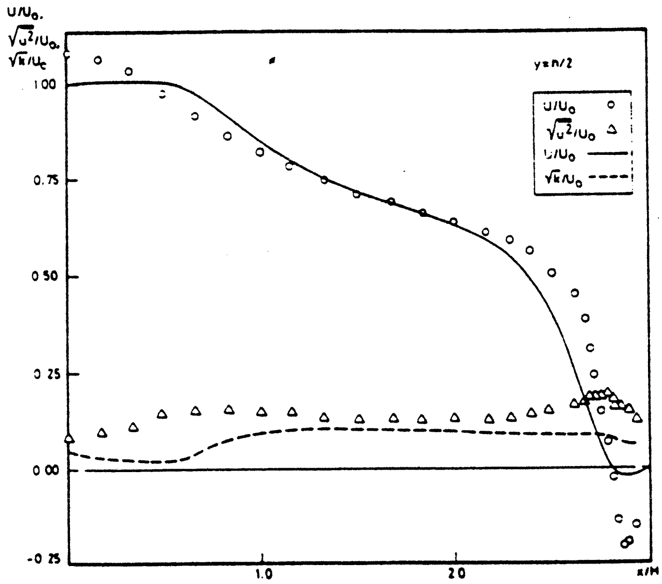

FIGURE 3.8
Test case 2D1:
Skovgaard et al.
Comparison between computed and measured mean velocity and turbulent intensity in section y = h/2

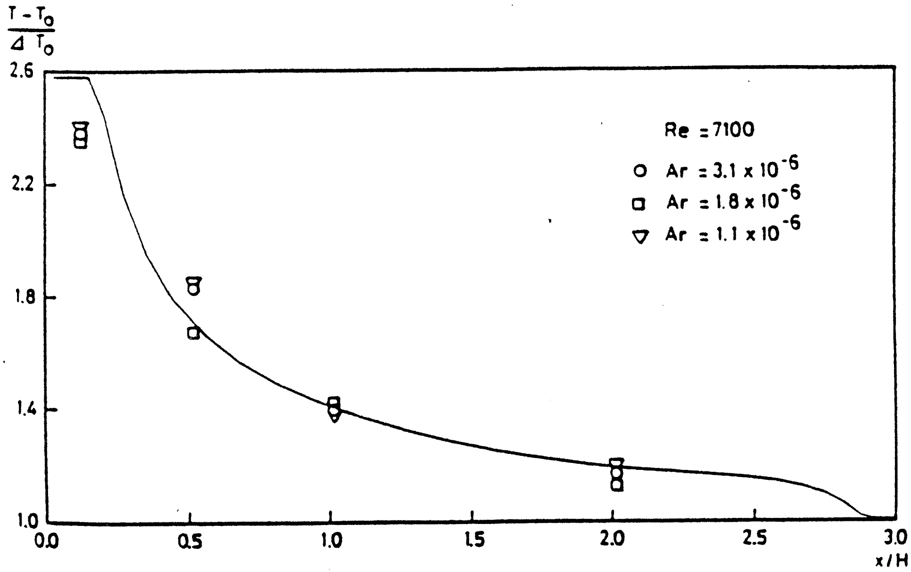

FIGURE 3.9
Test case 2D1: Lemaire
Comparison between computed and measured normalised concentrations in section y/H = 0.75

recirculation is predicted at the corner of the room opposite the inlet, although the magnitude is very much smaller than the measured values. Recirculation at the opposite corner near the floor was not predicted. Some comparisons of other simulation results with measurement indicate that a one-equation turbulence model under-predicts the velocity in the wall jet beneath the ceiling. Other codes compared are a TEACH derivative and a vorticity-streamfunction code.

Chen [80] used a low Reynolds number turbulence model. In the isothermal case, results appear similar to those of others, the main features being that the velocity and turbulence trends are well represented but the corner recirculation are not predicted and turbulence levels are under-predicted. A good correspondence between predicted and measured concentration was achieved. It was suggested that the small discrepancies were due to Reynolds number differences.

Said [79]. A three-dimensional grid of 37 x 34 x 15 (18870 cells) was used. The trends of velocity and turbulence intensity were reproduced quite well, but as with other simulations corner recirculations were not predicted, and turbulence levels were under-predicted.

Test case 2D2: non-isothermal

Chen [80]. No intermediate jet penetration length could be found. The critical Archimedes number at which the flow patterns changed was 0.143 (fig. 3.10). Measurements reported by Nielsen [68] indicated a critical Archimedes number of 0.02. However, Chen points out that the ratio of slot height to room height and Reynolds number used in the experiment were different to those specified in the simulations.

Heikkinen et al.[84] found that jet penetration length was equivalent to the room length at Archimedes number of 0.12 or less, and almost zero at Archimedes number of 0.16 or more. Intermediate jet penetrations were not found except during the course of iteration. It was stated as very important to ensure that the equations are well converged before accepting a solution. Good practice is to periodically inspect the solution during convergence, site the monitor location in an intelligent way and to inspect the traces of residual errors.

Lemaire [83] found that the predicted flow pattern was dependent on initial conditions. A hysteresis effect was evident. Again, as with the Heikkinen data, no intermediate penetration lengths were observed. Starting from uniform initial fields the Archimedes number at which the flow pattern changed was 0.173 to 0.175.

Vogl et al.[81]. The simulations confirm previous simulation results by predicting an absence of intermediate jet penetration length. The critical Archimedes number, which was 0.15 to 0.16, was found to be independent of starting conditions.

Said [79]. In the simulations, some three-dimensional effects are evident in the flow field plots which indicate a reduction in penetration length as the Archimedes number is increased. The highest Archimedes number modelled was 0.143 which correspond to the critical Archimedes number found by Chen. At this condition evidence of reverse flow exists at two-thirds distance along the room, although three-dimensional effects were strong making it is difficult to interpret the flow field (fig. 3.11). However, this is an important observation which needs further investigation through three-dimensional simulation.

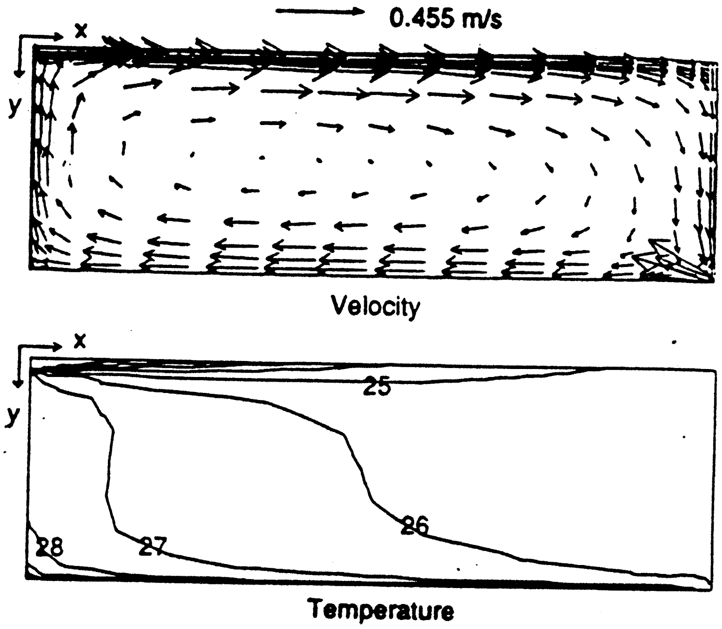

FIGURE 3.10a
Test case 2D2: Chen
Velocity and temperature distributions if Ar = 0.142

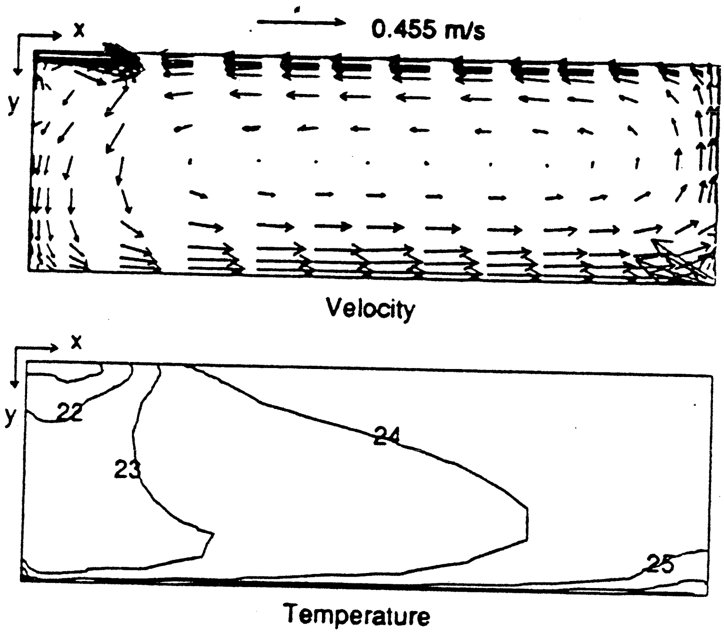

FIGURE 3.10b
Test case 2D2: Chen
Velocity and temperature distributions if Ar = 0.143

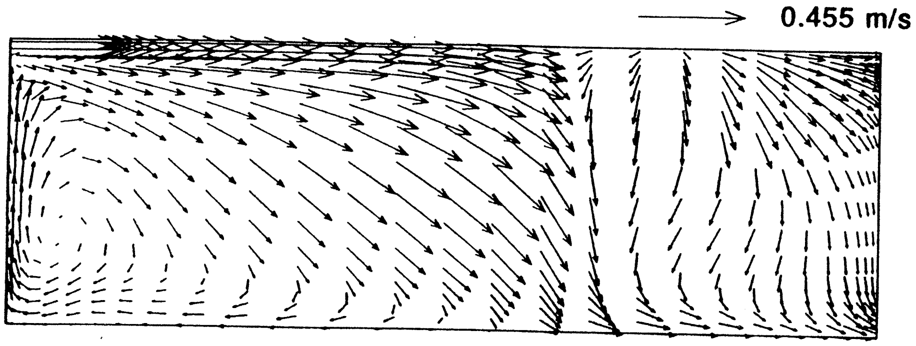

FIGURE 3.11 Test case 2D2 (3D-calc.): Said.
Velocity if Ar = 0.143.