International Energy Agency
Energy Conservation in buildings and community systems
Annex 20: Air Flow Patterns within Buildings
Subtask 1: Room air and contaminant flow

RESEARCH ITEM NO. 1.46NL

SIMULATION OF SIMPLE TEST CASES 2D1, 2D2

Date: May, 1991

Author: A.D. Lemaire
TNO- Building and Construction Research
P.O. Box 29
2600 AA Delft
THE NETHERLANDS

Report No: AN20.1-NL-90-TNO-TPD10
Report Type: Annex report
Distribution: Unlimited
Available: From the author

CONTENTS

1. INTRODUCTION

Scope. This paper presents the results of the numerical simulation of two testcases within the scope of the IEA Annex 20 work. The two testcases have been specified by Nielsen [1] and can be approximated by a two-dimensional geometry. There exist a number of measurements of the flow in the specified geometry. The geometry has also been used earlier to test computer codes and can therefore be standardized as a benchmark test for room air distribution models (Nielsen[1]).

Objectives. The two testcases provide a common basis for CFD-code inter-comparison under prescribed closely specified boundary conditions. They serve to evaluate the model's performance in predicting:

- isothermal two-dimensional flow for a given supply-jet Reynolds number (case 2D1);
- non-isothermal two-dimensional buoyant flow for a range of supply-jet Archimedes numbers (case2D2).

2. SIMULATION METHOD

Physical model. The time-averaged equations of conservation of mass , momentum and energy are solved with the computer code WISH3D. Air is considered an ideal gas and an incompressible fluid and the Boussinesq approximation is used to account for density variations. The turbulence is modelled with the well known k-ε model, using logarithmic wall functions near the wall (Launder, [2]). The following model constants were applied: σ_k = 1.0, σ_ϵ = 1.31, c_1 = 1.44, c_2 = 1.92, c_3 = 1.44, c_μ = 0.09. The transport of contaminants by convection and diffusion is based on the conservation law for chemical species. The molecular diffusion coefficient is replaced by an effective diffusion coefficient to account for turbulent mixing. An effective Schmidt number of 1 is assumed.

Computer code. The program WISH3D was developed by TNO/TPD (Lemaire [3]) and is based on the finite volume method with pressure relaxation (Patankar [4]). At present the program can be used as an engineering tool for a limited set of problems. New features will be added in the future.

Methodology. A staggered grid and a upwind difference scheme have been applied. A non-uniform grid was generated to produce a fine grid near the boundaries. A grid of 36 nodes in x-direction and 30 nodes in y-direction has been used. A converged solution was assumed if the absolute sum of the residuals was below 10^{-6} for each variable.

3. RESULTS

General conditions

Geometry. Figure 1 shows the geometry and system of coordinates. Air is supplied through the slit in the left wall and exhausted through the opening in the right wall. The dimensions are L/H = 3.0, h/H = 0.056 and t/H = 0.16. Using H = 3.0 m for the simulations gives h = 0.168 and t = 0.48 m.

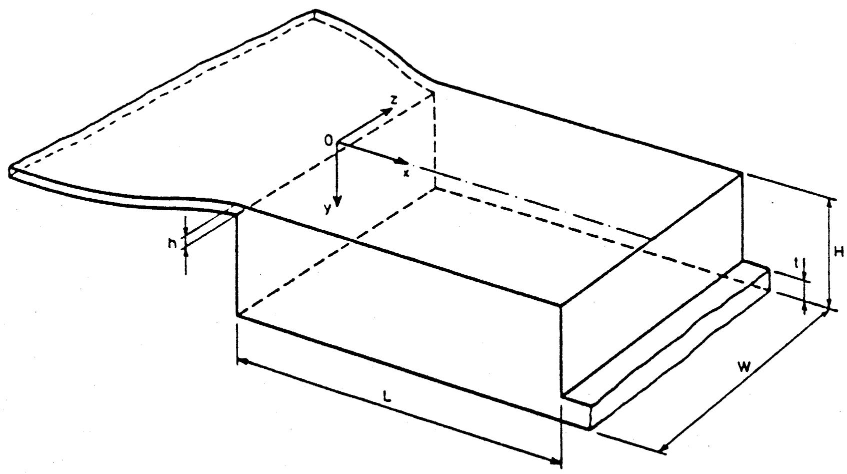


Figure 1. The geometry of the model and system of coordinates.

Supply conditions. Air of 20 °C is supplied. The supply velocity u_o is related to the Reynolds number Re_o defined by:

$$Re_o = \rho u_o h / \mu$$

with density ρ = 1.16 kg/m^3 and molecular dynamic viscosity μ= 1.17748 10^{-5} kg/(m.s). The Reynolds number is set at 5000, so inlet velocity u_o = 0.455 m/s. The inlet conditions for turbulent kinetic energy k_o and dissipation rate ϵ_o are given by:

$$k_o = 1.5(0.04.u_o)^2 \qquad \text{and} \qquad \epsilon_o = k_o^{1.5}/(h/10)$$

so $k_o = 4.97\ 10^{-4}\ m^2/s^2$ and $\epsilon_o = 6.59\ 10^{-4}\ m^2/s^3$.

Case2D1 (isothermal)

Fig 2 shows the velocity vectors and iso-contours of absolute velocity, turbulent intensity, and concentration. The massflux of contaminants is uniformly supplied through the floor. The concentrations are normalised with the mean concentration in the outlet. In figure 3 and 4 the computed profiles of mean velocity and rms velocity are compared with experimental data provided by Nielsen [1]. The rms velocity is calculated from the turbulent kinetic energy k using

$$k^{0.5} = 1.1\ u'^{2\ 0.5}$$

based on the assumption that $v'^2 = 0.6\ u'^2$ and $w'^2=0.8u'^2$ in a two-dimensional wall jet.

Figure 5 compares the measured and computed normalised concentrations.

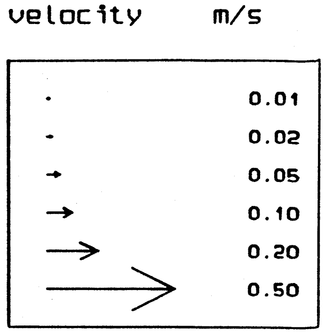
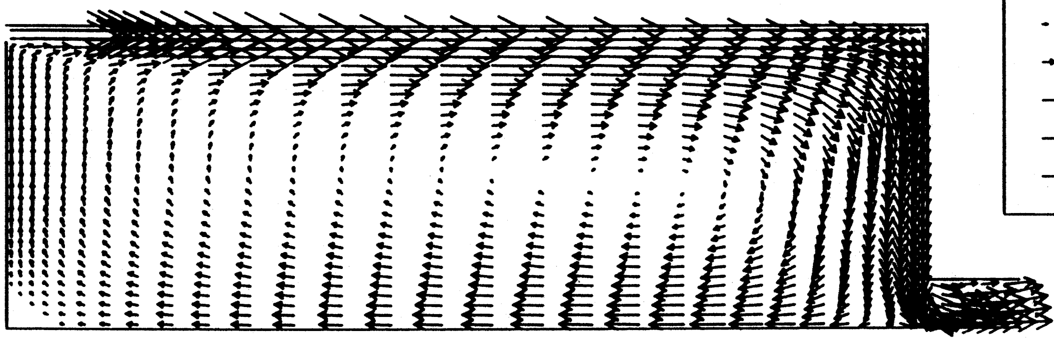
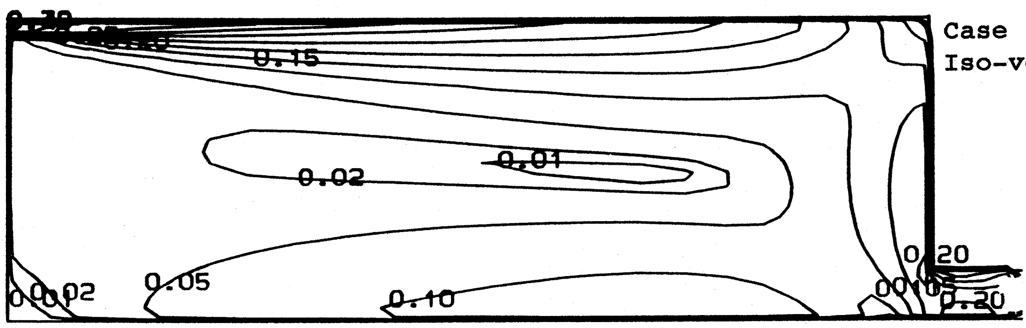


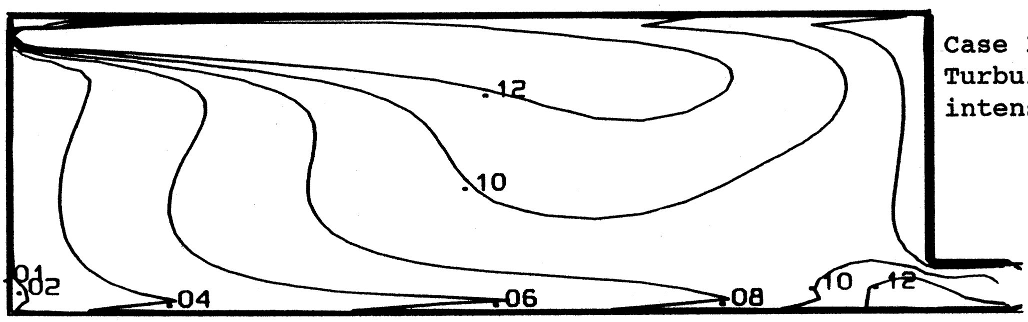
Figure 2. Case2d1: isothermal.



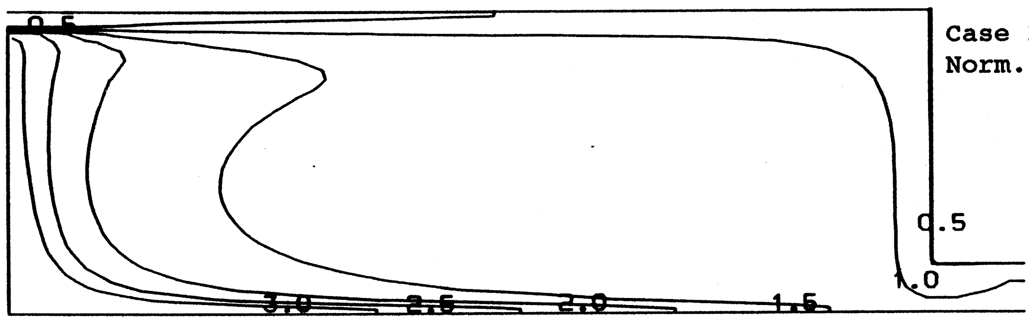
Case 2D1
Velocity vectors



Case 2D1
Iso-vels (m/s)



Case 2D1
Turbulent intensity



Case 2D1
Norm. Conc.

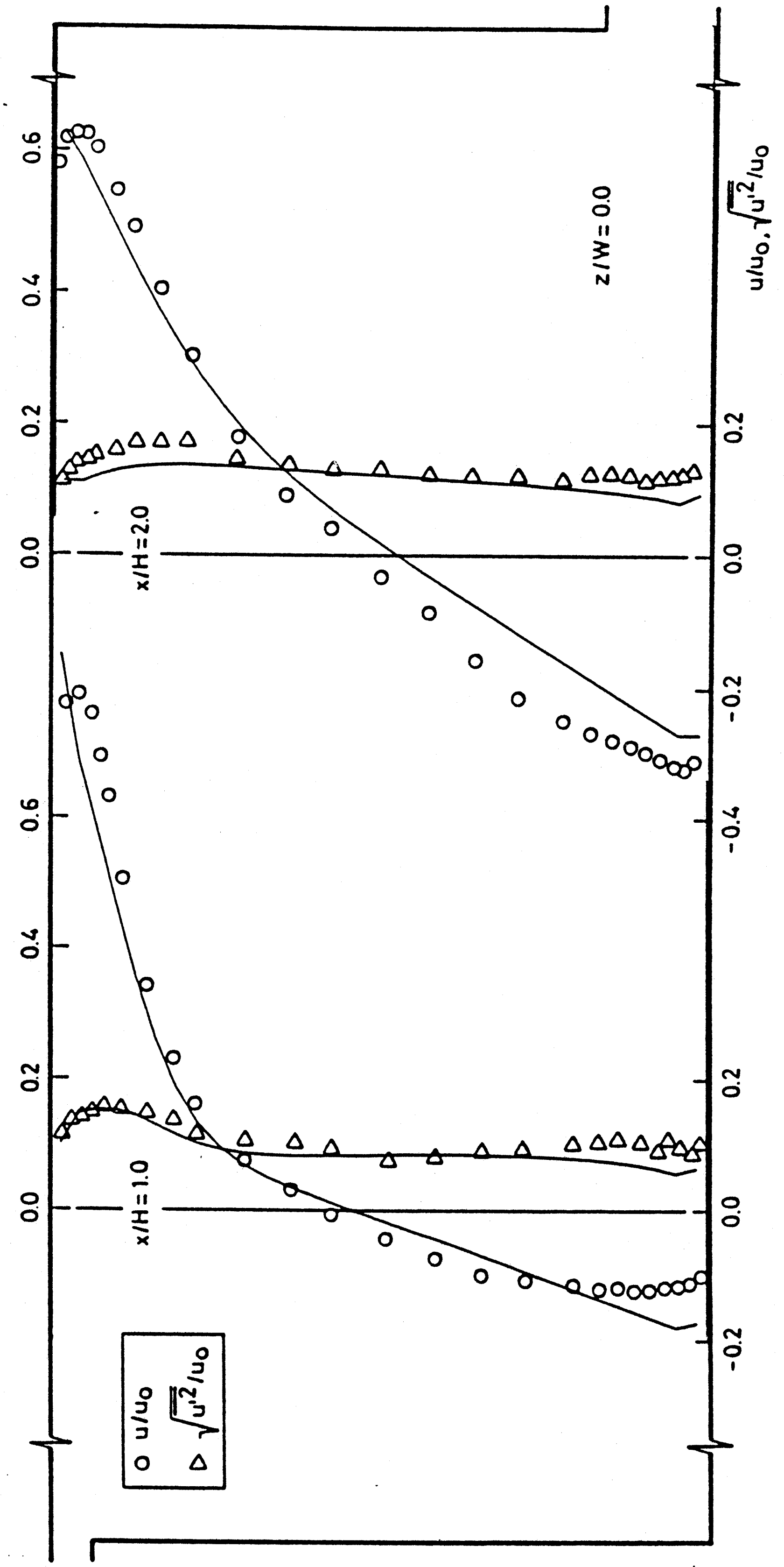


Figure 3. Comparison of measured and computed (solid-lines) mean and rms velocity.

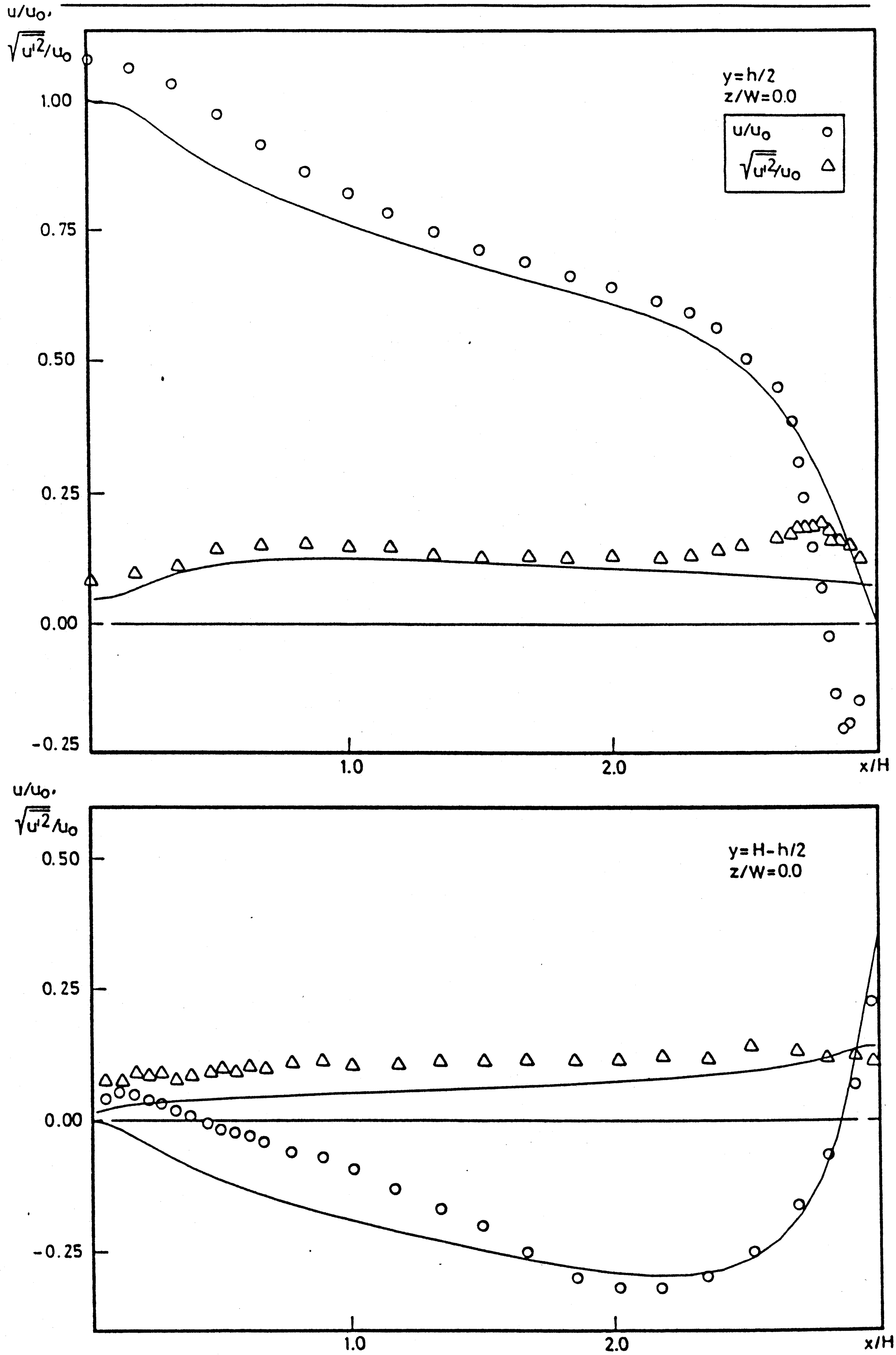


Figure 4. Comparison of measured and computed (solid-lines) mean and rms velocity near the ceiling and the floor.

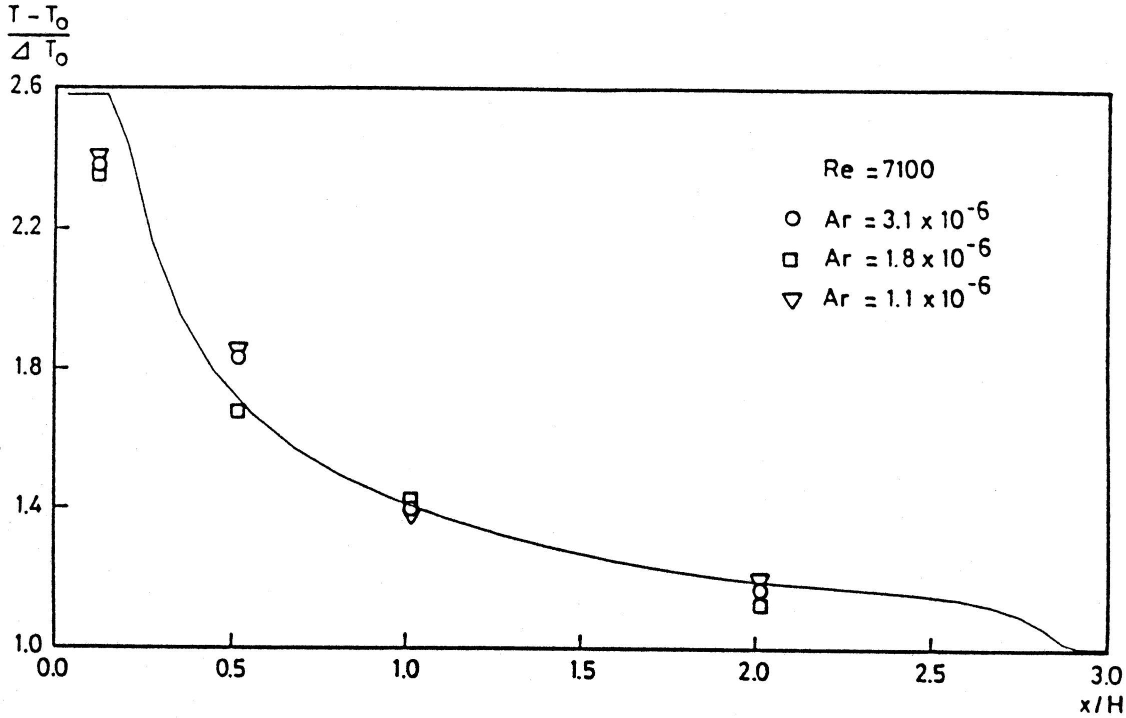


Figure 5. Comparison of measured and computed normalised concentrations at y/H = 0.75.

Case2d2 (non-isothermal)

The left and right wall and the ceiling are adiabatic. A constant heatflux is added along the floor and the Archimedes number is increased until a flow with a reduced penetration depth occurs. The Archimedes number Ar is defined by:

$$Ar = \beta g h \delta T_o \,/\, u_o^2$$

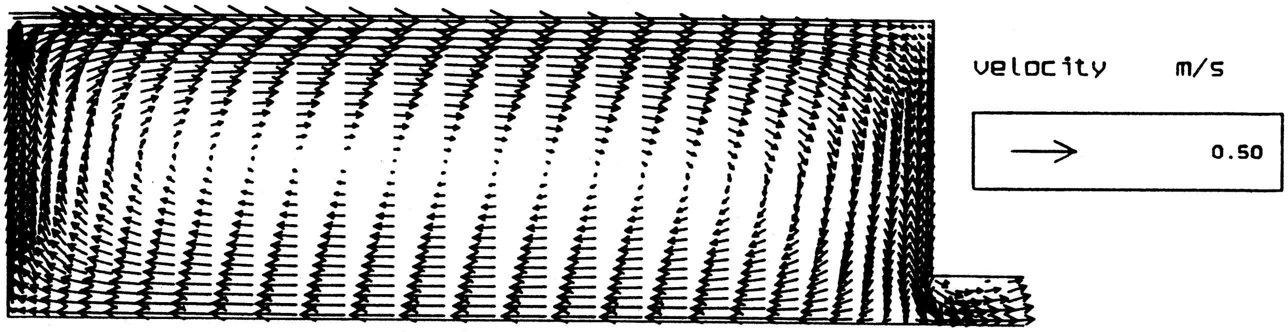
with coefficient of thermal expansion β = 3.395 10^{-3}, gravity g = 9.81 m/s^2 and δT_o temperature difference between supply and exhaust air

A number of runs were performed in order to predict the situation with a reduced penetration depth. A selection of the runs is presented in table 1. Figures 6 till 9 show the velocity vectors and iso-contours of absolute velocity, turbulent intensity, and temperature. 7.

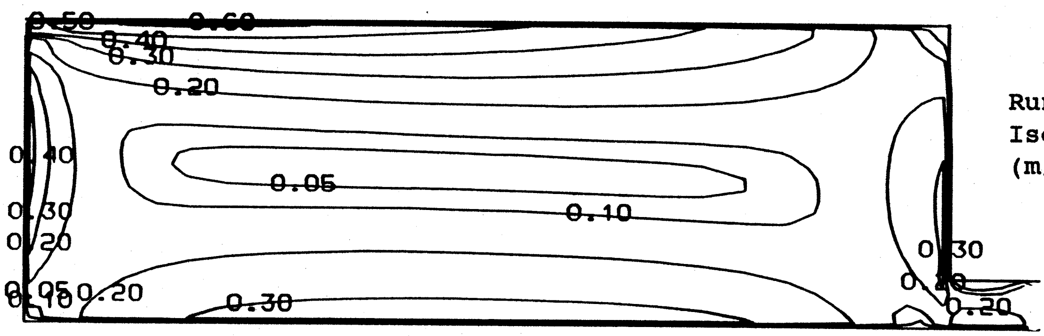
Two different types of initial fields have been used:

- uniform fields: velocities zero, temperature 20 °C, k = 10^{-6} m^2/s^2 and ε = 10^{-6} m^2/s^3.
- results from case2D1: isothermal forced convection.

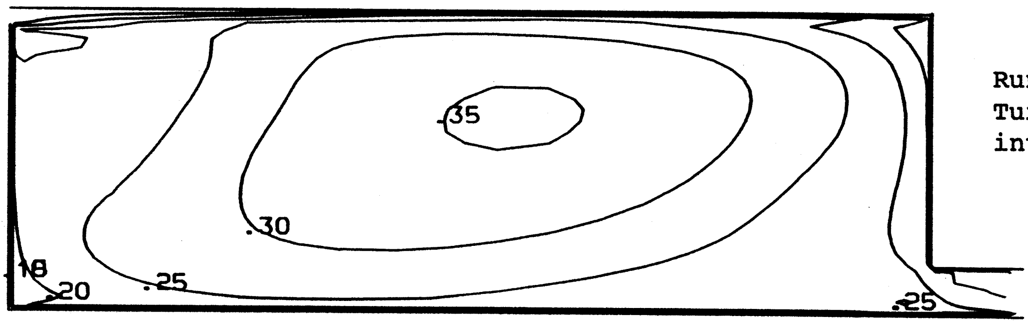
<u>Figure 6.</u> Case2D2 run 2D2U01NL: Ar = 0.173, from uniform fields.



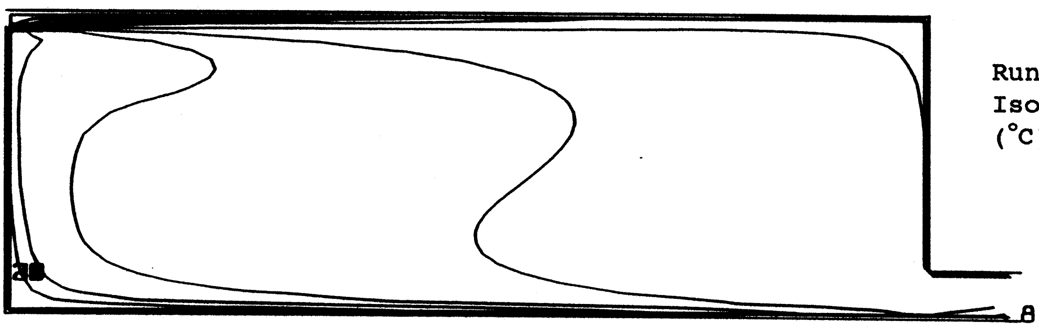
Run 2D2U01NL
Velocity vectors



Run 2D2U01NL
Iso-vels
(m/s)

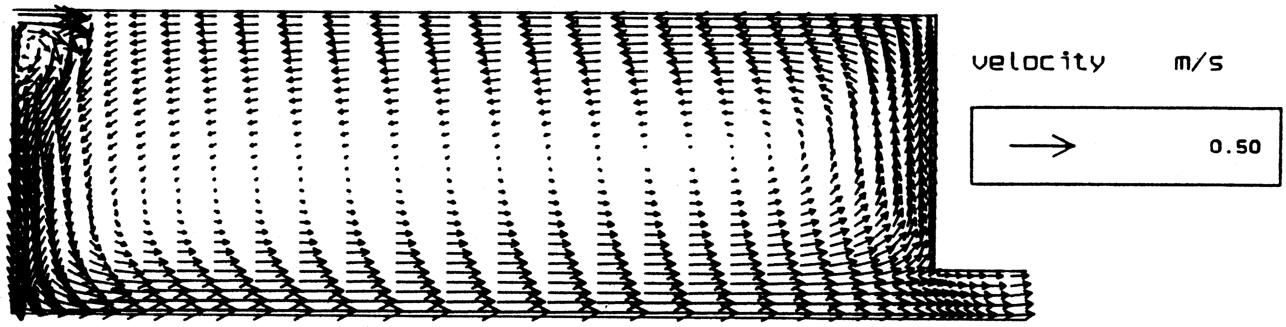


Run 2D2U01NL
Turbulent
intensity

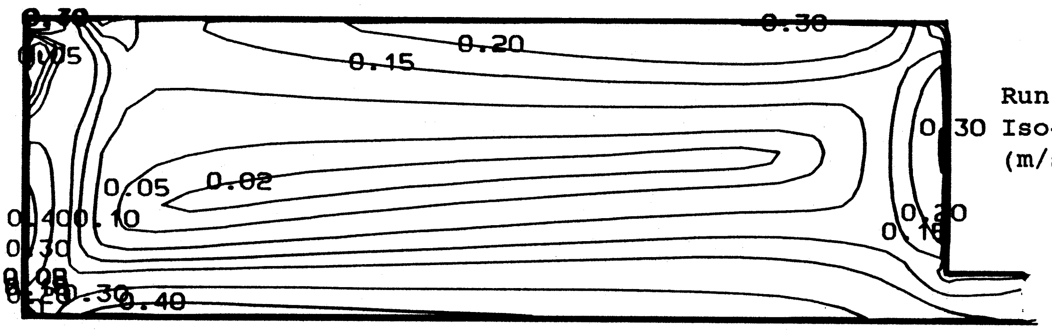


Run 2D2U01NL
Isotherms
(°C)

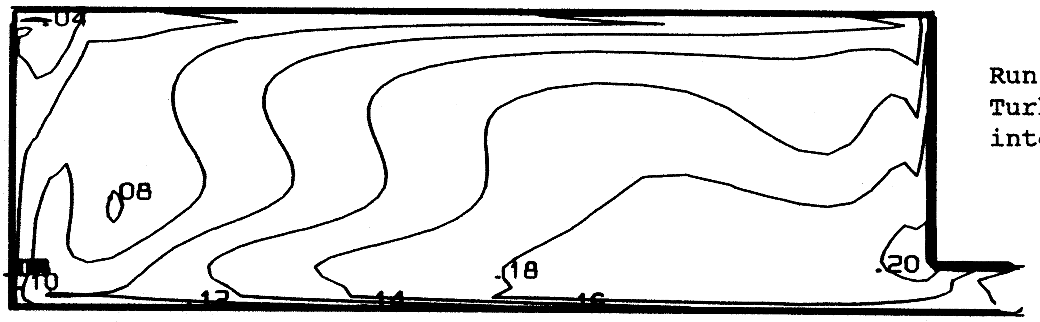
<u>Figure 7.</u> Case2D2 run 2D2U02NL: Ar = 0.175, from uniform fields.



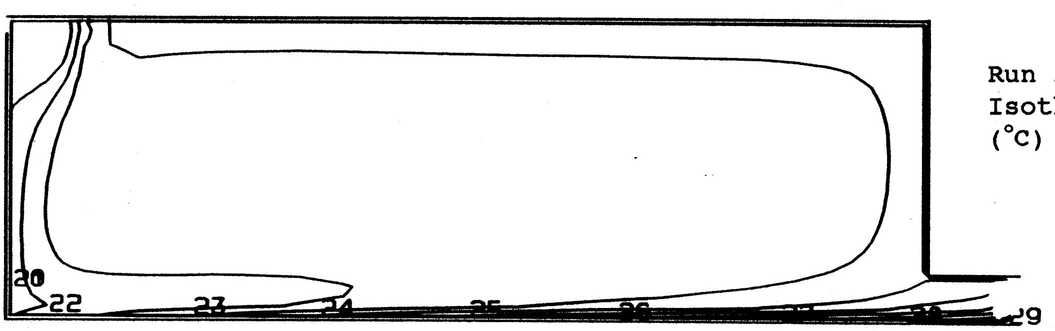
Run 2D2U02NL
Velocity vectors



Run 2D2U02NL
Iso-vels
(m/s)

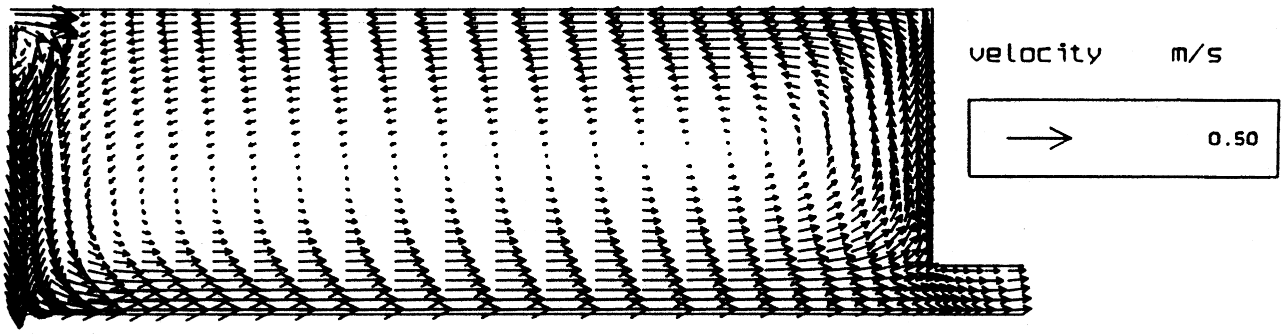


Run 2D2U02NL
Turbulent
intensity

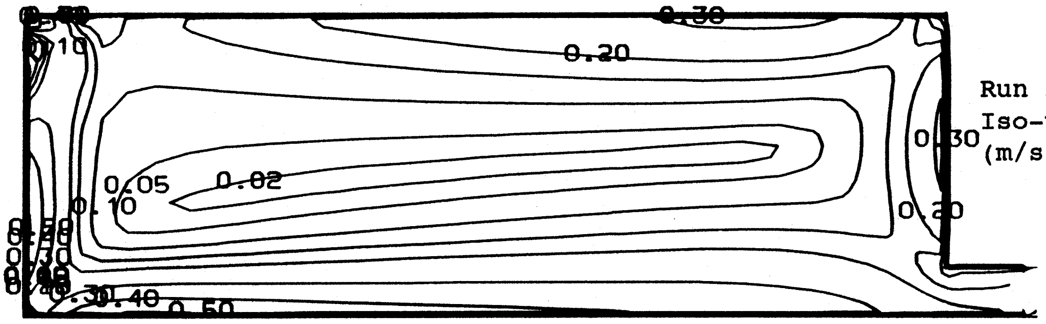


Run 2D2U02NL
Isotherms
(°C)

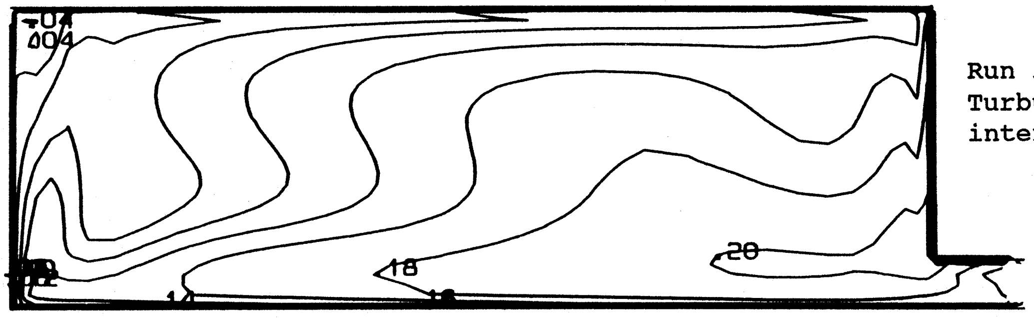
Figure 8. Case2D2 run 2D2U03NL: Ar = 0.20, from uniform fields.



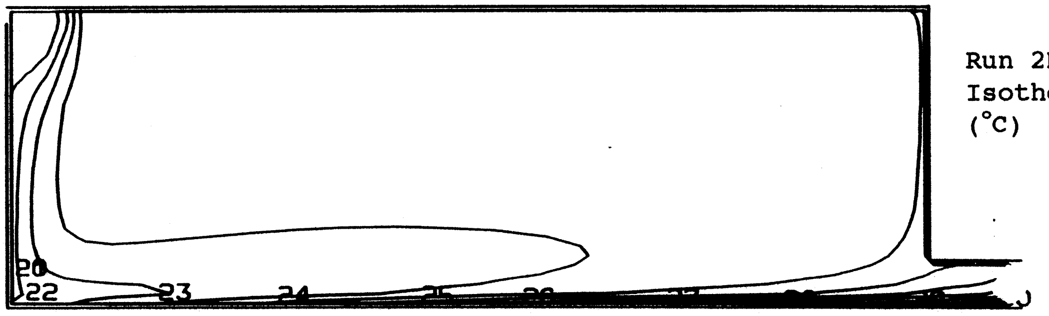
Run 2D2U03NL
Velocity vectors



Run 2D2U03NL
Iso-vels
(m/s)

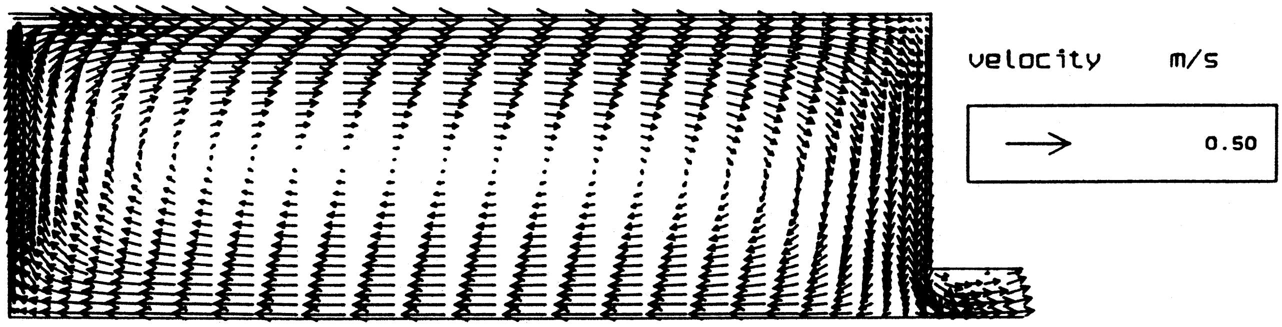


Run 2D2U03NL
Turbulent
intensity

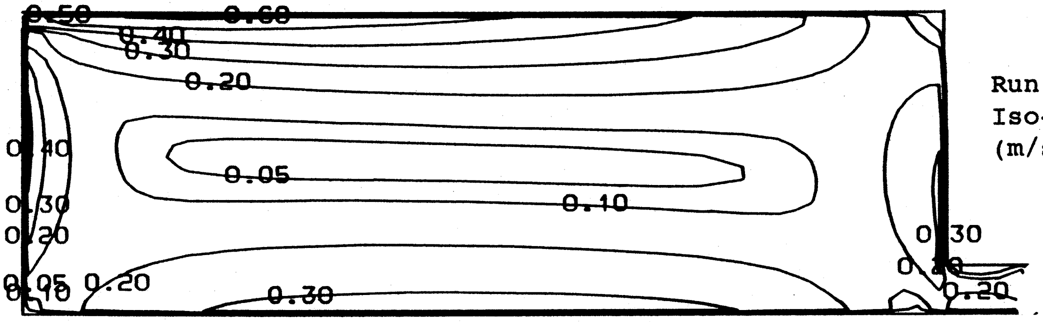


Run 2D2U03NL
Isotherms
(°C)

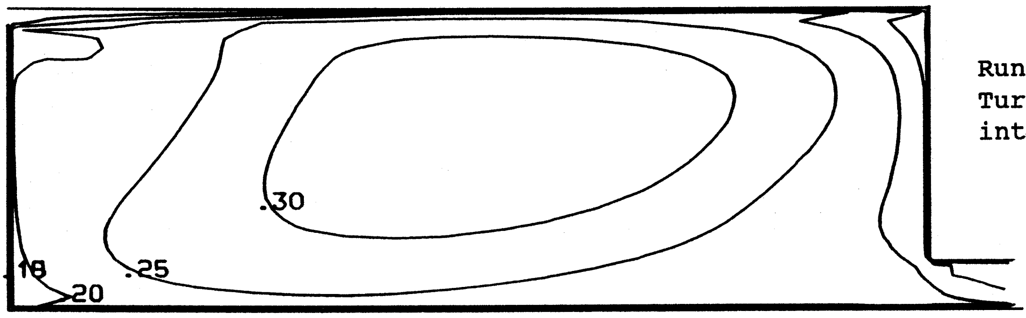
Figure 9. Case2D2 run 2D2F01NL: Ar = 0.20, from case 2D1 (isothermal).



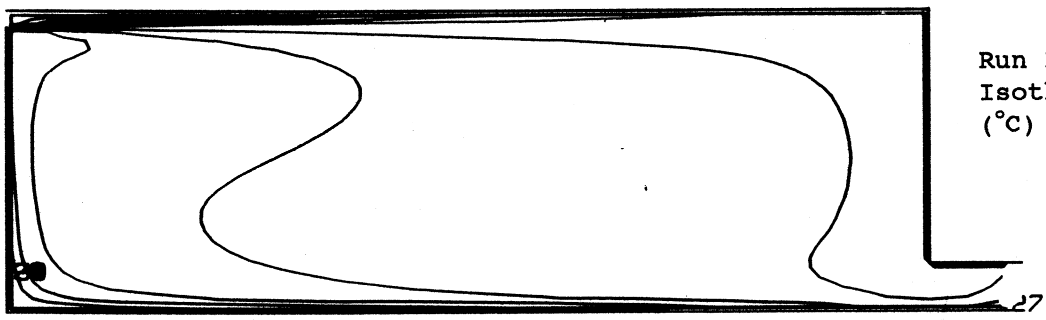
Run 2D2F01NL
Velocity vectors



Run 2D2F01NL
Iso-vels
(m/s)



Run 2D2F01NL
Turbulent
intensity



Run 2D2F01NL
Isotherms
(°C)

Table 1. Selection of simulated runs

RUN	Ar	heat flux (W/m^2)	initial fields
2D2U01NL	0.173	63.08	uniform
2D2U02NL	0.175	63.80	uniform
2D2U03NL	0.200	72.90	uniform
2D2F03NL	0.200	72.90	case2D1

4. CONCLUSIONS

Case2D1 (isothermal)

The computed velocity profiles agree reasonable with the available experimental data. The model fails however, to predict the counterflow near the end wall.

The computed turbulent intensities are lower then the measured values. As already suggested by Chen [5] overestimation of the velocity fluctuations in y and z-direction may be the reason.

The computed and measured concentration profile are in good agreement

Case 2D2 (non-isothermal).

As can be seen from figures 8 and 9 the flowpatterns depend on the initial fields. Starting from the fields of the isothermal case there is no reduction of penetration depth up until at least Ar=0.20. If uniform initial fields are used then the flow pattern changes completely between Ar = 0.173 and Ar = 0.175. For Ar = 0.173 the jet still reaches the endwall, whereas for Ar=0.175 the cold jet falls down almost immediately causing a reversal of the main vortex. Their seems to be no intermediate status. This phenomena has also been observed by Chen [5], but for Ar = 0.143. Both values are much higher as measured by Schwenke (Ar = 0.02) (see Nielsen[1]). The reason could be the different supply size and Reynolds number in the experiment (Chen[5]).

REFERENCES

[1] Nielsen, P.V., Specification of a Two-Dimensional Test Case, Research Item 1.45, IEA Annex 20 report, nov 1990

[2] Launder, B.E.and D.B. Spalding, The numerical computation of turbulent flows, Computer methods in mechanics and engineering, 3, pp 269-289, 1974

[3] Lemaire, A.D., User Manual WISH, TPD-TNO, Delft, 1990

[4] Patankar, S.V., Numerical heat transfer and fluid flow, Mc Graw Hill, New York

[5] Chen, Q., Simulation of simple test cases, IEA Annex 20 report, march 1991