International Energy Agency

Annex 20 "Air Flow Patterns Within Buildings"

Subtask 1 "Room Air and Contaminant Flow"

Research Item 1.46

Simulation of a Two-Dimensional Benchmark Test case

Author: M.N. Said
Institute for Research in Construction
National Research Council of Canada
Ottawa, Ontario, Canada K1A 0R6

Date:	March 1991
Report No:	AN20.1-CDN-90-IRC04
Report Type:	Annex report
Distribution:	Unlimited
Available:	The operating Agent or the author

Simulation of a Two-Dimensional Benchmark Test Case

This report presents the computational results for the two-dimensional benchmark test case described in Research Item # 1.45.

Code used: EXACT3

EXACT3 code

The EXACT3 [1]* code was developed at the National Institute of Standards and Technology (NIST), USA. The term EXACT stands for Explicit Time Marching Algorithm for Continuous Thermal Fluid Flow. EXACT3 is a three-dimensional finite difference code for simulating buoyant turbulent airflows within buildings. The code uses the high-Reynolds-number K-E turbulence closure. The buoyancy effect is accounted for by Boussinesq approximation. An explicit time marching technique is used to solve the set of non-linear coupled equations for momentum, energy, turbulent kinetic energy, and turbulent dissipation rate. EXACT3 uses a staggered grid system and an hybrid upwind/central combination scheme. A pressure relaxation method is used to satisfy the Poisson equation for mass conservation.

Velocity boundary values near solid surfaces are approximated by a power-law type velocity profile. The 1/7th power law was used in this study. The no-slip type boundary condition is used for the wall boundary condition for the turbulent kinetic energy. The wall boundary condition for the turbulent energy dissipation rate is derived from the logarithmic law condition.

The empirical coefficients of the turbulence model are given the widely used values:

$C_\mu = 0.09$, $C_1 = 1.22$, $C_2 = 1.92$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.0$, $\sigma_\theta = 0.9$

---

* numbers between brackets indicate references at the end of the report

Simulation Method

The specifications of the two-dimensional benchmark test case is described in Research Items # 1.45 [2]. The symmetry assumption was used and half the room domain (Figure 1) was simulated using a non-uniform grid of 37 x 34 x 15.

Air supply inlet conditions are:

Inlet opening height (h): 0.168 m
Inlet opening width (W): 3.0 m
supply airflow rate: 0.229 m^3/s
Inlet velocity horizontal component (U_o): 0.455 m/s
Inlet velocity vertical component (V_o): 0.0 m/s
Inlet turbulence intensity : 4%
Inlet turbulent kinetic energy (k_o): $1.5 \times (0.04 \times Uo)^2$
Inlet turbulent energy dissipation (E_o): $k_o^{1.5}/(0.1h)$

Reynolds No. (Re_o): 5000

where:

$$Re = U_o\ h\ /\ \nu$$

where: ν = air kinematic viscosity

The 1/7th power law was used to describe the Velocity boundary conditions near the solid surfaces. The simulation was conducted on an IBM main frame computer model 3090. The maximum error in the continuity was 5×10^{-4}. The simulation took @ 8 hours and 25,000 iterations.

Figures 2 to 8 show computation results for the isothermal case. In these figures, the turbulence intensity (rms) was computed as suggested by Nielsen [2] by:

$$\sqrt{\overline{u'^2}} = \sqrt{k}$$

Figures 5 to 8 compare computation results with measured data from Reference 2 for the isothermal case. Predicted data agree reasonably well with measurements.

Non-isothermal test case:

As suggested by Nielsen [2], the Archimedes number (as defined in Reference 2) was increased until a reduced penetration depth of the air supply jet took place. Computations were conducted using a Reynolds number of 5000, and Archimedes numbers (Ar) of 0.02, 0.06, 0.08, 0.1, 0.12, and 0.143. The later Archimedes number was selected based on Chen [3] results. A reduced penetration depth of the supply jet appears to occur at Ar = 0.143 (see Figures 9 to 14). There is considerable discrepancy (Figure 15) between predicted and measured temperature profile at y/H= 0.75.

References

1- Kurabuchi, T., Fang, J. B., and Grot, R.A.,"A Numerical Method for calculating Indoor Airflows Using a Turbulence Model", National Institute of Standards and Technology, Gaithersburg, MD 20899, USA, Report No. NISTIR 89-4211, January 1990.

2- Nielsen, P.V.,"Specification of a Two-Dimensional Test Case", Research item 1.45, November 1990, ISSN 0902-7513 R9040.

3- Chen, Q.,"Simulation of simple test cases", Annex report No. AN20.1-CH-90-ETHZ15, March, 1991.

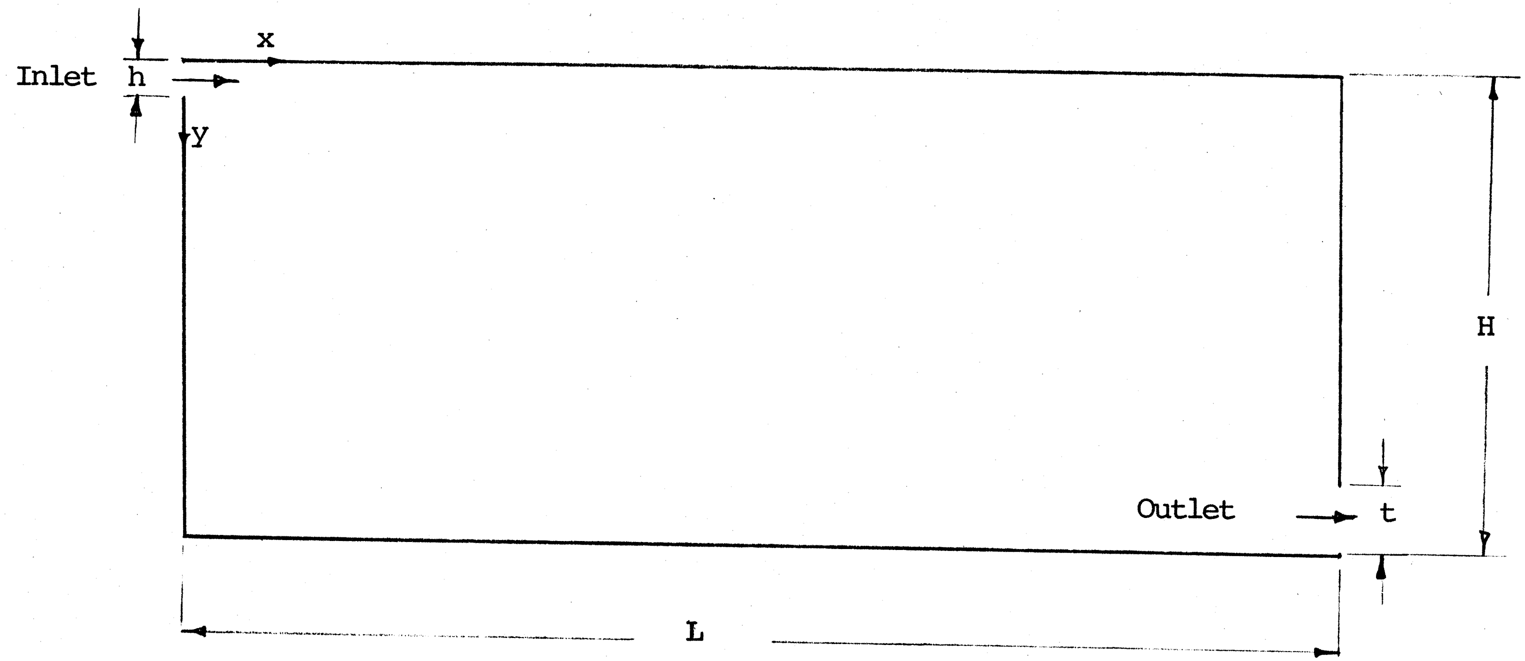


Figure 1. Geometry of the Two-Dimensional Benchmark Test Case.

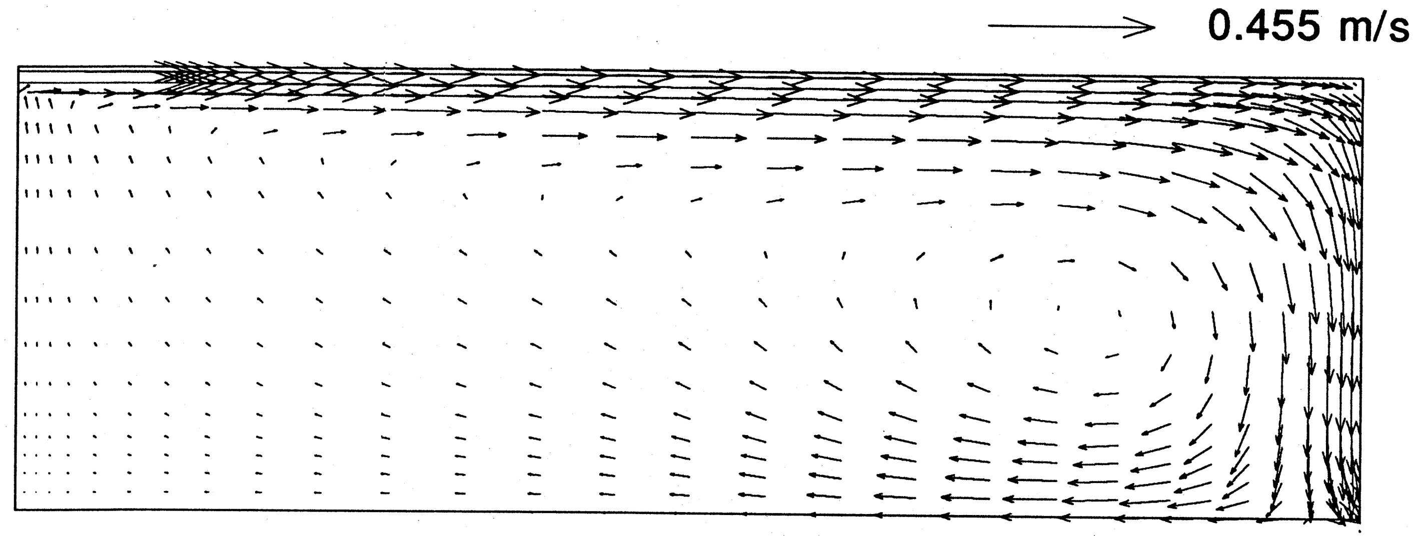


Figure 2. 2D Isothermal Forced Convection
Velocity (m/s) Vectors

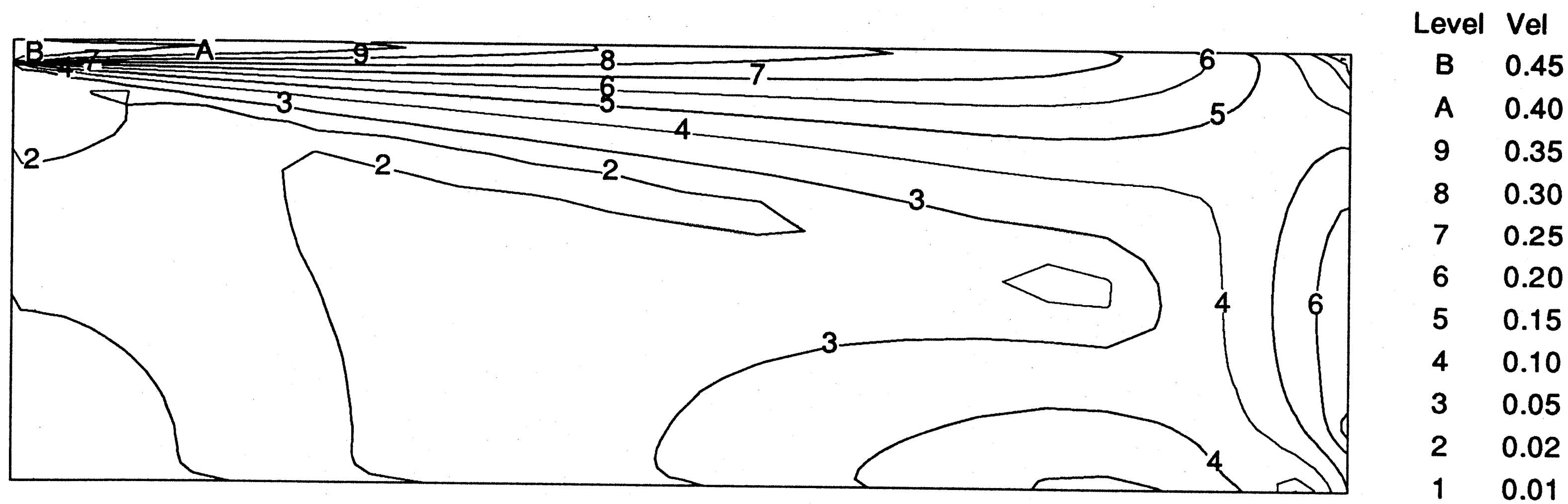


Figure 3. 2D Isothermal Forced Convection
Velocity (m/s) Contours

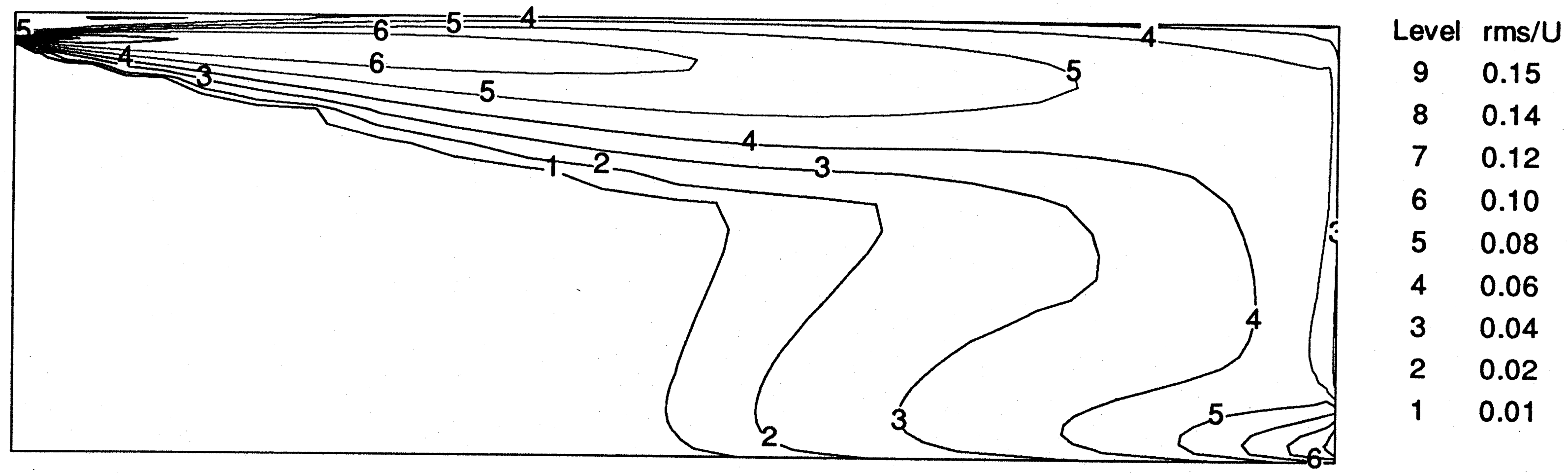


Figure 4. 2D Isothermal Forced Convection
rms velocity / Uo

Figure 5. 2D Isothermal Forced Convection

X/H = 1.0

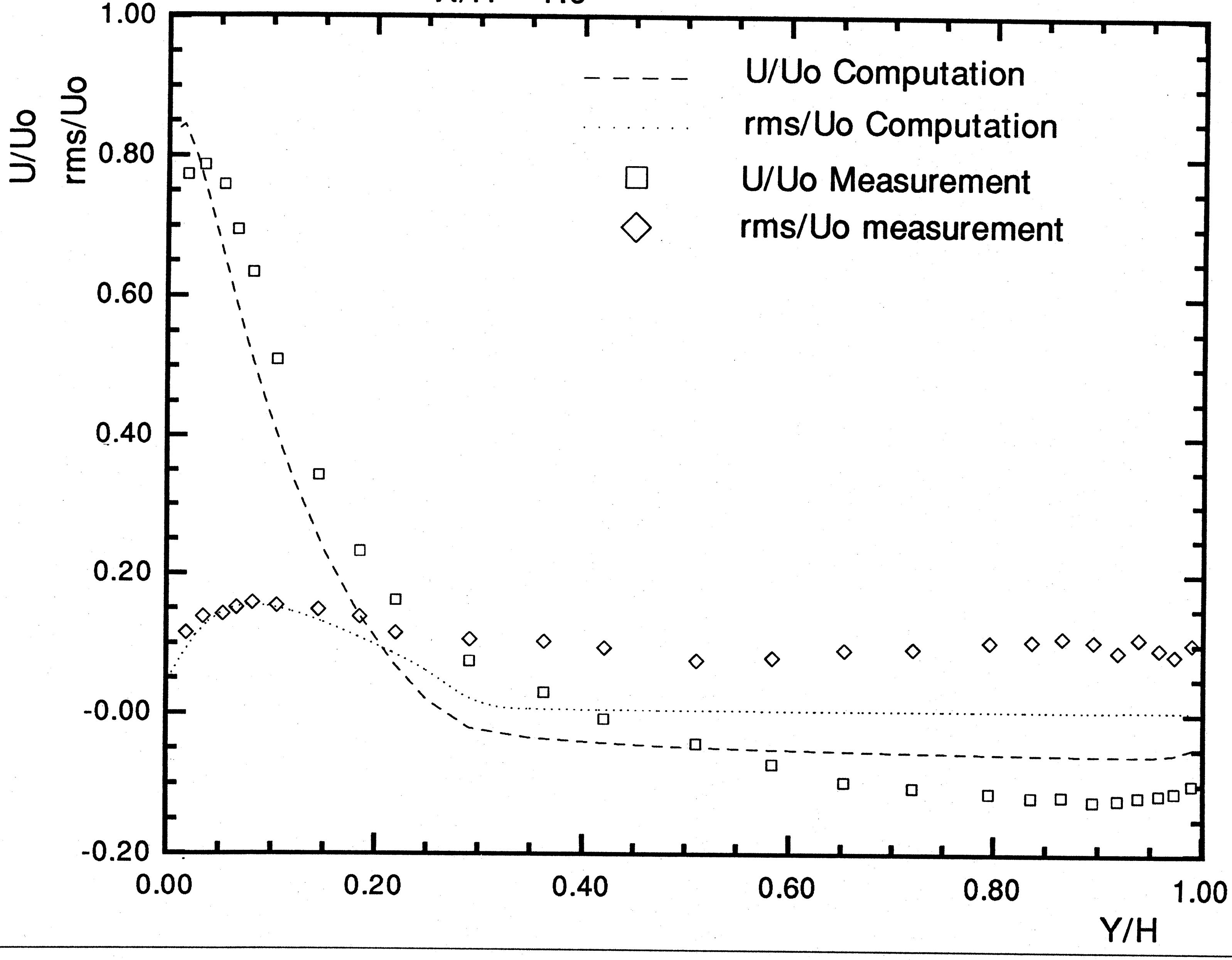


Figure 6. 2D Isothermal Forced Convection
X/H = 2.

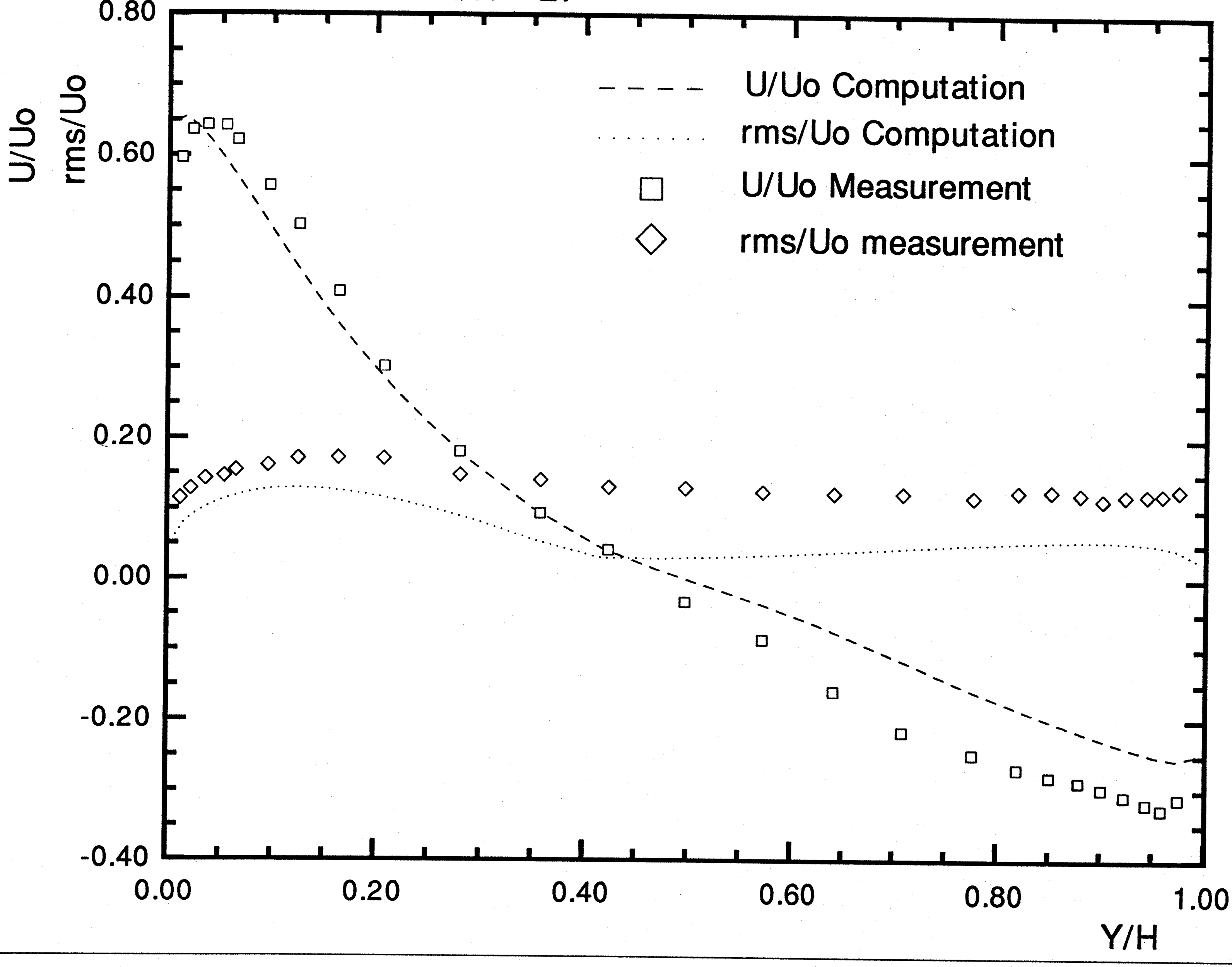


Figure 7. 2D Isothermal Forced Convection
Y/H = h/2

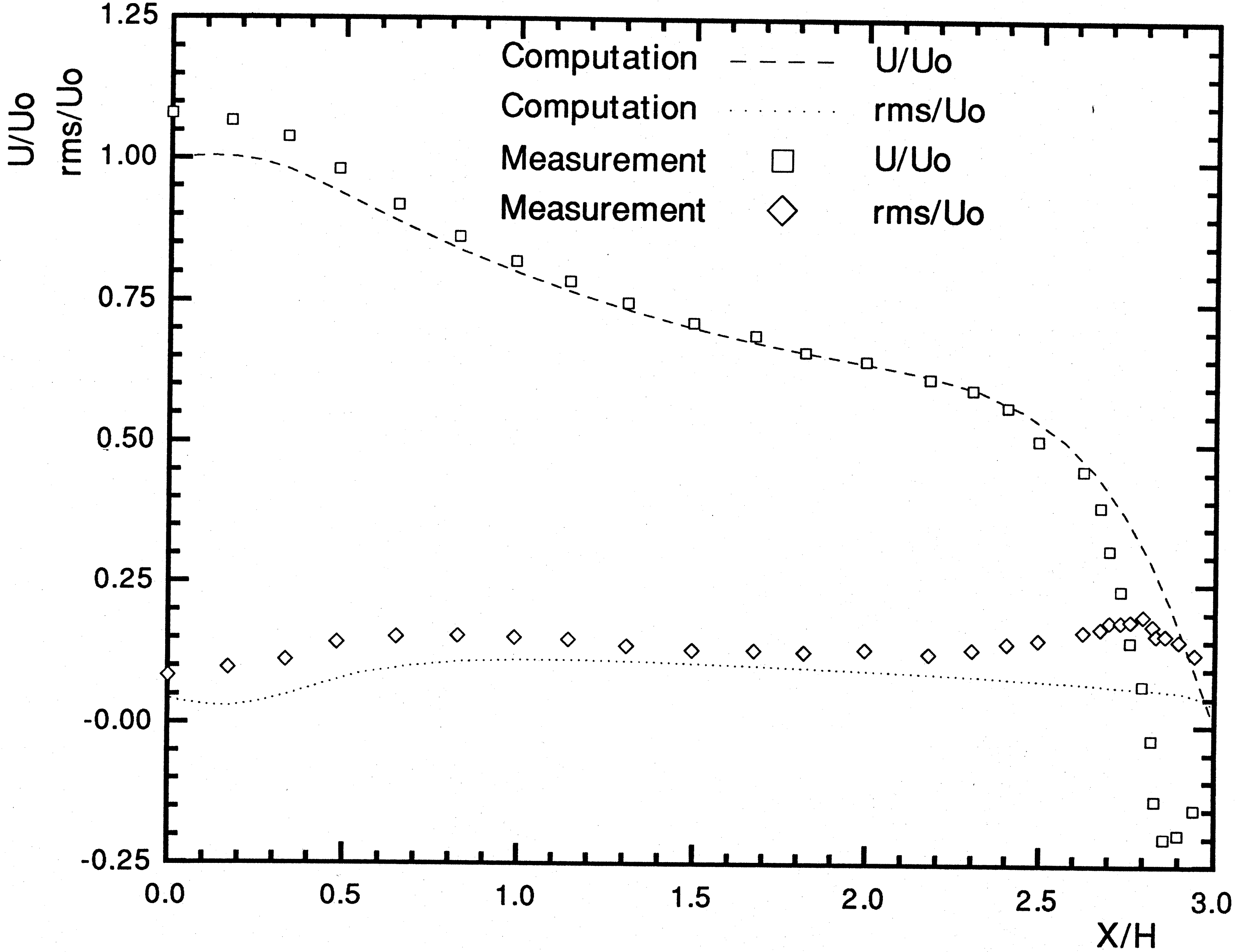
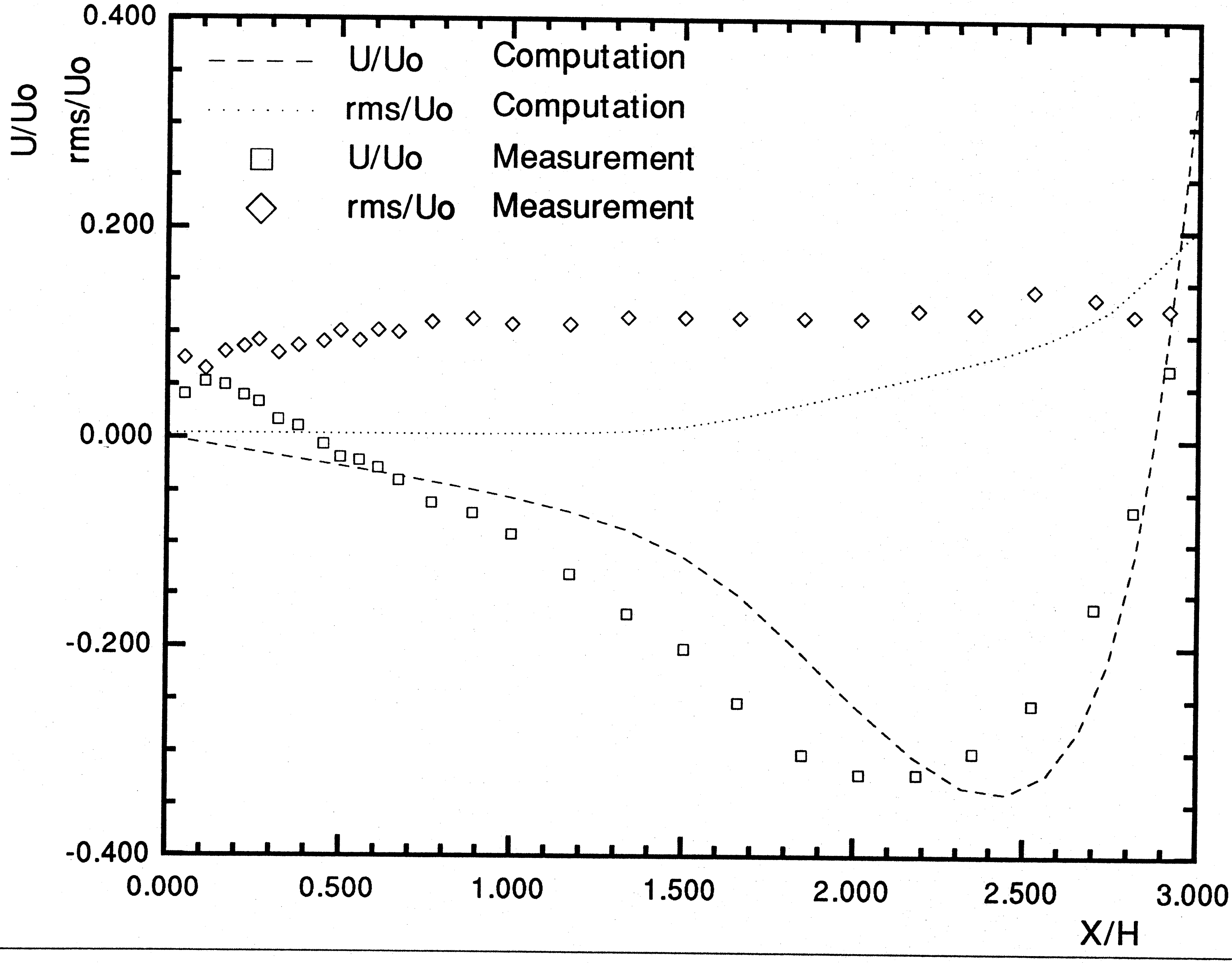


Figure 8. 2D Isothermal Forced Convection
Y/H = H-h/2



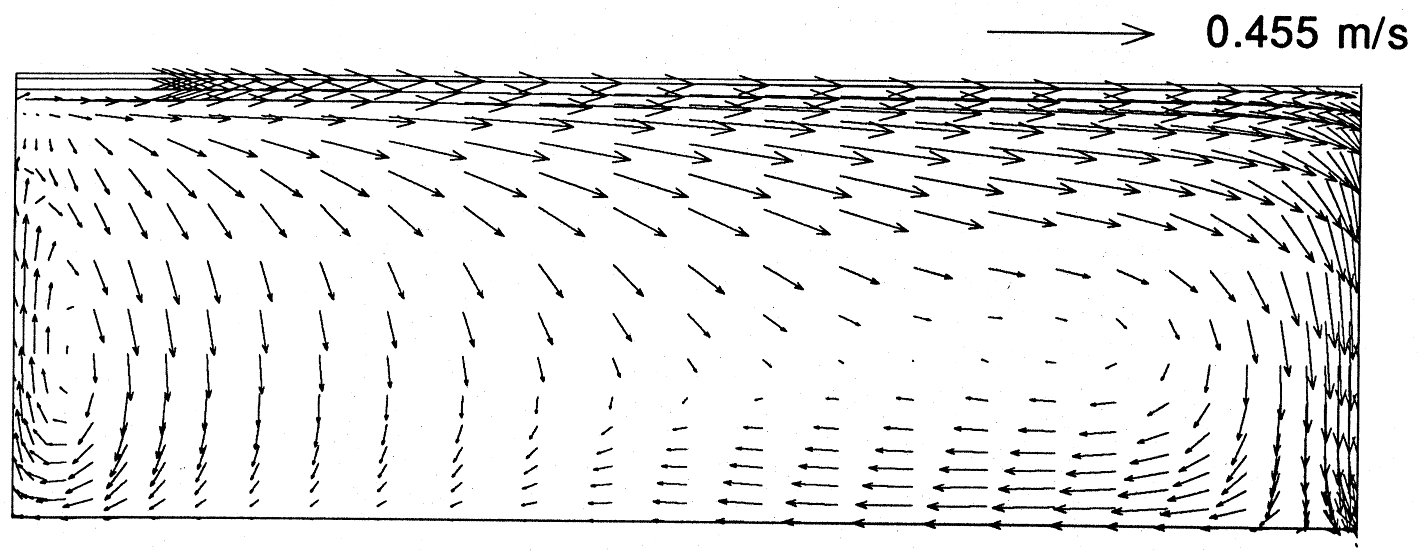


Figure 9. 2D Non-isothermal, Ar = 0.02
Velocity (m/s) Vectors

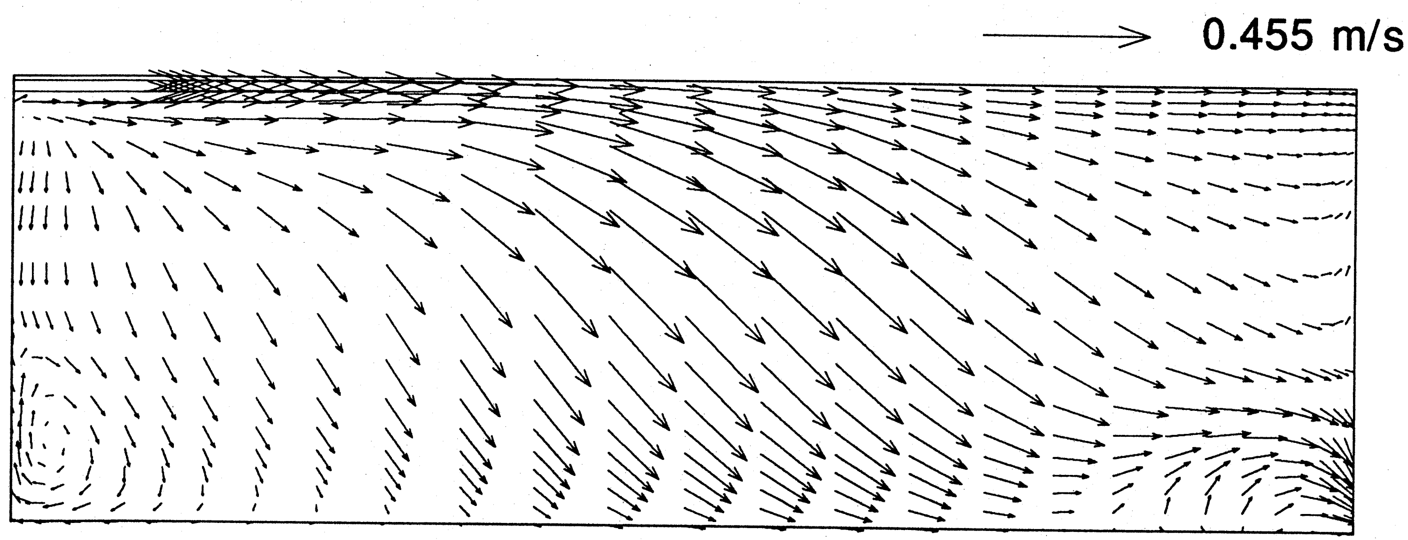


Figure 10. 2D Non-isothermal, Ar = 0.06
Velocity (m/s) Vectors

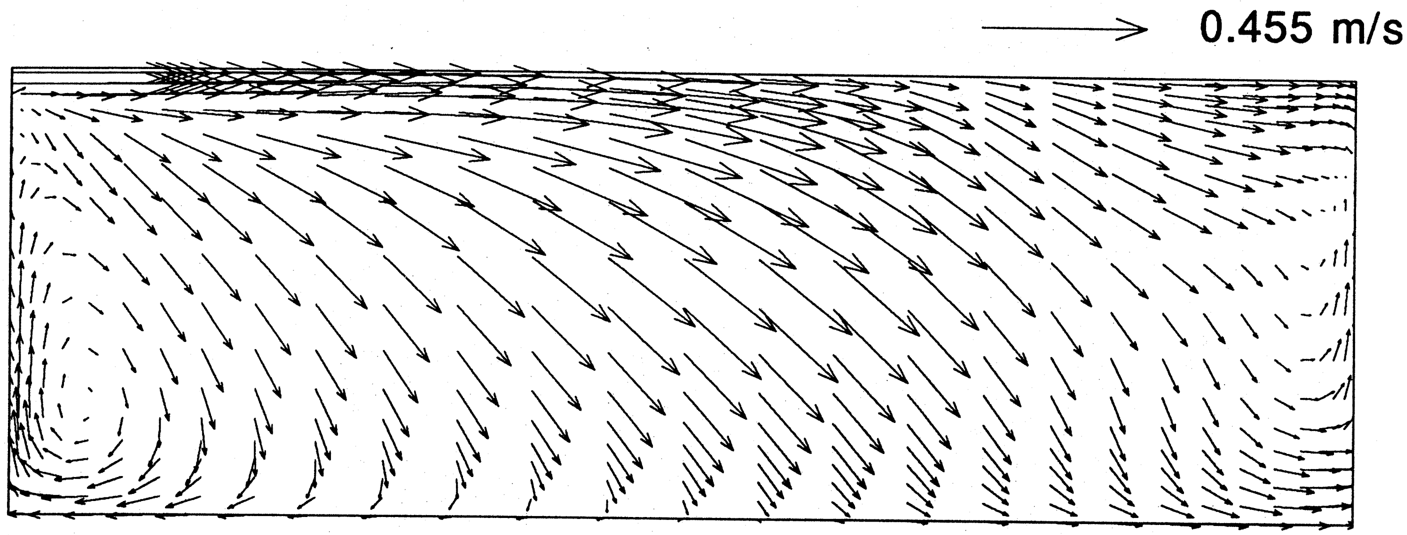


Figure 11. 2D Non-isothermal, Ar = 0.08
Velocity (m/s) Vectors

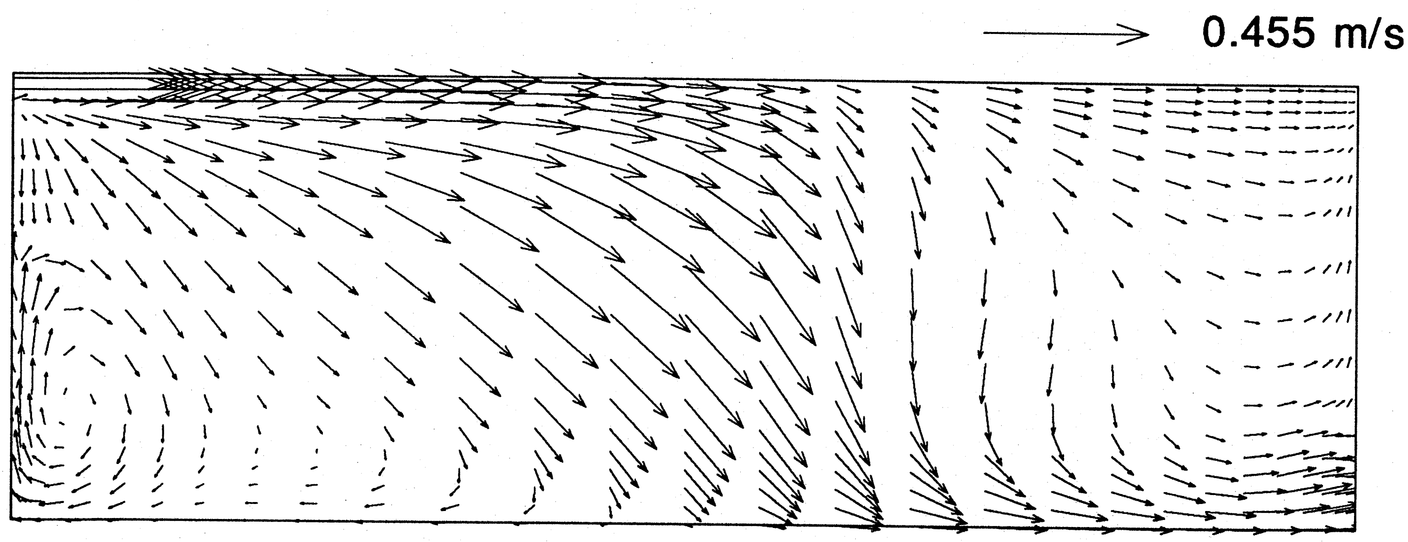
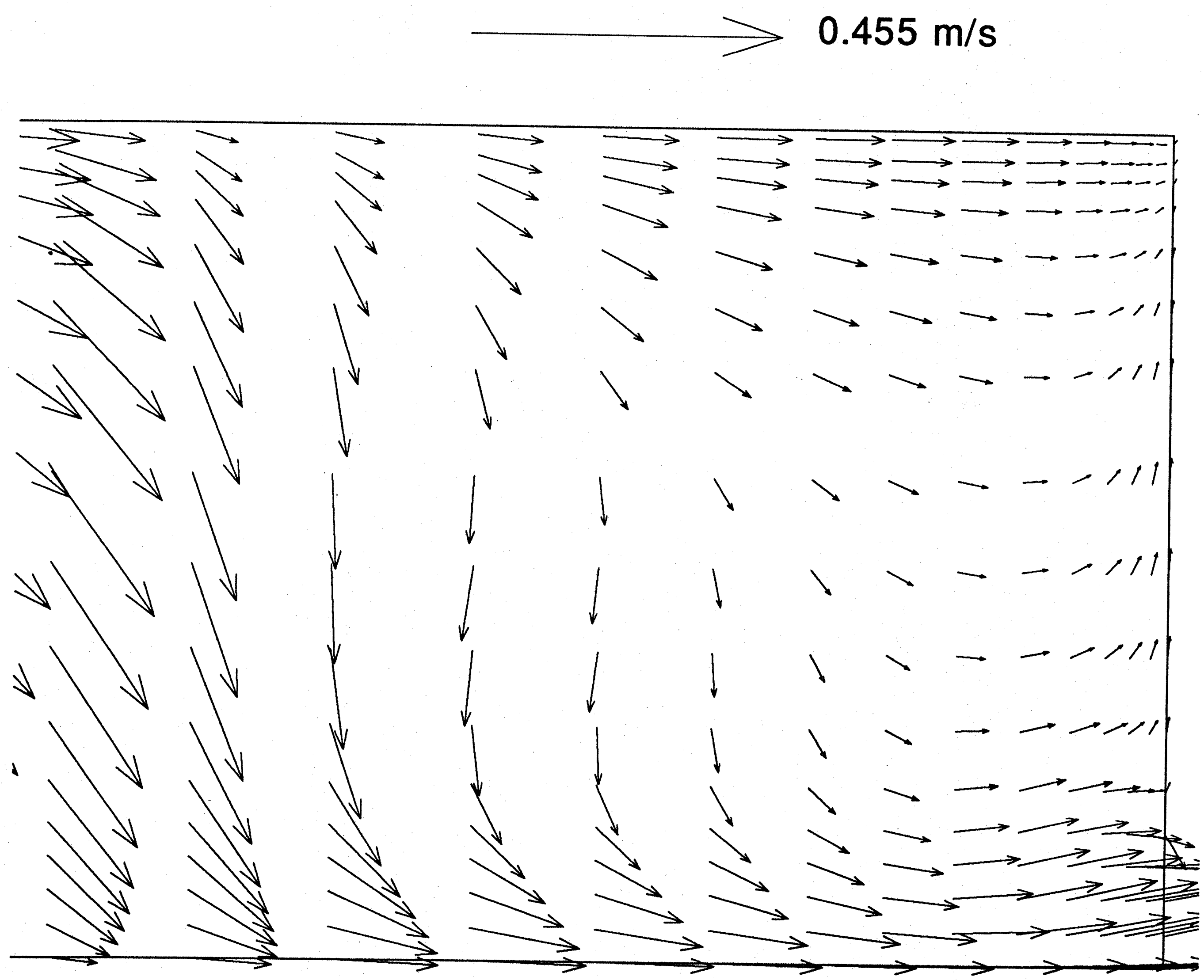


Figure 12. 2D Non-isothermal, Ar = 0.1
Velocity (m/s) Vectors

Figure 12a. 2D Nonisothermal. Ar = 0.1



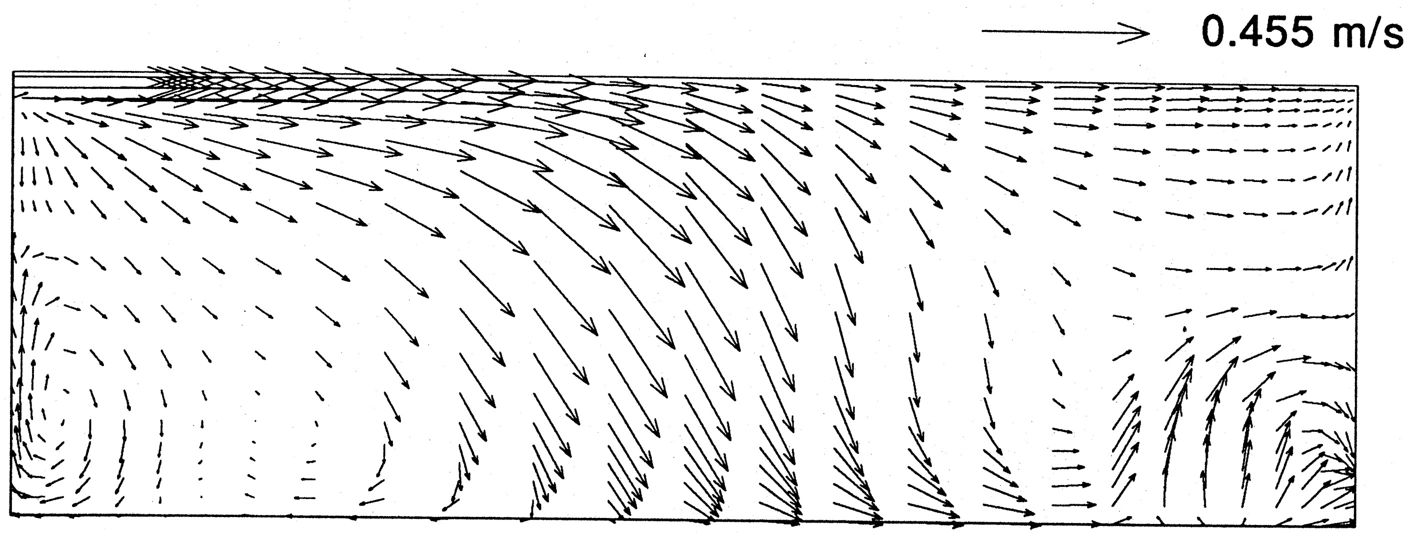
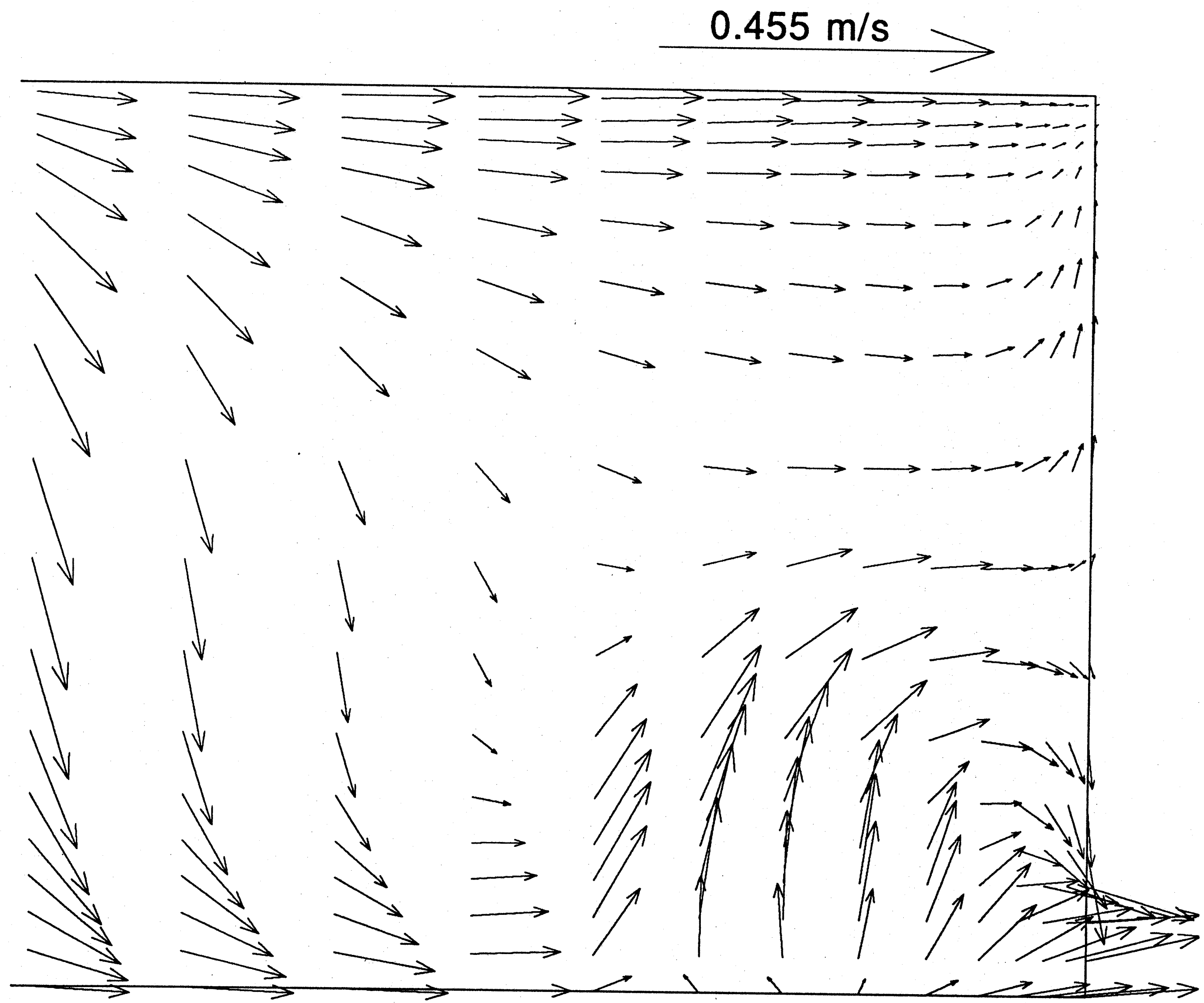


Figure 13. 2D Non-isothermal, Ar = 0.12
Velocity (m/s) Vectors

Figure 13a. 2D Nonisothermal, Ar = 0.12



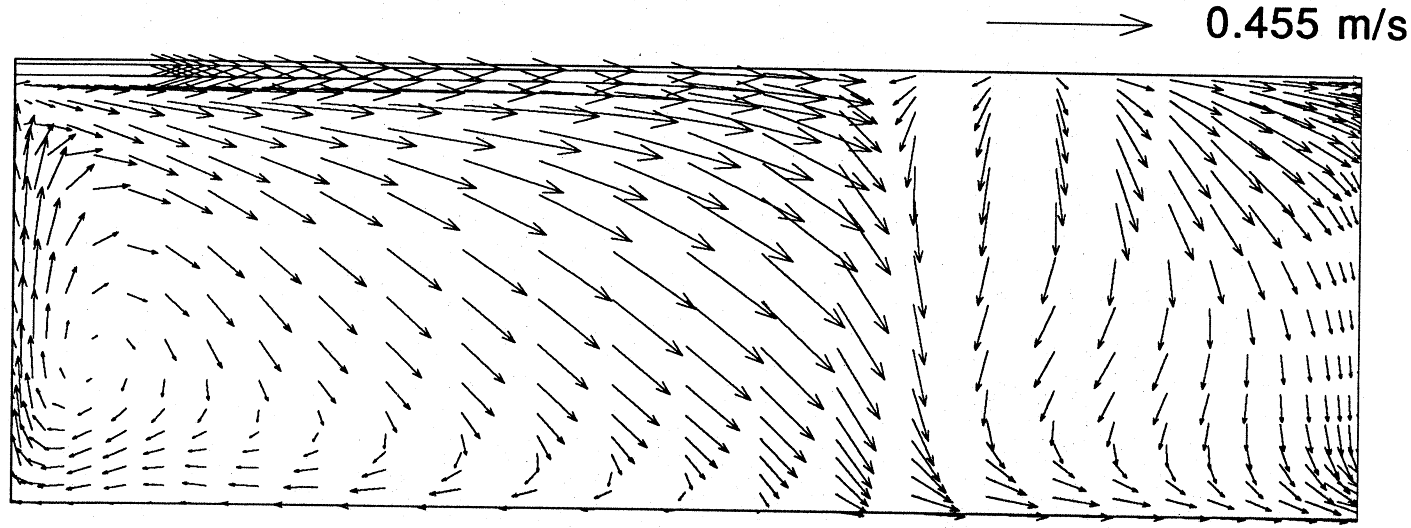


Figure 14. 2D Non-Isothermal Forced Convection, Ar = 0.143
Velocity (m/s) Vectors

Figure 15. 2D Non-Isothermal Forced Convection

Ar = 0.143