NUMERICAL MODELLING OF AIR FLOWS IN BUILDINGS AND DESIGN OF A DATA BASE OF EXPERIMENTS

P. Depecker *, J. Virgone **, G. Rusaouën **

* Professeur de l'Université Claude Bernard Lyon I, CETHIL
** Maître de Conférences de l'Université Claude Bernard Lyon I, CETHIL

ABSTRACT

During the recent years a lot of models describing air flows in buildings have been designed by Researchers. These models, which are based more or less on simplified approaches coming from fluid dynamics, share a high complexity level because of complex geornetrical structure and. complex components of buildíngs. This complexity is the source of bad, experimental fitting, most of all because scale models don't correspond to the needs for fitting.

From this background, the International Energy Agency has organized a search all over the world to collect information on projects which try to build data bases based on experiments.

We have started a large project with this goal. A flat at scale one with six rooms and an entrance hall has been equipped with. measurement systems. We are able to control and generate air flows in the flat. Air flow characteristics of components; (walls, doors ...) are known. So we are able using a gas tracking technique to determine air flows between rooms. In the same time we simulate; air flows characteristics in the flat with. a numerical code : STAR-CD1 .

This has led us to a comparison of numerical and. experimental results. We have found differences and failings of the code. These aspects are developed in the paper. They are the starting point of a discussion on CFD capabilities in the field of air flows simulation in buildings and especially when singularities (cracks or lacks of windows ...) are involved. We conclude by a look over local models which may lead to a better knowledge, and of course, simulation of the singularities.

INTRODUCTION

Seventies and eighties have been marked by a great effort made in numerical modelling of buildings. The aim was to predict their need in energy. These researches have produced a lot of results, models but also technical systems to be used. The researchers then progressively changed their goals, the new direction of search is now quality of environment. Among the questions involved in quality of environment is the quality of air and its contribution to thermal comfort.

The quality of air is identified by concentration and displacement of pollutants, being gases or solid particles. On the other hand, air is part of thermal comfort essentially through displacement measured by flow fields.

So the question of the quality of air is directly linked with the knowledge of flows induced by ventilation in rooms. in order to understand better these relationships researchers have developed a theoretical approach based on computational codes.

Such codes can be regrouped in two major families :

1) CFD codes which basis is fluid dynamics and the Navier Stokes equations. They are able to predict laminar or turbulent flows. These codes were not initially developed for the kind of problems we are looking at. They also requires large computational power. They are frame models which give velocity field, pressure and temperature in the rooms of the building on a more or less refined mesh of the ventilated areas[1].

2) Simpler codes based on models derived from fluid dynamics such as « zonal models in pressure ». The mesh required by such codes is coarser than the one of CFD codes. Such codes can be executed on small computers and the results are good according to the aim of our researches. They are able to predict the shape of flows and give a good description of ventilated flows.

[1] The flat was meshed using 48 000 cells, and mesh was refined near the walls.

Researchers specialized in our field have successfully strengthened these codes during the five last years [2] [3] [4]. The quick growth of these codes is essentially du to a fitting in terms of quality and complexity of the results with the field of building. But the complete experimental validation of these codes is still in progress and the lack of experimental data bases is crucial. This is the reason why, in the frame of Annex 23 of the IEA, we have began a program in order to create an experimental data base. It was closed by a thesis [5]. We are looking at some of the results of this program, essentially air flows between areas. They were obtained using an experimental device of the CETHIL : OPTIBAT described in part II. If the results are good for fitting with computational codes of the second family, our experience is that the cost required for such data base is too heavy and cannot be generalized to other room configurations. So we are actually looking at a new approach : we are trying to build data bases using codes of the first family. In part IV we present our very first results. They compare a CFD code (STAR CD) and experiments on OPTIBAT.

DESCRIPTION OF OPTIBAT

Our results were obtained with OPTIBAT. It is a flat at scale one including every room of a normal flat : living room, kitchen, bathroom, toilet, three bedrooms(see fig. [1]). It is installed in a large hall for experiments and is equipped for measures of temperatures and concentration of gases. OPTIBAT is a standard flat designed in a prefabricated way using a lot of concrete. Its structure is based on two opposite windowed fronts. Two climate control areas are facing the fronts which enables us to control pressure and temperature, and to simulate for example wind. The overall surface is 88 square meters with a height to the top of 2.5 meters. Fronts are greatly windowed (more than 50 %) and windows are equipped with steel sliding frames.

EXPERIMENTAL RESULTS - DATA BASE

1- Measure of wall permeability

This measures were made for every wall in OPTIBAT using two methods :

- guarded zone method
- passive method

The results of the two methods are matching together. So, the results of the first method are used for comparison with numerical simulations. The wall is described by a law of the form :

$$Q = K\,(\Delta P)^n$$

where K, n are experimentally fitted coefficients associated to the wall, Q is the flow and ΔP the pressure gradient. Figure [2] identifies the walls of the flat. Figure [3] gives the experimental values for the walls.

2 - Measure of flow between areas

The measures of flows between areas, independent of those of permeability, were made in indirect way. They are based on gas tracer techniques. These methods can be described as follows : a small flow of a particular gas(N_2O or SF_6) is injected in an area (a room of the flat). Then we wait until a stationary flow is obtained in the flat and we measure gas concentration in every area. From these concentrations we recover, using the continuity equation for air and gas, the flow between areas. This experiment is done for every area if using a single gas. If we are using several gases the total number of experiments is reduced and results are obtained faster[5]. The results are included in the Annex 23 of the IAE.

For the area i and the gas k we have :

$$\frac{V_i}{T_i}\frac{dC_i^k}{dt} = \frac{S_i^k}{T_k} + \sum_j C_j^k \frac{Q_{ij}(1-\delta_{ij})}{T_j} - \frac{C_i^k}{T_i}\sum_j Q_{ji}(1-\delta_{ji})$$

where : C_i^k is the concentration of gas k in area i
V_i the volume of area i
T_i the temperature of area i
S_i^k the gas injection in area i
Q_{ij} and Q_{ji} the flows between areas i and j
δ_{ij} the Kronecker symbol

The conversation of air can then be written

$$-\frac{Q_{i0}}{T_0} = \sum_{\substack{j=0 \\ j\neq i}} \frac{Q_{ji}}{T_i} - \sum_{\substack{j=1 \\ j\neq i}} \frac{Q_{ij}}{T_j}$$

NUMERICAL RESULTS

In the same time numerical simulations of flows in OPTIBAT were done using the CFD code STAR CD [6]. Velocity fields were achieved using the same pressure boundary condition as in the experiments. The aim was to get flows and to compare with experimental results.

The code is not able to deal with walls like we have in buildings. But, it is able to simulate them as porous media for which an equilibrium condition between pressure and forces is written :

$$K_i\, u_i = \frac{\partial P}{\partial \xi_i}$$

were ξ_i is the orthotropic direction
K_i the permeability
u_i the surface velocity in the direction ξ_i
P the pressure

The permeability Ki is expressed in a linear way

$$K_i = \alpha_i \|V_m\| + \beta_i$$

were $\|V_m\|$ is the average surface velocity
α_i, β_i are coefficients to be furnished to STAR CD by the user

This law is the only one included in STAR CD. We were obliged to use it and give values for α_i and β_i. These coefficients were obtained by fitting between the law of STAR CD and the classical law $Q=\Delta P^n$. As a consequence the coefficients α_i and β_i are experimental coefficients obtained in an indirect way from the measured coefficients K_i and n_i of the different walls of OPTIBAT. The method can be describe as follows :

STAR CD works with a law of the form

$$\Delta P = \rho(\alpha V_m + \beta)V_m$$

Which can also be written :

$$\Delta P = \frac{\rho\alpha}{S^2} Q^2 + \frac{\rho\beta}{S} Q = AQ^2 + BQ$$

By comparison with $Q=K(\Delta P)^n$ we determine coefficients A and B and then α, β needed by STAR CD. The code then gives velocity field on the surface of the walls, which gives the flow Q_{ij}^{num} between areas i and j :

$$Q_{ij}^{num} = \int V_n\, ds$$

These results can be compared to experimental ones :
$Q_{ij}^{exp}=K_{ij}(\Delta P_{ij})^{n}{}_{ij}$

The computational mesh is shown in figure [4]. Figure [6] shows the velocity field all over the flat. Pressure boundary conditions are those of the experiments [see figure [2]].

$\Delta P_1=16$ Pa $\Delta P_2= -81$ Pa $\Delta P_3= -2.8$ Pa

The simulations were done both in turbulent (K, ε) and laminar flows. Because of the weak velocities ($V\in[0.3\ 10^{-4}$ m/s; 0.05 m/s]) the results are equivalents, and the convergence is reached in 200 steps. So we consider the flow as laminar. The pressure field which is not shown here is nearly uniform.

Figure [5] is a comparison of flows Q_{ij}^{num} and Q_{ij}^{exp}. A large difference is visible, numerical results are 4 to 9 times as big as experimental ones.

So it seems rather difficult to directly use a CFD code such as STAR CD to simulate air infiltration in a building. Of course, STAR CD has not been designed to deal with such kind of problems. But our aim was to know if codes like STAR CD can be quickly adapted for such kind of original problems.

According to us, two reasons can explain these differences :

1) As said just before STAR CD has not been designed to simulate objects such as buildings which are complex and particulars. Pressure gradients are also weak compared to the volume of the flat.
The wall were described as filters with a uniform permeability to air. This uniformity is the cause of the uniform pressure filed which does not describe real flows between areas. We think that exploitation of such codes in the field of building must lead to a new modelling of wall permeability including cracks effects. The aim is to better describe flows through equipment : seals of window frames, joints between walls, shutters...

2) Boundary conditions used in the simulation can be changed. Pressure applied to the walls does not take into account boundary layer. Experimentally, the measures of pressure in the climate control area were done using a single point at the center of the area and not over a grid of points. Therefore, the associated pressure can be considered only as an average value on the front and we are not able to describe the experimental field in the climate control area.

The wall pressures were measured at only 5 cm of the front, but we are not sure that it is enough to include effects of proximity usually described by the pressure coefficient C_p which depicts the effect of over or under pressure in the boundary layer.

A modelling problem appears now clearly that was not, in these preliminary experiments, taken into account.

Finally, the pressure gradient are weak and even a small uncertainty on them can lead to differences on flows Q_{ij} between areas that we are not able to guess.

CONCLUSIONS

The aim of the researches described in this paper is to compare numerical and experimental air flow results in a flat. Our goal is, at the end, to replace long and costly experiments by the use of CFD codes in order to build data bases.

Our first results convince us to be careful, essentially because CFD codes link STAR CD are not well fitted to the filed of building. We are in front of two kinds of difficulties inducing two different levels of errors :

- errors in the modelling process du to a bad fitting between models and physical objects.
- errors coming from input in the code of not well known boundary conditions.

We think that future exploitation of CFD in the filed of building ventilation must be preceded by researches on well fitted models and adapted boundary conditions. These particularities of the field are part of the research projects of the CETHIL for the future years.

REFERENCES

[1] **Q. CHEN, A. MOSER, A. HUBER,** « Prediction of buoyant turbulent flow by a low Reynolds number K,ε model. », ASHRAE Transactions 96, 1990

[2] **F. ALLARD, C. INARD,** « Natural and mixed convection in rooms : prediction of thermal stratification and heat transfer by zonal model. », ISRACVE, Tokyo, 22-24 July 1992

[3] **M. SANDBERG, S. LINDSTROM,** « A model for ventilation by displacement. », ROOMVENT, Stockholm, 10-12 July 1987

[4] **C. INARD, D. BUTY,** « Simulation of test case d with zonal model. », International Energy Agency, Annex 20, Research report n° 1.26, May 1990, 29 p.

[5] **F. AMARA,** « Contribution à la caractérisation aéraulique des bâtiments. Etude des transferts aérauliques dans les locaux multizones », Thèse INSA de LYON, Avril 1993

[6] **V. DOGARU,** « Modélisation des écoulements de l'air dans la cellule OPTIBAT. », Rapport DEA Juillet 1994. Enc. P. DEPECKER, G. RUSAOUEN

FIGURES

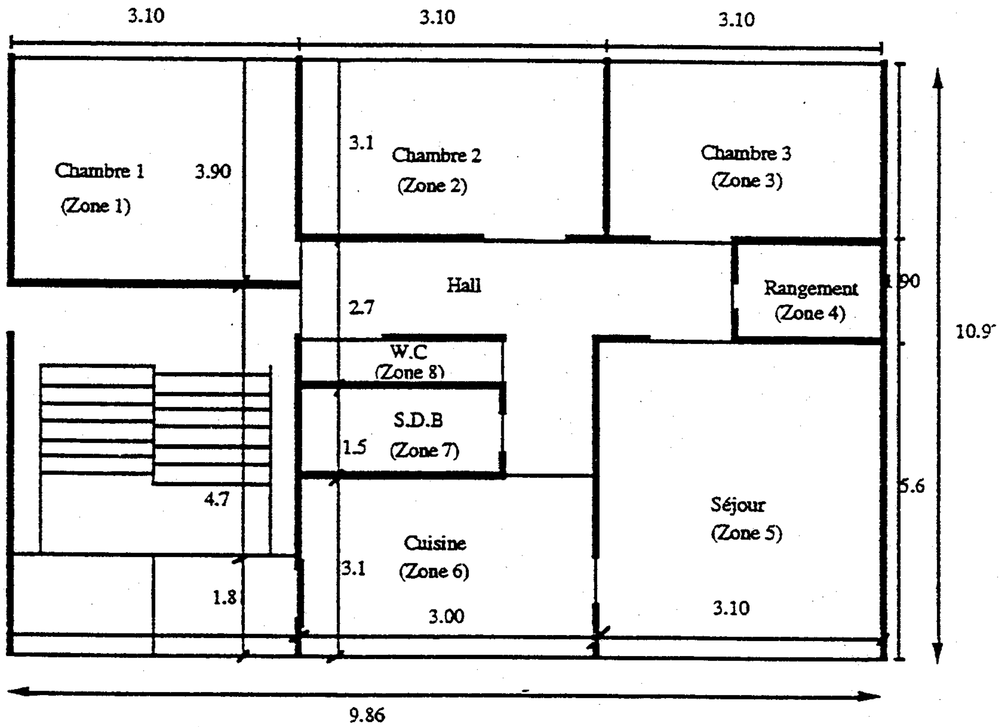


Figure 1-1 : Map of OPTIBAT

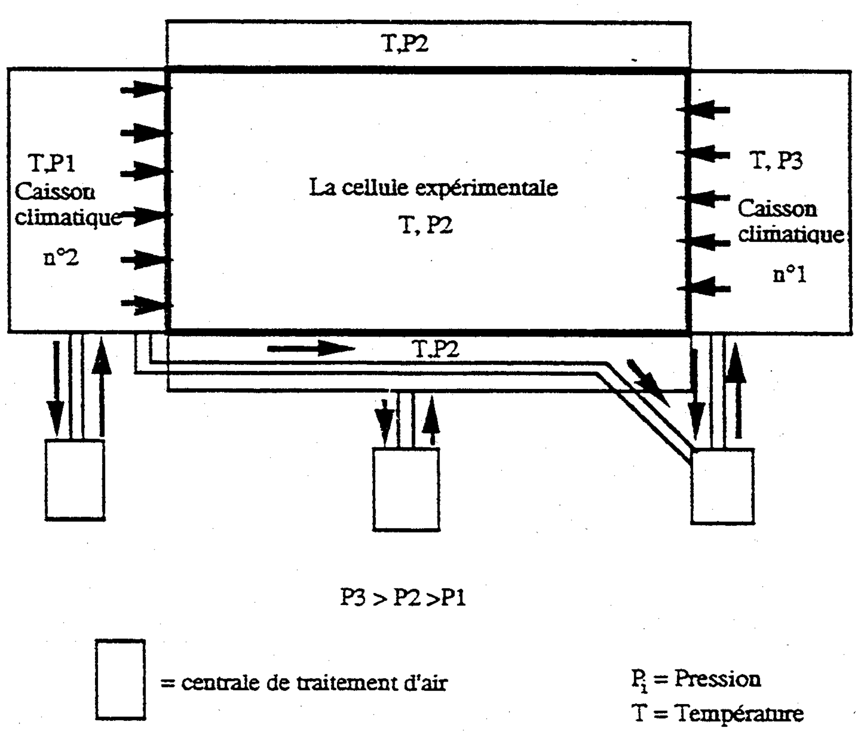


Figure 1-2 : Vertical cut of OPTIBAT

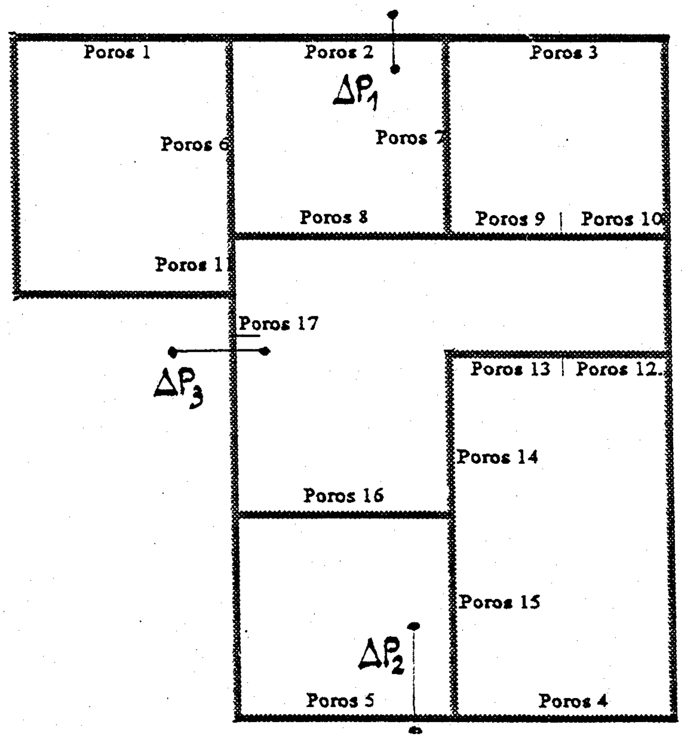


Figure 2 : References of the walls of OPTIBAT

	Méthode de la zone gardée		Méthode passive	
Désignation paroi	n	K [m³/hPaⁿ]	n	K [m³/hPaⁿ]
Poros 1	0,59±0.03	12,62±1.04	0,58±0.02	13,43±0.91
Poros 2	0,57±0.01	11,93±0.4	0,6±0.04	11,82±0.7
Poros 3	0,61±0.03	12,84±1.05	0,55±0.04	10,02±1.08
Poros 4	0,55±0.03	13,52±1.6	0,57±0.01	13,34±0.21
Poros 5	0,61±0.05	14,33±1.45	0,56±0.005	14,46±1.02
Poros 6	0,9±0.03	0,1±0.02	0,9±0.001	0,14±0.001
Poros 7	1.0±0.06	0,07±0.02	0,87±0.01	0,17±0.01
Poros 8	0,71±0.01	20,99±0.2	0,78±0.05	20,22±1.03
Poros 9	0,66±0.001	14,17±0.03	0,59±0.004	14,94±1.26
Poros 10	0,66±0.01	2,89±0.21	0,65±0.01	2,97±0.08
Poros 11	0,92±0.03	2,54±0.22	0,84±0.02	2,49±0.16
Poros 12	0,51±0.002	5,48±0.04	0,51±0.01	5,64±0.02
Poros 13	0,71±0.03	14,67±1.4	0,76±0.01	15,03±0.64
Poros 14	0,64±0.01	6,47±0.02	0,64±0.001	6,29±0.02
Poros 15	0,74±0.05	1,76±0.26	0,81±0.02	1,24±0.21
Poros 16	0,65±0.05	3,34±0.59	0,59±0.04	3,94±0.54

Figure 3 : Permeability of the walls of OPTIBAT

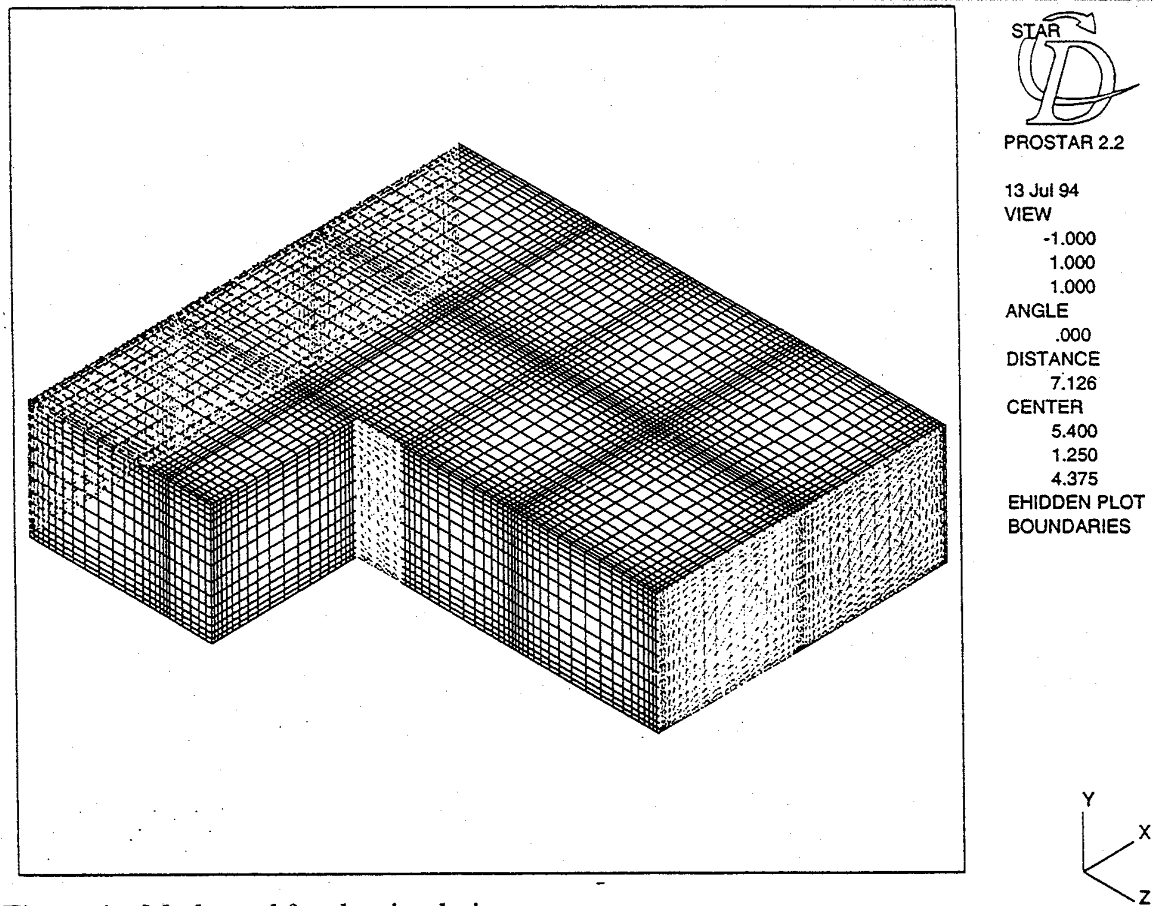
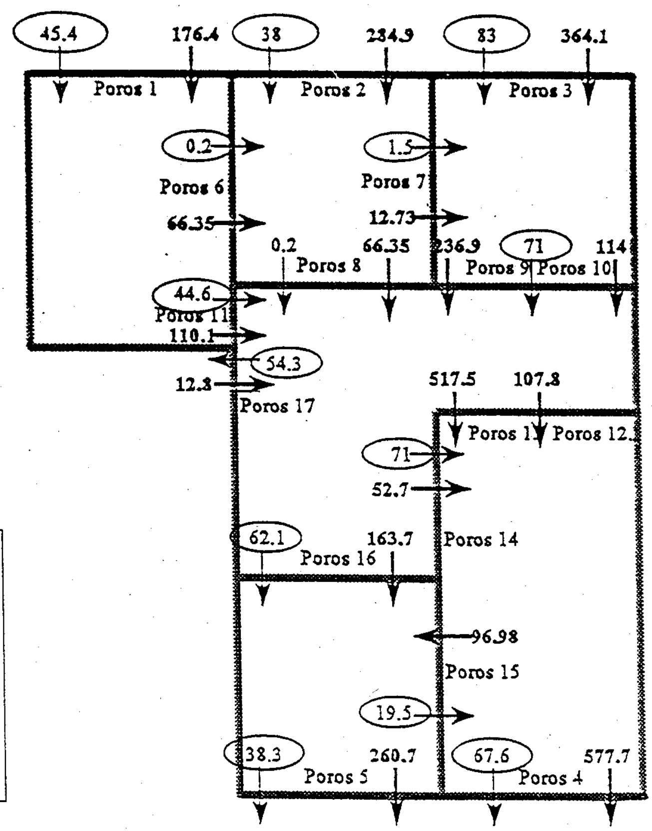


Figure 4 : Mesh used for the simulations



Values in ellipsis are experimental flows in m^3/s through the wall . Other values are numerical results.

For example in Poros1 we have :

45.4 m^3/s in the experiment
176.4 m^3/s in the simulation

Figure 5 : Comparison of numerical and experimental results

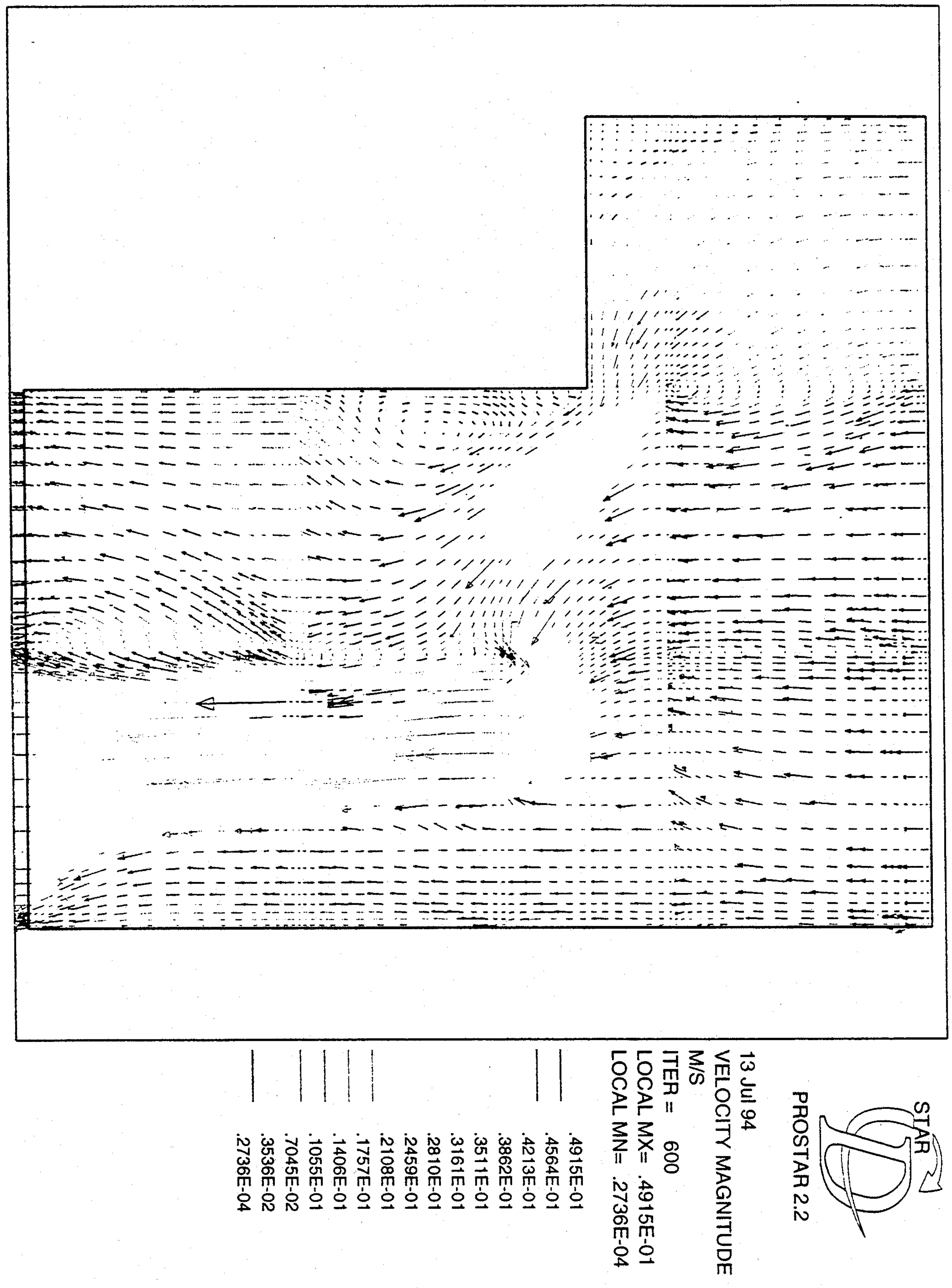


Figure 6 : Velocity field obtained by STAR CD