NUMERICAL AND EXPERIMENTAL ASSESSMENT OF A FLOW FIELD IN A VENTILATED INDUSTRIAL HALL

Jordan DENEV*, Peter STANKOV*, Dimiter STOYANOV* and Pavel SPASSOV**

*Department of Hydroaerodynamics, Technical University of Sofia, Kliment Ohridski Str. 8, Sofia 1000, Bulgaria
**EQE Bulgaria: Safety, Engineering and Management Consultants. Chr. Smirneski Str. 1, Eleven Floor, Sofia, Bulgaria

ABSTRACT

The paper presents the results from a numerical and experimental investigation of the velocity distribution in a ventilated industrial hall in the nuclear power plant in Bulgaria. The room with a complex geometry and irregular boundary conditions was calculated using the finite volume discretisation of the Reynolds-averaged Navier-Stokes equations closed by the standard k-ε turbulence model. The numerical results are compared with the experimental data in representative cross-sections which shows a satisfactory agreement. Discrepancies are found to be mainly due to uncertainties in the imposed boundary conditions as well as to the complex irregular structure of the flow.

INTRODUCTION

The previous experience of the authors (Stankov et al. 1992, Denev et al. 1996) shows that flows in ventilated rooms, which are determined by simple geometry and simple boundary conditions (and hence, possess a clear flow structure), can be predicted reliably by numerical computations. The present investigation concerns the flow study in a real industrial hall situated in the nuclear power plant in Bulgaria. The battery hall presents a case with a complex flow structure and significantly irregular boundary conditions.

The purpose of the present investigation is to obtain both numerical and experimental data on the complex velocity field in the battery hall. The sources of discrepancies are sought and identified by means of comparison between the numerical and experimental results. Measures for decreasing the sources of discrepancies are outlined. These measures will be taken into account in subsequent numerical and experimental investigations. The investigations will be made for the same industrial hall, with the final aim of comparing refined turbulence models applied to such real practical flows.

THE NUMERICAL METHOD

The three-dimensional Reynolds averaged Navier-Stokes equations are used in the investigation. The system of partial differential equations is closed by solving two additional transport equations: for the turbulent kinetic energy k and for its dissipation rate ε (the standard version of the k-ε turbulence model is used). The discretisation is made by the method of finite volumes on a collocated numerical grid. The SIMPLE-algorithm was utilised for coupling the velocity and pressure fields. The numerical code FASTEST/3D, developed at LSTM (University of Erlangen-Nuernberg), was used. More details about the method and its algorithmic details can be found in Peric (1985) and in FASTEST Manual (1993).

Having in mind the small temperature differences in the room air and the surrounding walls, the assumption of isothermal flow conditions is made. The temperature difference measured in the planes Y=2.95m and Y=7.45m (see Fig.1) is within the narrow range of 25.1÷26.1 ^{o}C. The mean temperature of the walls varies from 22.9 ^{o}C for the coldest one - the north wall (Y=0m) up to 27.4 ^{o}C for the ceiling (the warmest one). However, the continuation of the present study planned in the near future is aimed to include also turbulence models for refined modelling of the buoyancy forces like those of Ince and Launder (1989), Hanjalic and Vasic (1993) and Hanjalic, Durst and Kenjeres (1994). These models have already been implemented in the FASTEST/3D numerical code.

The number of control volumes used was 48x51x26 = 63 648. This allowed a resolution fine enough to capture the geometry of the batteries, the geometry of the ventilation ducts as well as to impose the necessary boundary conditions. The numerical mesh used in the computations is given in Fig. 1.

The computation was made on an HP C110 workstation. 8431 outer iterations for the SIMPLE algorithm were necessary to reach the convergence criterion (this took about 16.5 hours CPU-time).

GEOMETRY OF THE INVESTIGATED ROOM

The geometry of the hall, whose dimensions are 8.10m x 11.80m x 4.95 m, is presented in Fig. 1. The figure shows the layout of the supply and exhaust duct systems, of the batteries and of the main hall beam located on the ceiling. The planes of Y=*const* in which the measurements are made, are also shown.

The four rectangular air-supply openings are positioned at a height of 2.78m above the floor at Y=11.60m (note that they are positioned inside the room, not on the surrounding walls). They are 0.2m wide and 0.3m high. The plane of each supply outlet is inclined at 45 degrees downwards in relation to the horizontal plane.

The 7 outlet openings in the exhaust duct (also having dimensions of 0.2m x 0.3m) are positioned at the opposite part of the hall. Four of them are placed near the ceiling at Z=4.80m and three - near the floor at Z=0.7m.

BOUNDARY CONDITIONS

The most important boundary conditions for the development of the investigated flow are those in the supply outlet openings. For the proper evaluation of the velocity in these openings, measurements in nine evenly distributed points were made for each opening. The results of the measurements are summarised in Table 1. Note that the supply outlets are numbered starting from left to right in Fig. 1.

Table 1. The measured velocities C, [m/s] in each supply outlet.

Supply outlet 1			Supply outlet 2		
14.21	13.18	10.16	16.24	10.65	11.25
13.28	13.55	7.57	19.37	15.38	10.86
8.04	16.14	12.40	16.78	17.50	17.37
mean C=12.06[m/s]			mean C=15.04[m/s]		

Supply outlet 3			Supply outlet 4		
17.17	17.55	13.35	20.50	15.80	12.25
17.00	18.40	15.55	15.85	15.50	16.15
15.45	16.15	12.00	16.80	16.85	11.50
mean C=15.85[m/s]			mean C=15.69[m/s]		

As can be seen in Table 1, the velocities in the supply openings show a considerable irregular distribution. The difficulties in evaluating the proper direction of the flow are increased by the presence of flat iron positioned perpendicular to the plane of the openings at their lower part. This iron serves to redirect the jet trajectories which should be normal to the plane of the openings. The iron is trapezoid in shape; its dimensions are of the same order as those of the supply openings.

The used in the computations boundary conditions, which correspond to the measurements, are listed in Table 2. The values are obtained using additional information for the direction of the velocity vectors, for which no precise measurements exist at the present moment.

Table 2. The velocity components u, v and w, [m/s] imposed as a boundary condition in each supply outlet. Here $C = \sqrt{u^2 + v^2 + w^2}$

Supply outlet 1	Supply outlet 2
u = 1.71	u = 2.14
v = -6.92	v = - 8.63
w = -9.77	w = -12.18
C=12.06[m/s]	C=15.04[m/s]

Supply outlet 3	Supply outlet 4
u = 2.25	u = 2.23
v = - 9.09	v = - 9.01
w = -12.84	w = -12.72
C=15.85[m/s]	C=15.69[m/s]

The velocity components given in Table 2 result in a flowrate of 6841 [m^3/s]. Here we should mention that the ventilation is designed in such a way that the flowrate exhausted from the room is higher than the supplied airflow. As a result a considerable amount of air penetrates through the door crack. However, a balanced flowrate is assumed for the computations. For a more precise evaluation of the flowrate balance, additional measurements of the flowrate penetrating through the door is necessary.

Additional developments of the numerical code FASTEST/3D were made in order to allow to handle the particular conditions of the predicted battery hall such as:

- treatment of complex ventilating inlets and outlets positioned inside (not on the boundaries of) the room;
- obstacles such as batteries, HVAC pipelines and the main hall beam.

THE EXPERIMENT

The experimental investigation includes velocity measurements in planes parallel to the plane of the supply openings, i.e. planes with Y=const. An omnidirectional TSI 8475 air speed transducer was used. The accuracy of the measuring equipment is ±3.0% of reading and ±1.0% of full scale of selected range (which for the particular measurements performed was 2.0m/s). During the measurements the velocity transducer was fixed on a specially designed stand.

The mean value of the velocity at each point was obtained for a measurement period of 5 minutes with a time step of 0.1s. The signal was directly transferred from the data acquisition system to the hard disk of a portable computer system.

The time period was considered to be sufficient after a series of successive measurements at some particular points of the hall which showed a repeatability of the mean velocity values within the limits of 8%.

RESULTS AND DISCUSSION

Numerical results for particular cross-sections (vertical planes at Y=7.45m and 9.15m) of the battery hall compared with the respective experimental data are shown in Figs. 2 and 3. The cross-sections are somewhat representative since they are between the batteries where people move during their work. The comparison between the experimental data and the numerical results shows satisfactory qualitative agreement, which means that the type of velocity profiles should be similar for both measurements and calculations. At the same time, good qualitative agreement can be seen at a height of 1.5m < Z < 3m above the floor, but not in the region adjacent to floor. The reason for this should be sought in:

- the complex structure of the flow;
- the uncertainties in determinating the boundary conditions;
- the superposition of the above two reasons.

The flow structure in the battery hall does not follow any of the common ventilation schemes which are characterised by their simplicity and a clearly identified flow direction. Unlike such idealised conditions, the situation in the battery hall is the real one and both the calculations and the measurements follow these real flow conditions. In the case under investigation, the real flow conditions are characterised by jets which are directed downwards; so they reach the batteries within a short distance and after that their free-jet trajectories are destroyed. Hence, there is no dominating flow direction in the measured planes (Y=7.45m and Y=9.15m); instead, many recirculation zones are present there which cause the very complex and irregular flow patterns. These complex patterns can be seen in Fig. 4 which presents the velocity vectors in a horizontal plane. The height of the plane (Z=2.0 m) corresponds to the height where measurements are made (cf. Figs. 2 and 3). The four supply-air jets and theyr direction, imposed as a boundary condition, are also seen in the figure.

The complexity of the flow is additionally increased by the fact that the batteries are located irregularly and nonsymmetrically in relation to the geometry of the room. Another reason for the complexity is that the real direction of the jet trajectories is different for the four different supply outlets (see Table 2).

However, the presence of recirculation zones is known to be a weak point in the performance of the standard k-ε turbulence model (see e.g. Leschziner and Rodi 1981). This is the mechanism by which the complexity of the flow influences directly the accuracy of the numerical results and is a source of the discrepancies observed.

The boundary conditions are determined by using omnidirectional velocity transducers; obviously only the size, but not the direction of the velocity vectors is precisely determined by the measurements. Furthermore, the measured values are positioned exactly in the supply openings thus not permitting to obtain the necessary information about the further distribution of the jet trajectories. Obviously, in order to solve this problem, the measurements should be extended in the area downflow the supply openings.

The fact that the boundary conditions in the supply outlets need more precise evaluation, is confirmed by comparing the numerical and experimental data in a plane clearly dominated by the boundary conditions (Y=10.80m). In this plane, positioned close to the supply outlets, no satisfactory agreement with the experimental data is obtained. Therefore, all other subsequent planes are also influenced by the boundary conditions causing a great deal of the discrepancies observed.

The uncertainties in evaluating the boundary conditions superpose over the complex flow geometry. Hence, small deviations in the jet trajectories can result in the jet striking the side surface of the batteries (instead, e.g. of the upper surface). As a consequence a completely different flow situation may occur. This confirms again the need for more detailed measurements in front of the supply outlets.

Other possible sources of discrepancies are the deviation from the isothermal assumption and the disbalance in the flowrate of the ventilation system, which was not accounted for in the present computations.

CONCLUSIONS

A comparison was made between the numerical and experimental data on the velocity distribution in a real industrial hall with complex geometry. It was made along vertical lines in two planes positioned parallel to the plane of the four supply openings.

A satisfactory qualitative agreement between the measurements and the experimental data is observed. A quantitative agreement was found only for the points at a height above the floor in the range of 1.5m < Z < 3m. The discrepancies are found to be mainly due to uncertainties in evaluating the proper boundary conditions and to the complex flow structure established in the room.

Though the boundary conditions are measured in detail in the planes of the four supply openings, uncertainties in the proper evaluation of the direction of the jet trajectories still exist. This uncertainty is increased by the presence of flat irons below the supply openings. More detailed experimental data, involving the area downflow the supply openings, are necessary.

Despite the above mentioned difficulties experienced in the present study, the numerical simulation proves to be a valuable tool in the process of investigating such complex real flows as it is much more difficult and time consuming to carry out the corresponding experimental investigation.

ACKNOWLEDGEMENTS

The investigation was supported financially by the EU - COPERNICUS Program, Project CP 94 - 1490.

REFERENCES

Denev, J.A., P.Stankov, Y.Dinkov and D.Markov, “Three - dimensional room air flow computation: variation of buoyancy forces”, IMACS Series in Computational and Applied Mathematics, Vol. 3, Iterative Methods in Linear Algebra, II, Editors: S.D. Margenov and P.S. Vassilevski, pp.462-471, 1996.

FASTEST Manual, “FASTEST - Parallel multigrid solver for flows in complex geometries. Flow predictor description” Lehrstuhl fuer Stroemungsmechanik, Universitaet Erlangen-Nuernberg, 1993.

Hanjalic, K., S. Kenjeres and F. Durst “Numerical study of natural convection in partitioned 2-dimensional enclosures at transitional Rayleigh numbers”, Heat Transfer 1994, Proceedings of the Tenth International Heat Transfer Conference, Brighton, UK, Vol. 5, 1994

Hanjalic, K. and S. Vasic “Computation of turbulent natural convection in rectangular enclosures with an algebraic flux model” Int. J. Heat Mass Transfer 36(14)3603-3624 , 1993.

Ince, N.Z. and B.E. Launder “On the computation of buoyancy-driven turbulent flows in rectangular enclosures” Int. J. Heat and Fluid Flow, 10(2)110-117, 1989.

Leschziner, M.A. and W. Rodi “Calculation of annular and twin parallel jets using various discretization schemes and turbulence- model variations”, ASME, J. Fluids Engng., 103(2)352-360, 1981.

Peric, M. “A finite volume method for the prediction of three-dimensional fluid flow in complex ducts”, PhD Thesis, University of London, 299p., 1985.

Stankov P., J. Denev and Y. Dinkov, “Three dimensional room airflow: computation and estimation of the ventilation efficiency”, International Simposium Air Flow in Multizone Structures, Budapest, Hungary, September 9, Proceedings vol.2, 1992.

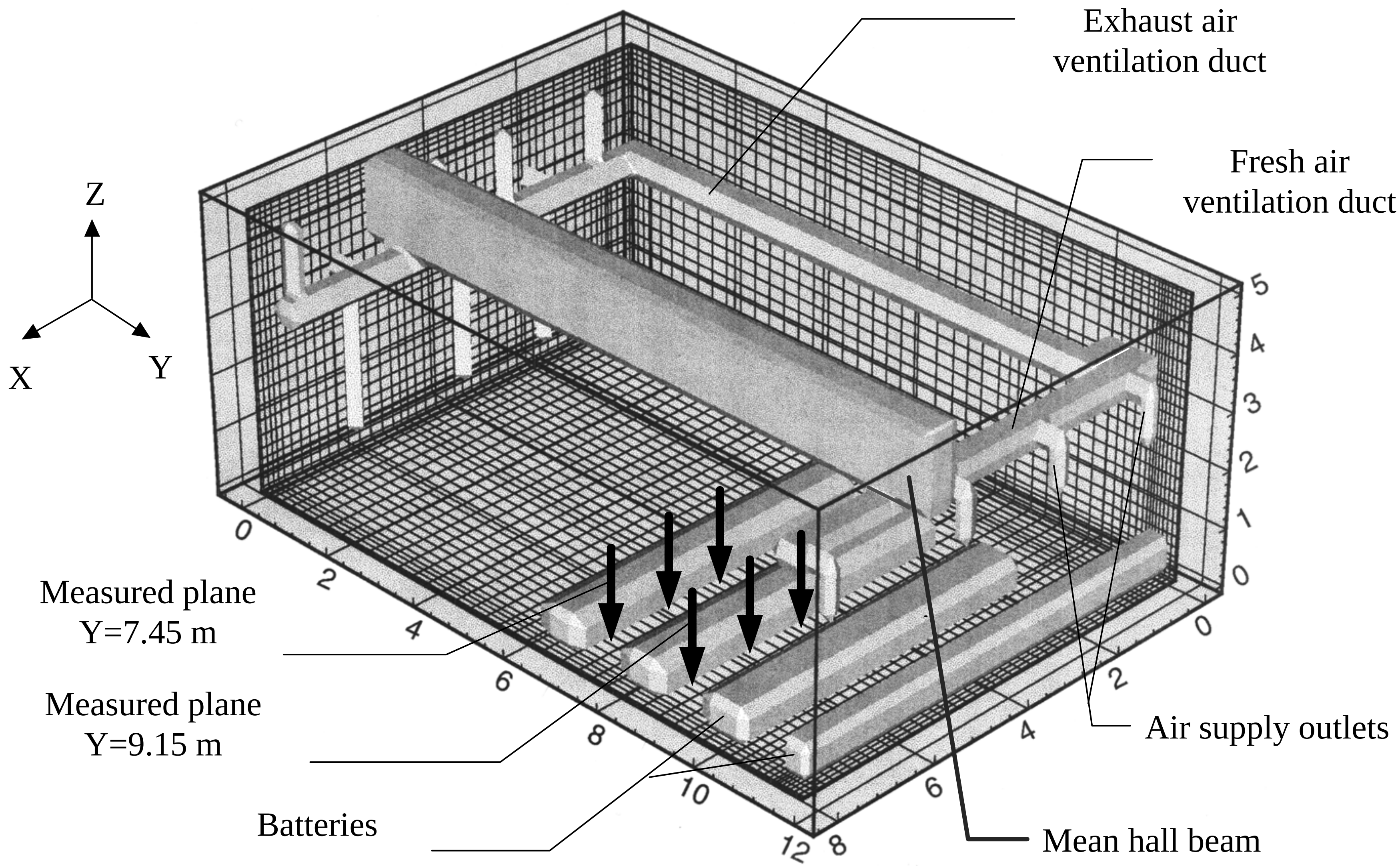

Fig. 1 The layout of the room with the numerical mesh

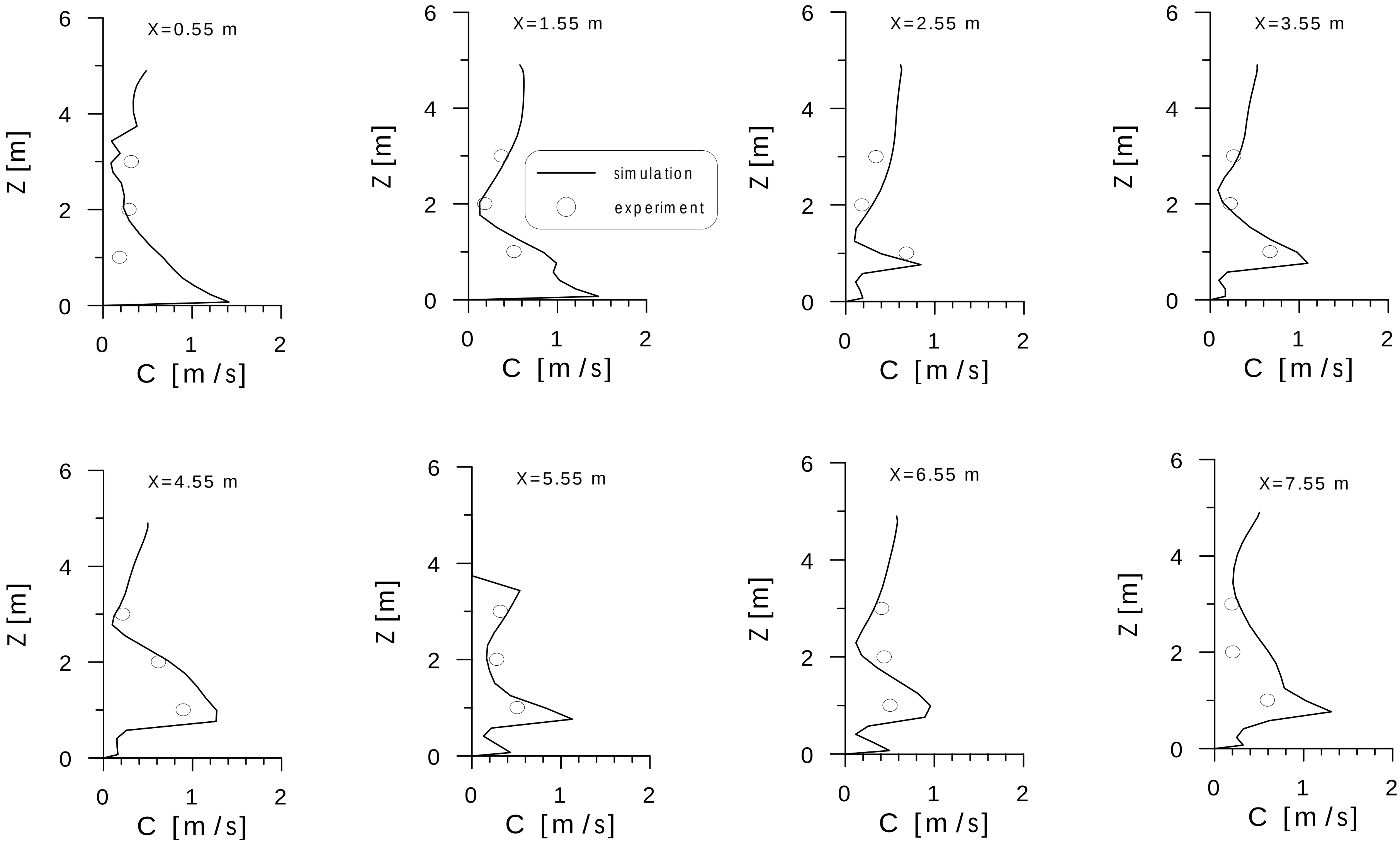

Fig. 2 The size of the velocity vectors at different locations in the plane Y=7.45 m

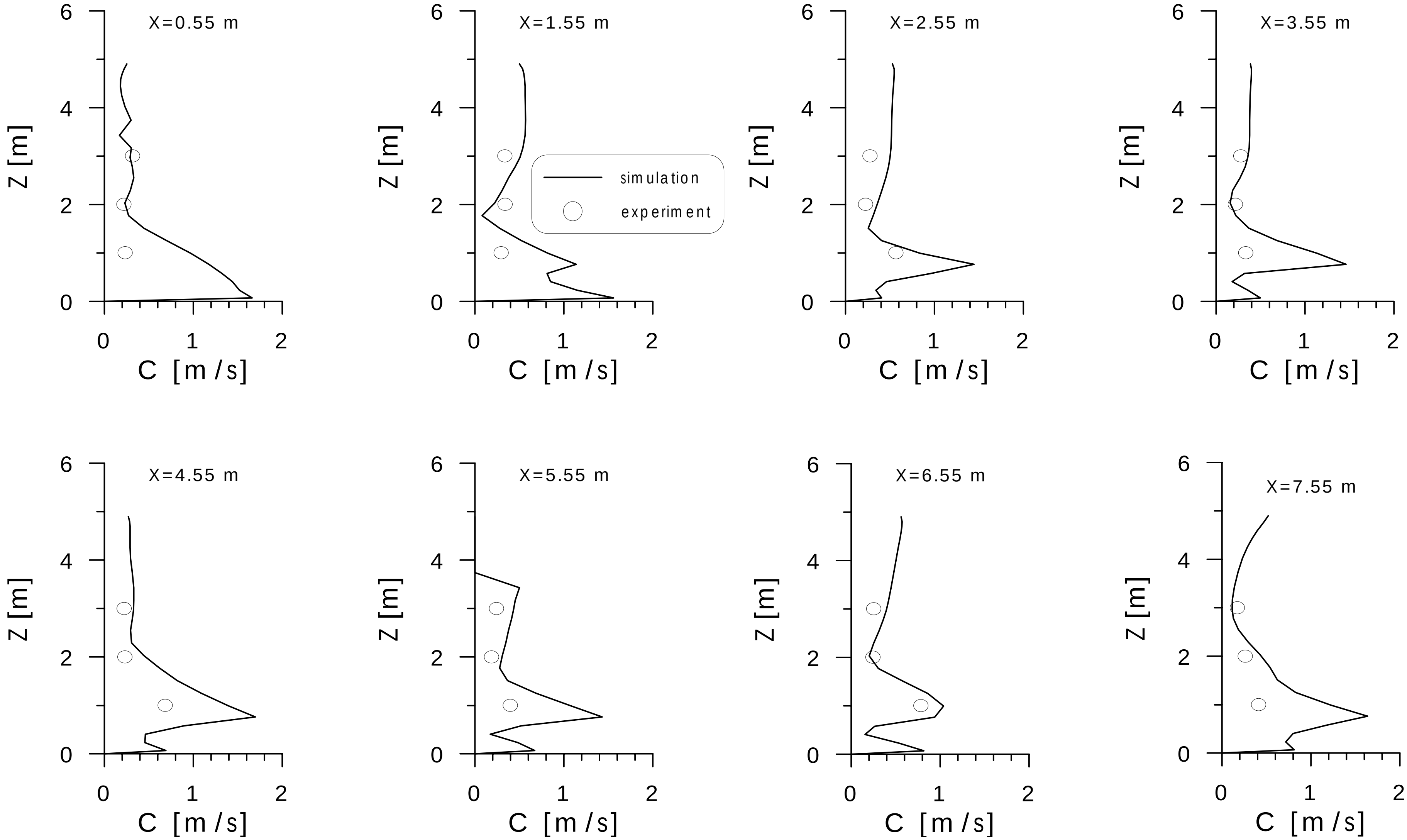

Fig. 3 The size of the velocity vectors at different locations in the plane Y=9.15 m

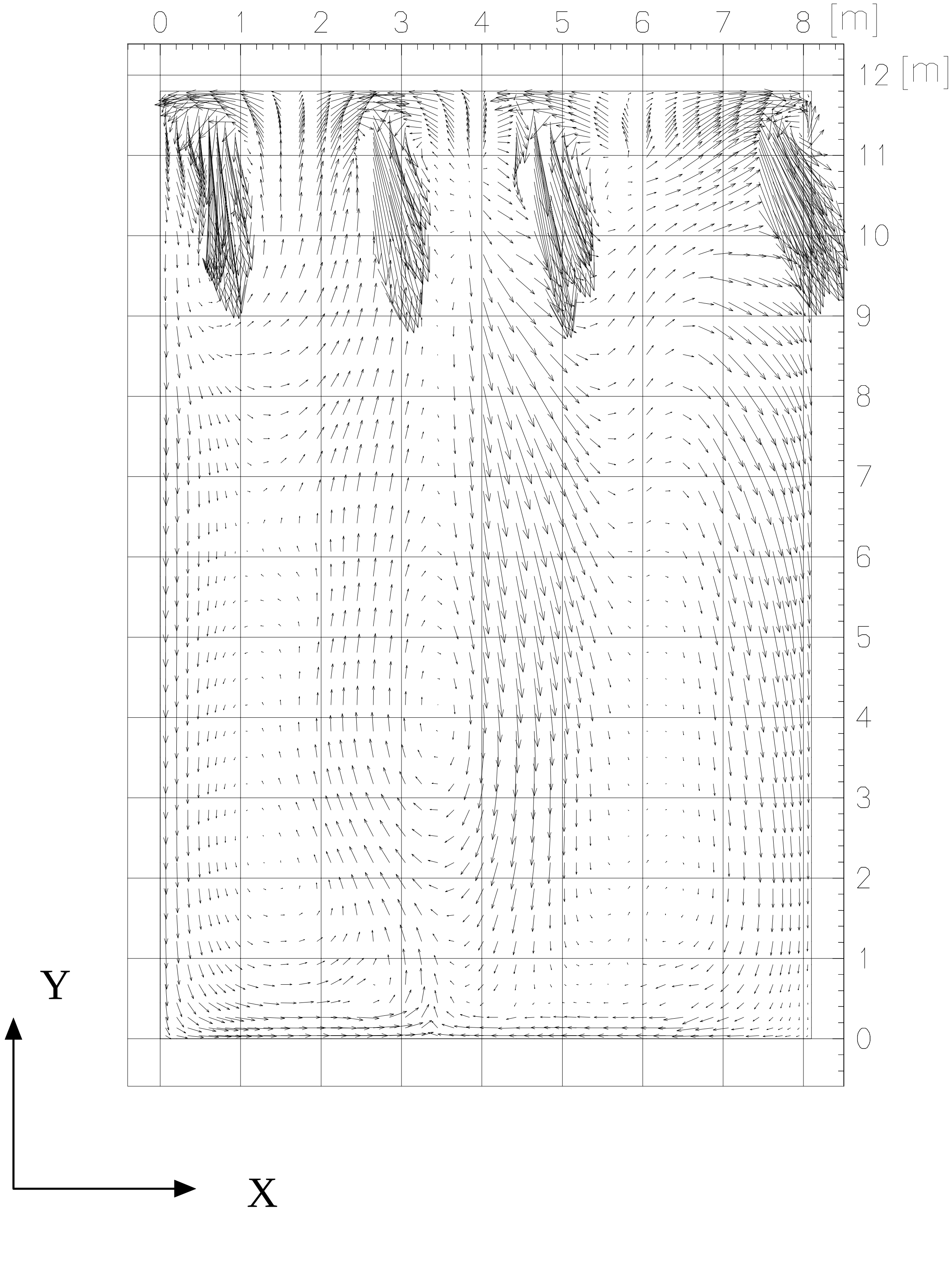

Fig. 4 Velocity vectors at a height of z = 2.0 [m]