Air Distribution in Rooms, (ROOMVENT 2000)
Editor: H.B. Awbi

13327

COMPARISON OF EXPERIMENTAL AND NUMERICAL TEST RESULTS OF THE AIRFLOW IN A ROOM WITH DISPLACEMENT VENTILATION

Z. Trzeciakiewicz, B. Lipska, Z. Popiołek, S. Mierzwiński
Silesian University of Technology, Gliwice, Poland

ABSTRACT

The paper presents a comparison between the results of experimental tests airflow pattern forming in a room with displacement ventilation and numerical calculation. The heat source in the room was a heating plate. Quasi-laminar diffusers supplied the air with the ventilation change rate from 1 to 7 h^{-1}. Temperature and velocity distributions in the plume and in its surroundings as well as the tracer gas concentrations in the background were measured. The airflow in the room was also predicted by means of CFD, using the standard k-ε turbulence model and standard log-law wall-functions. The long-wave radiation between the walls was considered. Boundary conditions in the diffusers, on the walls and those of the heat source used for numerical prediction came from experimental data. By comparing the results of measurement and calculation of non-dimensional vertical air temperature distribution, tracer gas concentration, distribution of parameters in the plume and changes in the plume velocity flux, the k-ε turbulence model applicability to displacement ventilation was verified.

KEYWORDS

Airflow, displacement ventilation, CFD, plume, temperature distribution, tracer gas

INTRODUCTION

Recently, CFD technique has been more and more often used to model airflow in ventilated rooms. One should realise that all calculation results require experimental validation, which only can confirm their reliability and the applicability of the models used for the calculation. It also refers to rooms with heat sources where popular nowadays displacement ventilation is applied. Experimental validation of different turbulence models in the case of that air distribution system modelling was described by Yuan et al. (1999) and Müller & Renz (1998): It should be expected that in a short period of time numerical technique development will make it possible for engineers to use CFD programmes in practice in order to design air distribution in ventilated rooms. At first, the simplest computer programmes VORTEX-2 comprising only the basic options of CFD modelling, including the standard turbulence k-ε model and standard log-law wall-functions will be applied. Since they comprise several simplifying assumptions, their validation by measurement is particularly necessary. Experimental verification of k-ε model in the case of two-dimensional, not fully turbulent flow was carried out by Beausolei-Morrison & Clarke (1998). The goal of the present tests was to experimentally verify the

possibility of the use of the above-mentioned model for the airflow simulation in the whole room with a heat source where displacement ventilation was applied. The simulation included determination of the position of the interface between the inflow and circulation zones, Trzeciakiewicz et al. (1999a).

MEASUREMENT STAND AND THE MEASUREMENTS

A test room of the size 3x3x3 m was used in order to compare the results of experimental tests of the airflow in the room with displacement ventilation and numerical calculation results. The test room is shown in Fig.1.

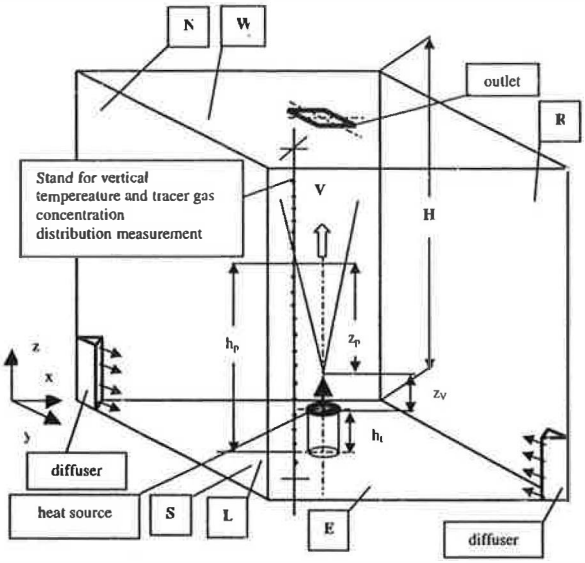

Figure 1: The test room with displacement ventilation.

The air was supplied through two quasi-laminar inlets, placed in two opposite corners of the room and removed through a ceiling outlet. The amount of the air supplied and removed corresponded to 1 to 7 h^{-1}. The heat source was a circular heating plate, 0.15 m in diameter, of the power of 600 W. The upper edge of the heat source was placed 0.5 m above the floor surface. Owing to the stationary system of thermocouples it was possible to determine the vertical temperature distribution in the plume surroundings. The movable measurement system made it possible to measure the air temperature distribution and velocity in the plume. It was also possible to measure the tracer gas concentration by means of the system at any elevation out the plume. The tracer gas was introduced to the plume just above the heat source, Trzeciakiewicz et al. (1999b).

CFD PREDICTION OF THE AIRFLOW

The airflow in the tested room was modelled numerically my means of the commercial programme using the standard k-ε turbulence model and standard log – law wall-functions. Computational Cartesian orthogonal grid was used in the numerical calculations. Owing to this, the shapes of the modelled diffusers and the heat source had to be modified to the form of a rectangle and a rectangular prism, respectively. Non uniform grid spacing was assumed, the number of nodes was 50x40x45=90000, with the grid refinement in the vicinity of ventilation openings and the heat source as well as at the wall surface. It was possible to take into account long-wave radiation between the walls of the room but without considering the heat sources. The boundary conditions on the walls were defined in the following way:

- The values of velocity, tracer gas concentration, turbulent kinetic energy and its dissipation rate were assumed equal zero,
- The temperature and emission coefficients of the wall internal surface were assumed according to the measurement data shown in Table 1.

Owing to the program limitations each of the column diffusers was reduced to the form of two rectangular openings placed in two opposite walls. The value of the airflow effective surface F_{ef}=0.0392 m^2 was maintained. The boundary conditions were defined directly in those openings. The mean velocity value V_N was determined on the basis of the momentum conservation law with the use of the measured values of the supply airflow rate. The values of temperature t_N and turbulence intensity Tu_N were determined on the basis of the measurement data too. This information is shown in Table 1.

TABLE 1
BOUNDARY CONDITIONS ON THE WALLS AND INLETS

		ε	n = 1 h^{-1}	n = 3 h^{-1}	n = 5 h^{-1}	n = 7 h^{-1}
			Inside wall temperature, °C			
walls	R	0.8	19.30	19.00	18.20	17.60
	E	0.8	20.00	19.70	18.90	18.30
	L	0.97	20.20	20.00	19.20	18.60
	W	0.97	19.55	19.25	18.45	17.85
	S	0.85	20.20	19.90	19.10	18.50
	N	0.85	23.75	23.45	22.65	22.05
inlet	velocity	m/s	0.0957	0.2871	0.4785	0.6699
	temperature	°C	17			
	turbulence intensity	%	5			

The outlet was assumed as a square 0.3x0.3 m, placed in the middle of the ceiling. The air parameters were calculated in this opening by the programme with the use of mass, momentum and energy conservation laws. The heat source i.e. the heating plate was modelled as a thin rectangular prism with a square base, placed together with the enclosure on a rectangular prism base, the height of which was the same as the one in the room modelled. The real surface of the heat exchange F = 0.0283 m^2 was maintained. When modelling the heat flow given away from the source, some problems occurred since it was not possible to separate the radiant and convective emission in the programme. On the other hand, when total heat power Q = 600 W was taken into account in the data for modelling significantly too high value of the predicted enthalpy excess in the plume was obtained when compared with the measurement data. It was thus necessary to consider the share of radiant heat in the total source emission which, as it was proved by Müller & Renz (1998), had essential effect on the final airflow pattern in the plume. The share of convective heat that ensured similar values of the plume enthalpy excess in the case of measurement and numerical calculation was evaluated with the use of the experimental data. This share was on the average 43% in the case tested, which corresponded to the heat power Q = 258 W. In addition, a point source of tracer gas emission was modelled. Its capacity was assumed 1 ml/h and it was placed in the middle of the heat source upper surface (Fig.1).

RESULTS OF THE MEASUREMENT AND PREDICTION

The numerical calculation results obtained for four air change frequencies were compared with measurement result in the room model. Fig. 2 presents the temperature excess Δt in the plume at three z_t elevations above the source level. It also shows the predicted distributions of the vertical component of the mean velocity vector v_z which are compared with the velocity value v_{zm} measured in the plume axis.

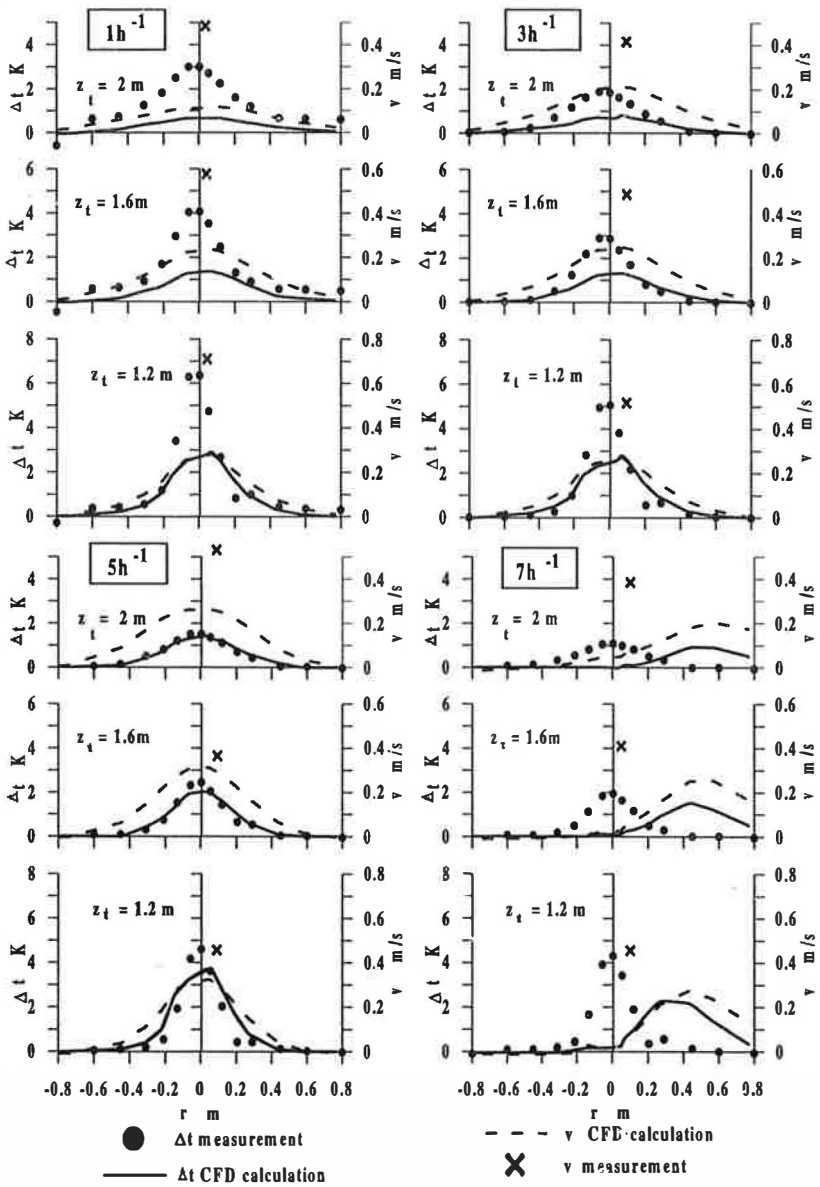

Figure 2: Measured and predicted distributions of the temperature excess and velocity in the plume cross-sections at three selected elevations above the heating plate and different air exchange rate.

Measured and calculated distributions of the air temperature simplex in the plume surroundings, defined as $(t_z-t_{inlet})/(t_{outlet}-t_{inlet})$ and the tracer gas concentration simplex defined as $(c_z-c_{min})/(c_{max}-c_{min})$ in the selected vertical axis of the room are compared in Fig. 3. A comparison was also made between the measured and predicted air volume flux in the plume $\dot{V}$. These values, shown in Fig. 4, were determined in both the cases on the basis of the air velocity in the plume axis v_{zm} and the velocity profile width, R_v. Those parameters were determined by approximating the profiles with Gaussian distribution and assuming the axial symmetry of their spatial distribution.

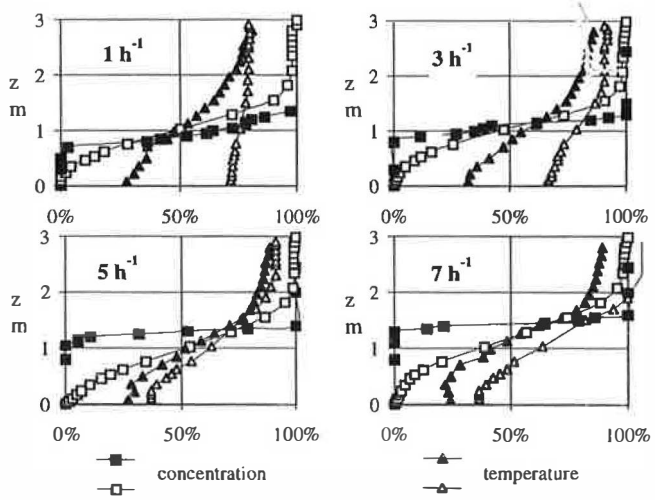

Figure 3: Results of vertical distribution measurements (filled markers) and calculations (empty markers) of non-dimensional vertical distributions of air temperature and tracer gas concentration in the plume surroundings in the room with displacement ventilation.

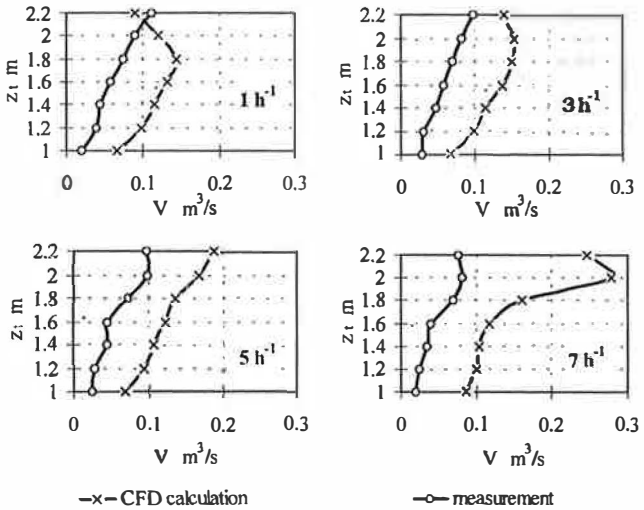

Figure 4: Air volume flux in the plume at different elevations above the floor level determined on the basis of the measured and predicted distributions of the air velocity in the plume for different values of the air change frequency.

DISCUSSION OF THE TEST RESULTS

The results of comparison between the measured and predicted values of the air parameters, prove that:

- The numerical model, based on CFD technique and comprising the standard k-ε turbulence model and standard log-law wall functions, may represent the airflow pattern in a room with displacement

ventilation, even though there are discrepancies between the calculated and measured values of the parameters in the plume and its surroundings. This conclusion is justified by the calculated temperature distributions in the plume cross-section, which can be approximated with Gaussian distribution as in a real plume. It also refers to non-dimensional vertical air temperature distribution in the plume surroundings.

- When measuring and calculating the tracer gas concentration distributions in the plume surroundings, close to the interface between the inflow and circulation zone, similar results were acquired. Thus, it made the prediction of the interface position possible.
- It was impossible to precisely determine the convective heat emission, which would produce the same enthalpy excess value in the plume as the value calculated on the basis of measurements. This value was only estimated on the basis of previous calculation and measurement by a process of trial end error. This could cause discrepancies in the temperature excess values in the plume. The calculation results were probably also affected by not taking into account as a boundary condition the radiant heat emission from the source.
- The values of the air volume flux and velocity profile width calculated on the basis of the numerical prediction are higher than the values determined when using the measurement data. This may result from the measurement method inaccuracy. Since the information about the air velocity distributions in the plume cross-section was not complete, it was assumed that the plume velocity profile width R_V was equal to the temperature width R_t. On the basis of the numerical calculation results it was found that $R_t<R_V$ by about 30% on the average. Apparently, it was not compensated by the measured air velocity values in the plume axis, higher than the calculated ones.

CONCLUSIONS

1. The numerical model using the CFD technique standard options may be used to find the qualitative airflow pattern in a room with displacement ventilation. The quantitative data, however, are not fully satisfactory in this case.
2. The numerical calculation results show that it is possible to use CFD modelling as a alternative method to predict the position of the interface between the inflow and circulation zones.
3. In order to get more precise results of the numerical calculation it is necessary to use a model in which it is possible to separately consider convective and radiant emission from the heat source and to take into account the long–wave radiation between the heat source and the walls.

ACKNOWLEDGEMENTS

The research was carried out within the Research Project No 8 T10B056 17 supported by the Polish State Committee for Scientific Research (KBN).

REFERENCES

Beausolei-Morrison I., Clarke J.(1998): The implications of using the standard k-ε turbulence model to simulate room air flows, which are not fully turbulent. Proc. Roomvent '98, vol. 1. Stockholm

Müller D., Renz U.(1998): Measurement and predictions of room airflow patterns using different turbulence models. Proc. Roomvent '98 Stockholm

Trzeciakiewicz Z., Popiołek Z., Mierzwiński S.(1999a): Displacement ventilation forming at different airflow rates. Proc. The 8th International Conference Indoor Air '99, Edinburgh, Scotland, VIII 8-13,

Trzeciakiewicz Z. et. al: (1999b): Tests of plumes in rooms with displacement ventilation. Final report of the research project 7 T07G 039 11 (in Polish)

Yuan X., Chen Q., Glicksman L.R.(1999): Measurements and Computations of Room Airflow with Displacement Ventilation. ASHRAE Transactions, Vol.105, Part 1, pp 340-350