VALIDATION OF COMPUTER FLUID DYNAMIC SIMULATION FOR DISPLACEMENT VENTILATION

Edit Stevensné Száday
Department of Building Services Engineering, Faculty of Mechanical Engineering
Budapest University of Technology and Economics (BUTE)
Műegyetem rkp. 3., Budapest, H-1111, Hungary
e-mail: szaday@rit.bme.hu

KEYWORDS
Displacement ventilation, CFD simulation, measurement, analytical method

ABSTRACT

Computer simulations are used to determine information that current design regulations do not take into account and that would be either impossible or uneconomical to discover through direct measurement. At the same time, these simulations depend on data provided by the design regulations. In this article, I describe how I used Computer Fluid Dynamics (CFD) to design a simulation of the occupied zone, compared the results with those of direct measurement, and applied an analytical method to verify the results. By entering measured values for the inlet velocity, the inlet temperature, the outlet temperature, and the radiator average surface temperature into equations and running 290 points of iteration, my method yielded field distributions of the air temperature and velocity in part of the occupied zone. The analytical method I used was based on REHVA (Federation of European Heating and Air-conditioning Associations) guidelines.

INTRODUCTION

For the user of the premises, the occupied zone is the most important element of the air conditioning system. The evaluation of the entire system depends on whether the draft and thermal comfort criteria are met there, as well as whether the contaminant requirements are satisfied. The current design regulations and standards are based on average values which do not provide information concerning the fields for air temperature, air velocity, and contaminant concentration. Therefore we cannot reach any conclusions regarding the potential recirculations and the thoroughness of the ventilation within the occupied zone. Experimental methods or computer simulations can yield such information.
Prior to the 1980's, direct measurement was the only method available to determine the fields for air temperature, air velocity, and contaminant concentration. With the arrival of Computational Fluid Dynamics codes, however, came the possibility of a comprehensive analysis of the occupied zone. To solve the partial differential equations, describing the chosen model, we need the initial and boundary conditions including, among others, supply airflow rate and temperature—both of which could be derived from the design procedure. The current design guidelines (in the absence of sufficient computer capacity and software) are still very much in practice.

MEASUREMENT SYSTEM

The measurements were taken at the Ventilation Laboratory of our department on the air distribution measuring system.
The examination chamber is inside our Laboratory so the "outside" temperature was constant during the measurements. We disabled any radiation from outside by putting shade on the windows. The measurement started after the temperature became steady which we could monitor with the help of the gradient measuring pole.
The collection of the temperature and velocity data happened in three surfaces perpendicular to the inlet face (see Fig. 1.). Data were collected on all three surfaces at 8 heights in 51 points each through a computer and evaluated with excel.
A radiator served as our heat source with 51ºC forward and 42ºC return warm water temperatures.
The measured air flow rate was $0.22 m^3/s$ which entered the chamber through a low velocity air terminal device.
The outlet was in the middle of the ceiling.

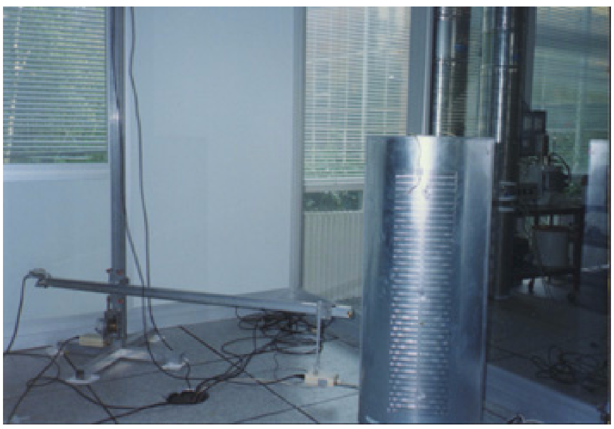

Fig. 1: Measurement setup

Proceedings 16th European Simulation Symposium
György Lipovszki, István Molnár © SCS Press, 2004
ISBN 1-56555-286-5(book) / ISBN 1-84233-106-x(CD)

COMPUTER FLUID DYNAMIC SIMULATION

The computer simulation solves a set of partial differential equations with a numerical method. These partial differential equations are the conservation of mass, momentum, energy, and contaminant concentrations.

A three-dimensional steady-state numerical simulation has been performed to examine the displacement ventilation in cooling conditions according to the measurement setup.

The simulations have been implemented using the commercial code FLUENT. A computational grid of 152460 cells has been chosen, after having verified the grid independence of the results. The reference calculation hypotheses are:

- fixed temperature boundary conditions at the radiator surface,
- other walls are adiabatic,
- standard k-ε turbulence model,
- standard wall functions,
- velocity inlet on the round face of the inlet unit
- pressure outlet on the ceiling.

As a result this method provides the field distributions of the air temperature and velocity (Fig. 2 and Fig. 3).

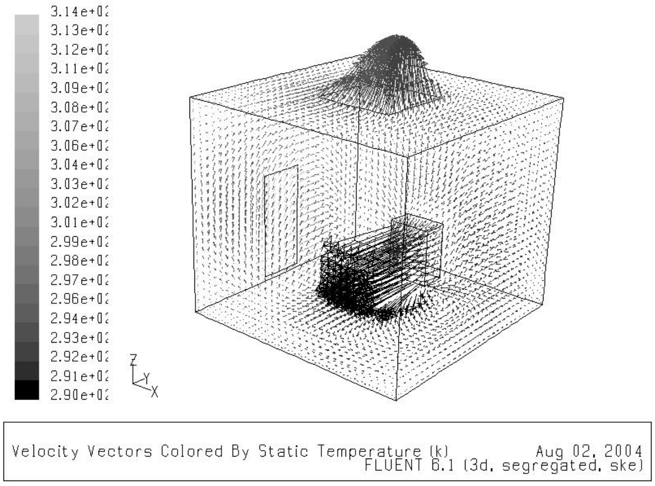

Fig. 2: Velocity vectors colored by temperature

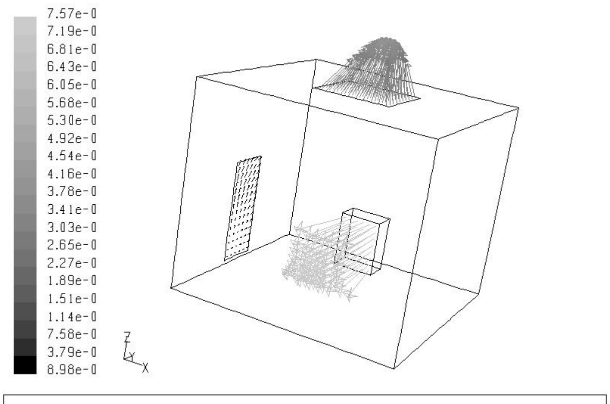

Fig. 3: Velocity vectors colored by velocity

COMPARISON BETWEEN MEASURED DATA AND SIMULATION RESULTS

With CFD simulation there are countless small details which need to be worked out in order to get the most reliable model, and thus the solution closest to the real case. The CFD simulation method provides the most detailed information about the entire room. However, validation of this method is necessary. On the other hand relying only on measurement results is not sufficient due to the multiple ways in which errors can occur (instrumental, human, recording, etc.).

There is always a question of how to compare the different results. In my work I have created the mesh for the CFD simulation but the points I got after the iteration were too many to handle. I had to create certain surfaces (on which data were collected during the measurements) in my model. (Fig. 4.)

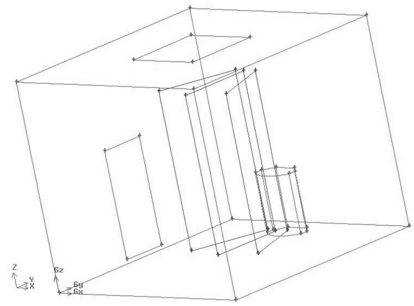

Fig. 4: Grid of the numerical method

After that I created and listed only the points of interest, where the measurements were taken with coordinates, temperature, and velocity values. Only then was I able to compare the two results and derive the necessary consequences. Fig.5 and Fig.6 show the velocities from the measurement and the simulation. As can be seen from these figures the CFD model follows the real experimental setup so the measured values are within an acceptable range.

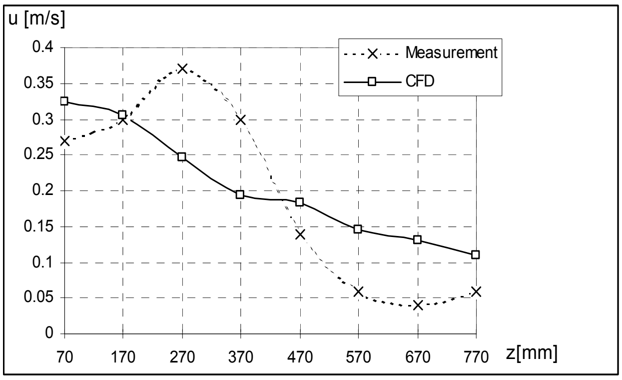

Fig. 5: Measured (dotted line) and calculated (CFD, continuous line) velocity magnitudes at different height (distance from the inlet unit was 1015mm)

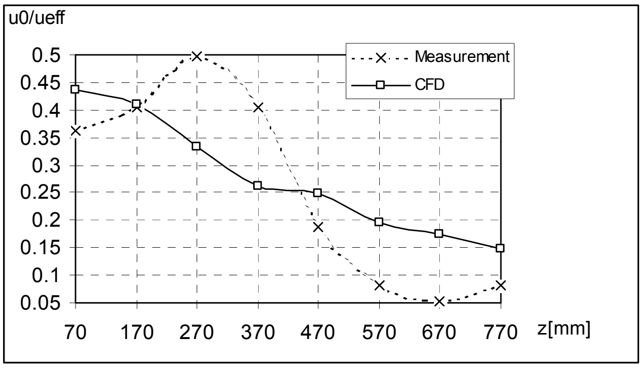

Fig. 6: Measured (dotted line) and calculated (CFD, continuous line) dimensionless velocity magnitudes at different height (distance from the inlet unit was 1015mm)

CHECKING RESULTS WITH ANALYTICAL METHOD

The design procedure used for validation was developed by REHVA.

The determination of the supply airflow rate depends on the goal of the air conditioning (or ventilation). On the basis of this the procedure distinguishes between design criteria for *contaminant stratification* and for *excess heat removal*.

As our subject is non-industrial premises, the contaminant is CO_2. The indicator for the process taking place in the room is: $\Delta h/\Delta x=+\infty$.

❖ Design criteria for contaminant stratification

1. Determination of the Input data

The input data required for both design criteria are the following:

- room size
- location and number of people, type of human activity
- location, number and specification of other heat and contaminant sources
- requirements for the occupied zone: design air temperature of the occupied zone (at theheight of 1.1 m for sedentary, 1.7 m for standing occupants), acceptable maximum air velocity near the floor, acceptable maximum contaminant concentration at inhalation level, air temperature difference between the head (1.1 or 1.7 m) and ankle level (0.1 m), and the required air flow rate.

2. Selection of stratification height

The height of the lower stratification layer must be set slightly above the height of the inhalation level.

3. The condition of contaminant stratification is that the density of the contaminant is less than the density of the air surrounding it. With the help of appropriate literature, we determine the convection air flow rate around the various heat sources at a given height on the basis of convective heat emission, location and characteristic measurements. (M is the number of the heat sources.)

4. It is necessary to ensure contaminant concentrations lower than the allowed value in the occupied zone, so the supply air flow rate ($\dot{V}_s$;[m^3/s]) above the inhalation level (1.1 or 1.7 m) must be kept in balance with the sum of ascending air flow rates from the heat sources, minus descending air flow rates from the cooler surfaces ($\sum_{i=1}^{M}\dot{V}_{conv,i}$;[m^3/s]).

$$\dot{V}_s = \sum_{i=1}^{M}\dot{V}_{conv,i} \tag{1}$$

5. Calculation of the exhaust contaminant concentration (c_e;[mg/m^3])

In light of the following conditions, the contaminant concentration of the exhaust air can be calculated with the help of equation (2).

Conditions for equation (2):

- the air-conditioning is continuous ($\dot{V}_s$=constant)
- the contaminant concentration of supply air (c_s;[mg/m^3]) is constant
- the indoor air is uniform,
- there is no local exhaust in the room ($\dot{V}_s = \dot{V}_e$),
- the source of contamination is constant ($\dot{C}$;[mg/s])

$$c_e = c_s + \frac{\dot{C}}{\dot{V}_s} \tag{2}$$

6. Evaluation of the contaminant concentration of the inhaled air (c_{exp};[mg/m^3])

As a result of human heat sources, fresh air replaces the ascending air. At the point of inhalation the air quality is higher (in our case CO_2 level is lower) than measured with no person at that point. This process can be expressed numerically with the help of the Personal Exposure Index (ε_{exp}):

$$\varepsilon_{exp} = \frac{c_e - c_s}{c_{exp} - c_s} \tag{3}$$

From equation (3) the inhalation contaminant concentration:

$$c_{exp} = \frac{1}{\varepsilon_{exp}} \cdot (c_e - c_s) + c_s \tag{4}$$

where ε_{exp} can be derived from the literature as the function of the supply air flow rate.

Another way to determine the inhalation contaminant concentration is to assume that the inhalation and the supply contaminant concentration difference is 0.5-0.7 times the exhaust and the supply contaminant concentration difference. Assuming a value of 0.5, the inhalation contaminant concentration:

$$c_{exp} = 0,5 \cdot (c_e - c_s) + c_s \tag{5}$$

In the event that the inhaled contaminant concentration determined by either equation (4) or (5) is above the acceptable level the increase of the supply air flow is necessary

❖ Design criteria for excess heat removal

1. Determination of the Input data

2. Calculation of the cooling load

The calculation of the cooling load can happen according to standards or cooling load programs.

3. Calculation of the maximum temperature increase from supply to exhaust air

The design procedure assumes constant vertical temperature gradient in the room. The temperature of the supply air along the floor increases from T_s to T_f. According to the so called "50% rule" T_f temperature is the arithmetic mean of the supply air and exhaust air temperature (T_e) (see equation 6) and can be calculated with the design air temperature (T_{oz}) and the maximum acceptable temperature gradient ($\left(\frac{\Delta T}{H}\right)_{max}$;[K/m]) (see equation 7). The air temperature rises from T_f to T_e. The desired temperature difference, then, can be calculated with the aid of the maximum acceptable temperature gradient (see equation 8).

$$T_f = \frac{T_s + T_e}{2} \tag{6}$$

$$T_f = T_{oz} - \left(\frac{\Delta T}{H}\right)_{max} \cdot z, \quad z=1,1\text{m} \tag{7}$$

$$T_e - T_s = 2 \cdot \left(\frac{\Delta T}{H}\right)_{max} \cdot H, \tag{8}$$

where H;[m] is the interior height

4. Determination of the supply and exhaust air temperature

The supply air temperature from equations 6 and 8:

$$T_s = T_f - \left(\frac{\Delta T}{H}\right)_{max} \cdot H \tag{9}$$

$$T_e = 2 \cdot \left(\frac{\Delta T}{H}\right)_{max} \cdot H + T_s \tag{10}$$

5. Determination of the supply air flow rate

The supply air flow rate can be calculated from the heat removed from the space according to equation 11.

$$\dot{V}_s = \frac{\dot{Q}_t}{\rho \cdot c_p \cdot (T_e - T_s)} \tag{11}$$

6. Recalculation of the temperature increase along the floor (T_f-T_s)

The temperature increase along the floor according to equation 12 [2,3]:

$$T_f - T_s = \frac{1}{\rho \cdot c_p \cdot \dot{V}_s \cdot \left(\frac{1}{\alpha_{rad}} + \frac{1}{\alpha_{conv}}\right) + 1} \cdot (T_e - T_s) \tag{12}$$

❖ Common last steps for both criteria

7. Verification of the calculated supply air flow rate against codes and standards

8. Selection of the supply air flow rate

The supply air flow rate for the system ($\dot{V}_{s,s}$) is chosen as the biggest among the calculated air flow rates from equations 1 and 11, and the required air flow rate according to the regulations.

9. Recalculation of the vertical temperature distribution in the room and estimation of the pollutant stratification height

The temperature difference between the exhaust and supply air can be calculated from equation 11 as follows in (13):

$$T_e - T_s = \frac{\dot{Q}_t}{\rho \cdot c_p \cdot \dot{V}_{s,s}} \tag{13}$$

$$\left(\frac{\Delta T}{H}\right) = \frac{(T_e - T_s)}{2} \cdot \frac{1}{H} \tag{14}$$

$$T_f = T_{oz} - \left(\frac{\Delta T}{H}\right) \cdot z, \quad z=1,1\text{m} \tag{15}$$

$$T_s = T_f - \left(\frac{\Delta T}{H}\right) \cdot H \tag{16}$$

$$T_e = 2 \cdot \left(\frac{\Delta T}{H}\right) \cdot H + T_s \tag{17}$$

In the event of $\left(\frac{\Delta T}{H}\right) \rangle \left(\frac{\Delta T}{H}\right)_{max}$ the increase of the supply air flow rate is necessary, with the increased air flow rate the vertical temperature distribution in the room needs to be recalculated with equations 13-17.

The pollutant stratification height can be determined with iteration from the sum of the convection air flows around the various heat sources which is balanced with the supply air flow rate for the system ($\dot{V}_{s,s}$).

10. Selection of diffusers, verification of the adjacent zones

A suitable diffuser needs to be chosen to achieve the required performance. It is strongly recommended to use diffusers from manufacturers who supply their products with reliable documentation. The calculation of the adjacent zone depends on, among others, the discharge angle and the type of the diffuser.

With the help of this method the airflow rate, the supply air temperature, and the exhaust air temperature can be calculated. The results of the calculation can be seen on Fig. 7. The inlet velocity can be computed from the airflow rate with the help of the effective area given by the inlet catalog.

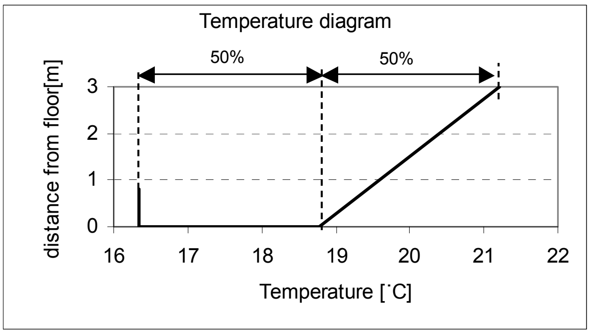

Fig. 7: Temperature diagram

CONSEQUENCES

Checking the results with an analytical method is relatively quick and it is the key in validating the CFD simulation. The measuring equipment is very expensive, and often it is not even possible to measure the desired parameters for instance when the building does not even exist yet. Doing the calculations right and comparing them with the measured data would give us a powerful tool which together with the CFD simulation creates a fast way to evaluate the desired system.

REFERENCES

Skistad, H. (1994) "Displacement Ventilation" ISBN 0 86380 147 1

Kis Piroska Z.; (1994) "Examining Ventilation of Offices" MSc Thesis, Budapest University of Technology and Economics (Hungarian)

Eberhardt P.; (1995) "Room Ventilation with Displacement Ventilation" MSc Thesis, Budapest University of Technology and Economics (Hungarian)

Mundt E.; (1995) "Displacement Ventilation Systems – Convection Flows and Temperature Gradients", Building and Environment, Vol. 30, No. 1, 129-133

Tóth A.; (1998) "Examining Occupied Zone Measurements for Displacement Ventilation" MSc Thesis, Budapest University of Technology and Economics (Hungarian)

Yuan X., Chen Q., Glicksman, L.R. (1998) "A Critical Review of Displacement Ventilation", ASHRAE Transactions, 104(1A), pp. 78-91 7.

Yuan X., Chen Q., Glicksman, L.R. (1999 A) "Models for Prediction of Temperature Difference and Ventilation Effectiveness with Displacement Ventilation", ASHRAE Transactions, 105(1), pp. 353-367

Yuan X., Chen Q., Glicksman, L.R. (1999 B) "Performance Evaluation and Development of Design Guidelines for Displacement Ventilation", ASHRAE Transactions, 105(1), pp. 298-309

Yuan X., Chen Q., Glicksman, L.R. Hu Y., Yang X. (1999 C) "Measurements and Computations of Room Airflow with Displacement Ventilation", ASHRAE Transactions, 105(1), pp. 340-352

Chen Q., et al. (1999) "Performance Evaluation and Development of Design Guidelines for Displacement Ventilation" ASHRAE Research Project –RP-949

Skistad H., Mundt E., Nielsen P. V., Hagström K., Railio J.; (2002)"Displacement ventilation in non-industrial premises" Rehva, Guidebook No 1.

Stevensné Száday E., Magyar T.; (2004) "Design Procedures for Displacement Ventilation (part 1.)", Magyar Epuletgepeszet, LIII. 2004/No.4. pp. 3-7. (Hungarian)

Stevensné Száday E.; (2004), "Design Procedure developed by REHVA for Displacement Ventilation." Conference on Mechanical Engineering 2004, Volume 2 pp. 321-325

Stevensné Száday E., Magyar T.; (2004) "Design Procedures for Displacement Ventilation (part 2.)", Magyar Epuletgepeszet, LIII. 2004/No.8. pp. 3-7. (Hungarian)

AUTHOR BIOGRAPHY

EDIT STEVENSÉ-SZÁDAY received her MSc in Mechanical Engineering at BUTE, in 1994. She has been working at the Faculty of Mechanical Engineering of BUTE since 1994, recently as an assistant researcher. Her main research interest is Displacement ventilation and simulation in which she is completing her dissertation. Her teaching areas include ventilation and air-conditioning systems.