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

INDOOR AND OUTDOOR AIRFLOW SIMULATION BY A ZERO EQUATION TURBULENCE MODEL

Bin Zhao, Xianting Li, Ying Li and Qisen Yan

Department of Thermal Engineering, Tsinghua University, Beijing, P.R. China

ABSTRACT

In order to simulate indoor air distribution and airflow around buildings quickly and accurately by CFD (Computational Fluid Dynamics) technique, a new zero-equation turbulence model and momentum method for inlet boundary condition are adopted. The new version of STACH-3, a three-dimensional CFD software is developed based on these. An example for outdoor airflow around an isolated building is given as well. For those high-density buildings with complex geometry, the TSM (Two Step Method) is proposed. Comparisons between the measured data and numerical results show that it is economic and engineering satisfied to simulate indoor and outdoor air distribution by the zero-equation turbulence model.

KEYWORDS

CFD, Turbulence Model, Inlet Boundary Condition, Momentum Method, TSM

INTRODUCTION

Nowadays it requires to simulate indoor and outdoor airflow more quickly and accurately in HVAC (Heating, Ventilating and Air Conditioning) industry. However, for most engineers who haven't large capacity computers, the $k-\varepsilon$ turbulence model for indoor and outdoor airflow simulation costs too much time, especially for those non-isothermal problems need LRN (Low-Reynolds Number) $k-\varepsilon$ turbulence model. Furthermore, the conventional method of describing supply opening for indoor airflow simulation can't introduce proper inlet boundary condition. Therefore, the new zero-equation turbulence model developed by Chen et al. (1998) is adopted to simulate indoor and outdoor airflow, with the momentum method (Chen et al., 1991) to describe the inlet boundary condition in the case of indoor airflow. As the model is developed by DNS (Directly Numerical Methods) data, it may get more accurate result. And, as consequence of not adding partial difference equation for Reynolds-Stress equations, it costs much less CPU time and computer capacity than those advanced models such as LRN $k-\varepsilon$ turbulence model.

THE ZERO-EQUATION TURBULENCE MODEL AND STACH-3

Indoor airflow is always mixed convection flow including thermal plumes, wall jets and flows with stratified temperature field. Nielsen pointed out that these flows require different turbulence models (Nielsen, 1998). To simulate natural convection and mix convection quickly and accurately, Chen et al. (1998) develop a new zero-equation turbulence model by DNS data. The model says:

$$\mu_t = 0.03874\rho V l \qquad (1)$$

where l is the distance to the nearest wall. With this, we can get the Reynolds Equations closed. All the control equations can be written in general format as following and details are listed in Table 1:

$$\frac{\partial}{\partial t}(\rho\varphi) + div(\rho \vec{u} \varphi - \Gamma_\varphi grad\varphi) = S_\varphi \qquad (2)$$

TABLE 1
THE CONTROL EQUATIONS WITH ZERO-EQUATION TURBULENCE MODEL

φ	Γ_φ	S_φ
1	0	0
u	μ_{eff}	$-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}(\mu_{eff}\frac{\partial u}{\partial x})+\frac{\partial}{\partial y}(\mu_{eff}\frac{\partial v}{\partial x})+\frac{\partial}{\partial z}(\mu_{eff}\frac{\partial w}{\partial x})+g_x(\rho-\rho_{ref})$
v	μ_{eff}	$-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}(\mu_{eff}\frac{\partial u}{\partial y})+\frac{\partial}{\partial y}(\mu_{eff}\frac{\partial v}{\partial y})+\frac{\partial}{\partial z}(\mu_{eff}\frac{\partial w}{\partial y})+g_y(\rho-\rho_{ref})$
w	μ_{eff}	$-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}(\mu_{eff}\frac{\partial u}{\partial z})+\frac{\partial}{\partial y}(\mu_{eff}\frac{\partial v}{\partial z})+\frac{\partial}{\partial z}(\mu_{eff}\frac{\partial w}{\partial z})+g_z(\rho-\rho_{ref})$
h	$\frac{\mu_{eff}}{\sigma_h}$	S_h σ_h:1.0
$\mu_{eff}=\mu_l+\mu_t$ $\mu_t = 0.03874\rho V l$		

Based on these equations, the new version of STACH-3 is developed. The finite volume method and SIMPLE algorithm are adopted in the software. The boundary condition of walls is treated as source term of momentum equations and energy equation by wall function. (Zhao et al., 1999(b))

INDOOR AIRFLOW

Momentum Method for Inlet Boundary Condition Description

To introduce the correct momentum flow of inlets into the room, the momentum method (Chen et al., 1991) is used for indoor airflow. That is:

$$Jin = m\frac{L}{Ae} = m\cdot\frac{L}{A}\div\frac{Ae}{A} = m\cdot\frac{L}{A}\div\frac{Ae}{A} = m\cdot\frac{L}{A}\div f \qquad (3)$$

where f is the ratio of effective area Ae to gross area A of the supply openings or diffusers, Jin is inlet momentum flow rate, m and L are inlet mass and volume flow rate respectively. Emvin et al. (1996) pointed out that the momentum method is not proper for refined grids. But to the new zero-equation model and engineering application, the grids are coarse enough to get less error.

Non-isothermal Ventilation

To compare with the measured data, we simulate the non-isothermal ventilation case in which the experiment was made by H. B. AWBI (AWBI, 1989) with STACH-3. The test room has a square floor of length 4.2m and height 2.8m. The air is supplied from a 24mm continuous slot diffuser in the ceiling spanning the width of the room and at a distance 1.2m from the wall. The room load was produced by electrically heated tapes laid over the floor area. So it's a 2-D case with uniform load distribution. Figurel shows the velocity and temperature distribution in the occupied zone of the room for 60 l/s/m flow rate. The numerical results agree very well with the measured data, where the maximal difference of velocity and temperature is only 0.08m/s and 0.8° C respectively. The mesh for the case is 28• 17• 3, which only costs 5 minutes to get convergence on a PIII 450 personal computer with 128M RAM. Therefore, it's cheap and quickly to simulate many cases in designing process for engineering application by STACH-3.

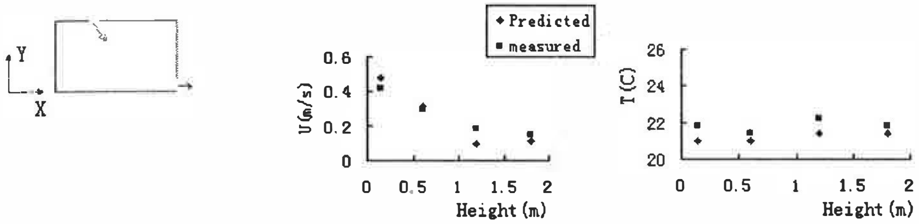

Fig.1 Mean velocity and temperature in the occupied zone

Displacement Ventilation

A displacement ventilation case is also simulated by STACH-3. The measured data is from the experiment of Yuan et al. (1998). There are 1 supply diffuser, 1 exhaust, 2 occupants, 2 computers, 2 tables, 2 boxes, and 6 lamps in the room. Figure 2 shows the comparison between the measured and calculated airflow pattern at Z=1.8m, while Figure 3 illustrates the calculated mean velocity and temperature VS. measured data at X=0.78m, Z=1.83m. They both show good agreement between the numerical and experimental results. The maximal difference of mean velocity and temperature is only 0.05m/s and 1.5° C respectively. For the mesh of 26• 26• 24, it costs about 12 hours to get convergence on the computer mentioned before. More comparisons at other place show that it is really engineering satisfied to simulate the problem by the new zero-equation turbulence model with momentum method to describe the inlet boundary condition.

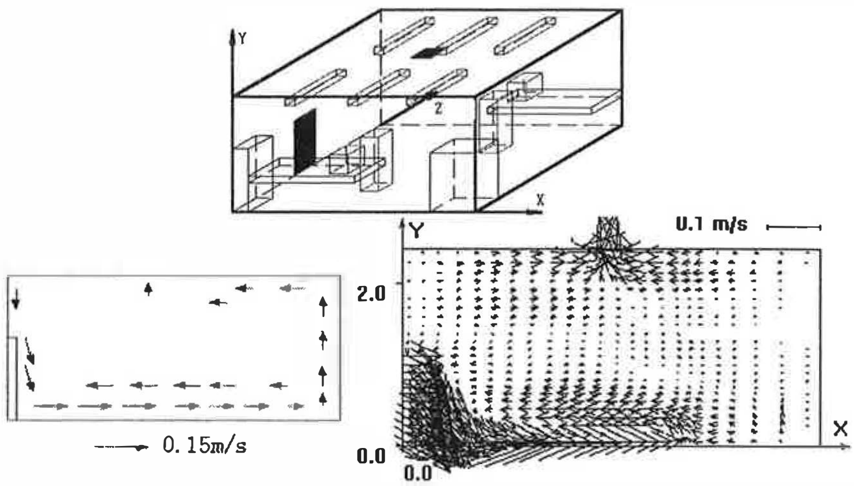

Fig.2 Airflow pattern at Z=1.8m (left: measured, right: calculated)

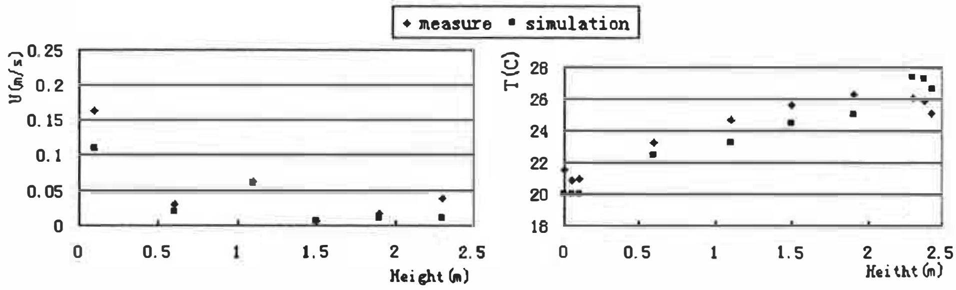

Fig. 3 Mean velocity and temperature distribution at X=0.78m, Z=1.78m

OUTDOOR AIRFLOW

Airflow around an Isolated Building

Airflow around buildings is also important for HVAC engineers. Here we use STACH-3 to simulate the airflow around an isolated building to validate the zero-equation turbulence model for outdoor airflow. The case is experimented by SHUZO et al. (1988) in wind tunnel. Figure 4, 5 shows the comparison between measured and simulated airflow pattern of the vertical and horizontal section at the middle of the building, respectively.

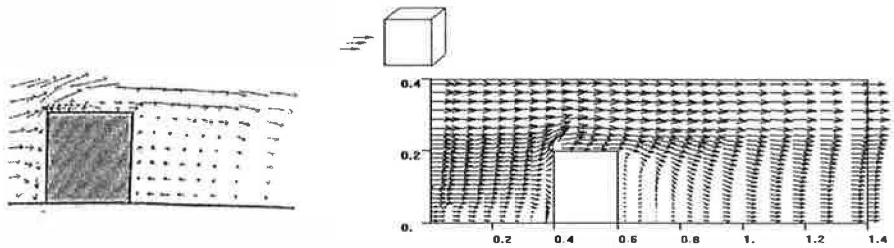

Fig.4 Airflow of the vertical section at the middle of the building (left: measured, right: simulated)

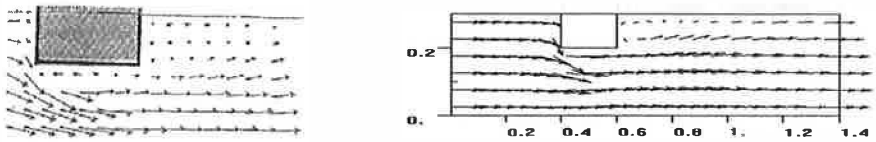

Fig.5 Airflow of the horizontal section at the middle of the building (l: measured, r: simulated)

The figures show that the zero-equation turbulence model can simulate the stagnation point at about 2/3 of the building's height and circumfluence behind the building rightly. On the same computer, it costs only 1 hours to get convergence for a mesh of 26• 28• 8.

TSM for High-Density Buildings with Complex Geometry

For those high-density building clusters, especially buildings with complicated geometry, a new method called TSM (Two Step Method, Zhao et al., 1999(a)) can be used. In TSM, the computational process is divided into two orders. In the first order, the chosen computational domain is large enough, maybe several times larger than the building area. Thus the boundary condition is easy to describe, as the buildings have no influence on it. Complicated-geometry buildings are simplified as regular blocks. In the second order, the computational domain is just the area we concerned. The boundary condition is provided by the results of the first order with interpolation method. Figure 6 shows the process. Detailed information about TSM can be found in reference (Zhao et al., 1999(b)).

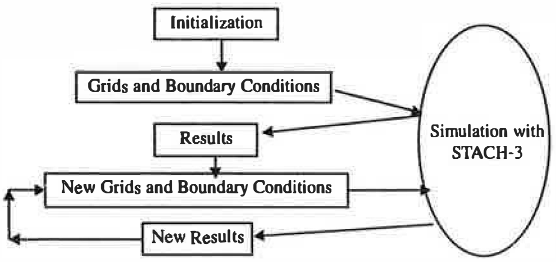

Fig.6 Flow sheet of TSM method

CONCLUSION

Non-isothermal and displacement ventilation simulations were made with STACH-3 which is based on the zero-equation turbulence model and momentum method of inlet boundary condition description. The results agree well with the measured data. The same thing happens to outdoor airflow simulation. The conclusion that can be drawn is that it is effective and economic to simulate these problems by the zero-equation turbulence model. It takes rather short CPU time to get engineering satisfied results.

References

AWBI H.B. (1989). Application of Computational Fluid Dynamics in Room Ventilation, *Building and Environment*, **24:1**, 73-84

Chen Qingyan and Alfred Moser. (1991). Simulation of A Multiple-Nozzle Diffuser, *Proceedings of the 12th AIVC Conference on Air Movement and Ventilation Control within buildings*, **2**, 1-13

Chen Qingyan and Xu Weiran. (1998). A Zero-equation turbulence model for indoor air flow simulation, *Energy and Building* **28**, 137-144

Emvin Peter and Davidson Lars. (1996). A Numerical Comparison of Three Inlet Approximations of the Diffuser in Case E1 Annex20, *ROOMVENT'96*, 222

MURAKAMI, SHUZO and AKASHI MOCHIDA. (1988). 3-D Numerical Simulation of Airflow around a Cubic Model by Means of the k-ε Model. *Journal of Wind Engineering and Industrial Aerodynamics*, **30**, 283-303

Nielsen, P. V. (1998). The Selection of Turbulence Models for Prediction of Room Airflow, *ASHRAE Transactions*, **104:1,** 1119-1127

Yuan Xiaoxiong, Chen Qingyan, and Glicksman R.. (1998). Performance Evaluation and Development of Design Guidelines for Displacement Ventilation, *Final Report to ASHRAE TC 5.3,* 151-154

Zhao Bin, Li Xianting, and Li Ying (1999(b)). The manual of STACH-3 (new version), *Inner Report of Dept. Thermal Engineering*, Tsinghua University, P.R. China, 1-10

Zhao Bin, Li Ying, Li Xianting and Yan Qisen (1999(a)). Numerical Analysis of Wind Effect on High-Density Buildings with TSM, *Proceedings of Building Simulation'99*, **2**, 823-831