한국대기환경학회 2000춘계학술대회 논문집
Proceedings of 30th Meeting of KOSAE (2000) pp. 162-164
Proceedings of Korean Society for Atmospheric Environment

DR6) EFFECT OF TURBULENCE AT INLET BOUNDARY ON AIR MOVEMENT IN A ROOM

Heekwan Lee and Hazim B. Awbi
Built Environmental Engineering, University of Reading
Whiteknights, PO Box 219, Reading RG6 6AW, UK

INTRODUCTION

The numerical simulation of air movement in a room using CFD (Computational Fluid Dynamics) requires a complicated set of input data. This includes physical data, such as space geometry, characteristics of supply air flow and contaminant source, etc. as well as computational domain. Among the input data, the boundary conditions related to the inlet are particularly crucial in order to achieve accurate computation results, although there are many other parameters which may also affect the results. The *turbulence intensity* of the supply air is one of the parameters that need to be specified in CFD simulations. The *turbulence intensity* may not only influence the air movement in a room but also affect the occupants comfort. In this study, the *turbulence intensity* of the supply air is varied in CFD simulations to determine its effect on the air movement in a room.

TURBULENCE IN ROOM AIRFLOW

In CFD simulation of room air flow, the turbulence intensity at the inlet boundary is one of the paramteters to be speficied. This turbulence intensity (I) is defined by Equation (1) and indicates the degree of turbulence fluctuation. Combining Equations (1) and (2) under the assumption of isotropic flow in Equation (3) gives Equation (4) which indicates the amount of kinetic energy (k) in a turbulent air flow determined by its velocity (U) and turbulence intensity. This kinetic energy is in turn used to calculate the turbulence energy dissipation (ε) in Equation (5), see Awbi (1991) for more reading.

$$I_u = \frac{\overline{u'}}{\overline{u}} \quad (1)$$

$$k = 0.5[(\overline{u'})^2 + (\overline{v'})^2 + (\overline{w'})^2] \quad (2)$$

$$\overline{u'} = \overline{v'} = \overline{w'} \quad (3)$$

$$k = \frac{3}{2} I_u^2 U^2 \quad (4)$$

$$\varepsilon = \frac{k^{1.5}}{\lambda H} \quad (5)$$

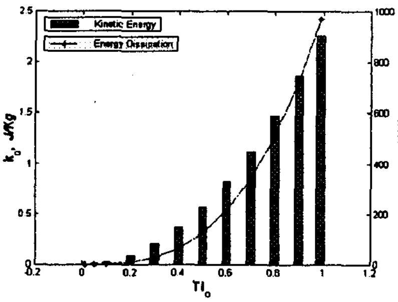

Figure 1. Turbulence intensity vs. its energy.

Figure 1 depicts the changes in turbulence energy by different turbulence intensity. It implies that increasing the turbulence intensity means adding more turbulence energy onto the airflow concerned and increasing its turbulence energy dissipation.

MODEL ROOM AND CFD SIMULATION

Model room conditions are used for CFD simulations to study the effect of turbulence intensity at the inlet boundary on the air movement in the test room. The model room (1.6*m*L by 0.8*m*D by 0.7*m*H) has two openings for ventilation inlet and outlet at ceiling level and is mechanically ventilated in mixing mode. The contaminant source is placed at the mid-height under the inlet opening. The *Reynolds Number* at the inlet opening is 9200 and this is kept constant for all the cases simulated, see Lee and Awbi (2000) for more detail of test conditions. A CFD code, *VORTEX*, is used for the simulations and the results are analyzed and visualized by using *VORVIS* code developed by the author.

RESULTS AND DISCUSSION

Figure 2 shows the simulated velocity profiles with different turbulence intensity at the inlet boundary. The air jet with lower intensity reaches to further inside of the test room compared to others with higher intensity. It means that the momentum of air at the inlet boundary is attenuated depending on its turbulence energy and higher turbulence energy spreads the air jet more quickly. Figure 3 depicts this phenomenon in terms of the potential core length, which indicates the distance from the inlet boundary for 90% of the inlet velocity in this study, of the air jet. It reveals that the intensity tested affects the spread of inlet air jet considerably.

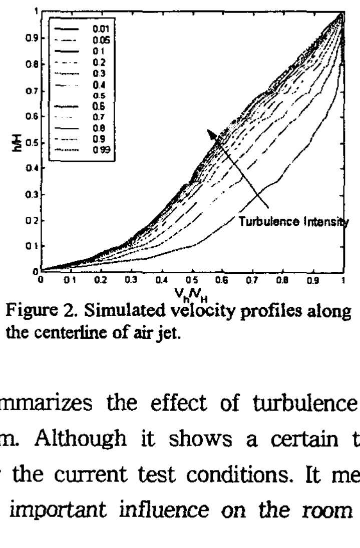

Figure 2. Simulated velocity profiles along the centerline of air jet.

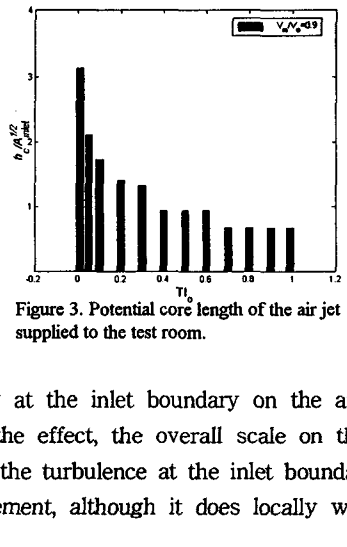

Figure 3. Potential core length of the air jet supplied to the test room.

Figure 4 summarizes the effect of turbulence intensity at the inlet boundary on the air change in the test room. Although it shows a certain trend of the effect, the overall scale on the y-axis is ignorable for the current test conditions. It means that the turbulence at the inlet boundary does not produce any important influence on the room air movement, although it does locally within the air jet.

Besides considering indoor air quality, the comfort for occupants is another important factor to provide better indoor environment. The concept of *percentage of dissatisfied* $(PD)^2$ in Equation (6) is introduced to see the effect. Although the results are not presented in this manuscript, no significant effect is observed from the results.

$$PD = (34 - T_a)(\bar{v} - 0.05)^{0.62}(0.37\bar{v}I + 3.14) \qquad (16)$$

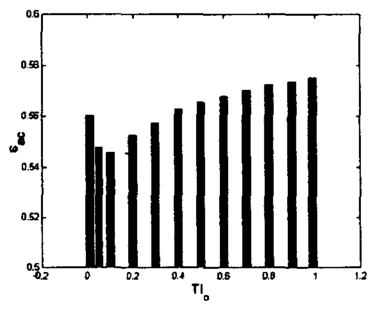

Figure 4. Air change in the test room due to different turbulence intensity at the inlet boundary.

SUMMARY

The effect of turbulence intensity at the inlet boundary on the room air movement is studied by carrying out CFD simulations. These are the major findings in this study:

- The turbulence intensity at the inlet boundary influences the spread of air jet considerably such that the jet with higher intensity attenuates quickly, e.g. decay of maximum velocity and potential core length.
- The air movement and contaminant spread is affected in minor degree by the inlet air with different turbulence intensity.
- The turbulence intensity at the boundary has minor effect on the percentage of dissatisfied (PD) in the test room, although in isothermal condition.

ACKNOWLEDGEMENT

This project is supported by the University of Reading, U.K., in part.

REFERENCES

1. Awbi, H B (1991) *Ventilation of Buildings*, E&FN Spon, London.
2. ANSI/ASHRAE Standard 55-1992, *Thermal Environmental Conditions for Human Occupancy*, ASHRAE, Atlanta.
3. Lee, H and Awbi, H B (2000) Effect of internal partitioning on indoor air quality in a room, *Proc. KOSAE Spring Conference*.