THE 5TH ASIAN COMPUTATIONAL FLUID DYNAMICS
BUSAN, KOREA, OCTOBER 27~30, 2003

A Numerical Investigation of Indoor Air Quality with CFD

V. K. Sin [1], *H. I. Sun* [2]

1. *Department of Electromechanical Engineering, University of Macau, vksin@umac.mo*
2. *Department of Electromechanical Engineering, University of Macau, ma26260@umac.mo*

Corresponding author ***V. K. Sin***

Abstract

Increasing interest in indoor air quality (IAQ) control has been found because of its serious effect on human health. To evaluate IAQ, thermal comfort in terms of temperature and velocity distributions of indoor air has to be analyzed in detail. Choice of location for installation of air-conditioner in a building will affect the performance of cooling effect and thermal comfort on the occupants, which in turn will affect the indoor air quality (IAQ) of the building. In this paper, we present a discussion on the proper location of the air-conditioner in order to obtain good thermal comfort for occupant of a typical bedroom in Macao. A set of carefully designed numerical experiments is run with the Computational Fluid Dynamics (CFD) software FLOVENT 3.2 [1]. Reynolds averaged Navier-Stokes equations are solved with finite volume technique and turbulence effects upon the mean flow characteristics is modeled with the $k-\varepsilon$ model. Assumption of steady state environment is made and only convective and conductive heat transfer from the occupant and air-conditioner are being concerned.

Keywords: *Indoor Air Quality (IAQ), Computational Fluid Dynamics (CFD), Turbulence.*

1. Introduction

A single bedroom of dimensions of 3.03m×3.33m×2.5m as shown in Figure 1 is being simulated. Two big glass windows with the U (overall heat transfer coefficient) of 5.9W/m^2 K are settled at the opposite side of the door. The initial indoor temperature is set as 28°C and the temperature outside the glass windows is considered as the outdoor air with 33°C. Except the windows, the room is bounded by wall with the U value of 2.35 W/m^2 K. An occupant who generates metabolic heat of 70W [2] is sleeping on the bed and the air-conditioner which supplies air with 16°C and flow rate of 200m^3/hr with 10% fresh air and 90% return air is located at the wall which is opposite to the occupant. During the simulation, a total number of 38850 non-uniform grids are used so that grid distribution around the occupant and the air-conditioner is fine enough to assure accurate numerical results.

2. Results

Temperature and velocity contours at a particular plan (z=1.52m which is at the location where the occupant sleeps) is shown in Figure 2 with air-conditioner located on the opposite side of the occupant. It is noted that calculated value of temperature and velocity are about 21-22°C and 0.1-0.2m/s, respectively, which are within the comfort values for indoor environment [3]. Optimal location of air-conditioner can be determined through the analyses of results of numerical experiments. The ability of the above-mentioned CFD software to predict flow characteristics is demonstrated and results obtained in the paper can provide information for solving the IAQ problem.

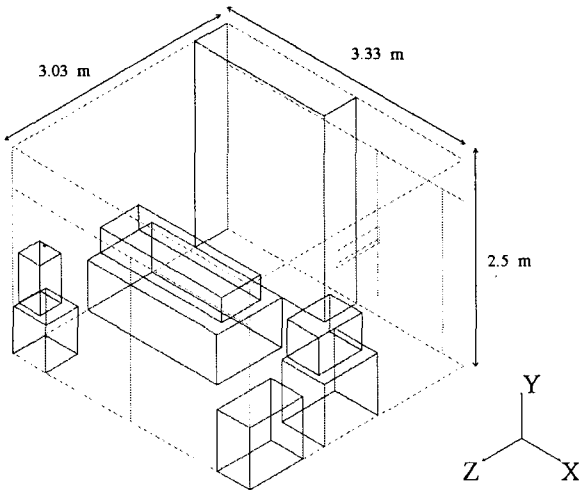

Fig. 1. Layout of the simulated room

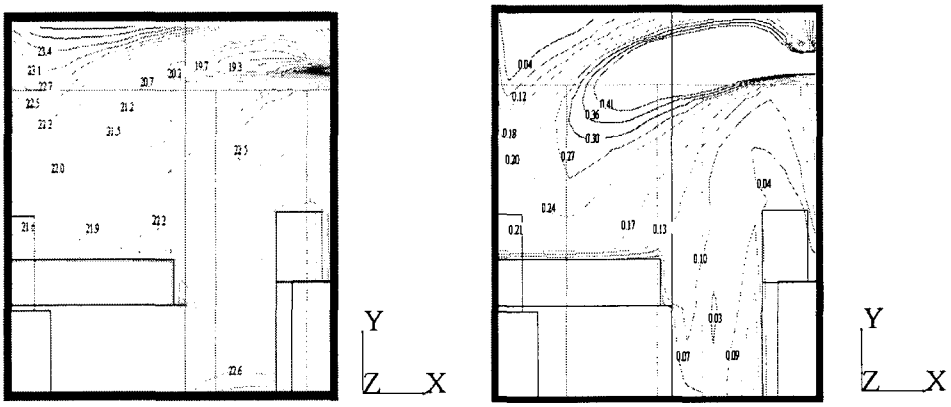

Fig. 2. Contours of the temperature (left) and velocity (right) of the cross section plane at the location of the occupant.

References

[1] Anonymous, Finite volume equations, FLOVENT/OL/MM/0801/1/0, Flomerics Ltd., August (2001).

[2] Wilbert F. Stocker and Jerold W. Jones, "*Refrigeration & Air Conditioning*", McGraw-Hill Book Company, (1982).

[3] Anonymous, Guidance Notes for the Management of Indoor Air Quality in Offices and Public Places(Draft), Hong Kong Indoor Air Quality Management Group, November (1999).