PAMUKKALE ÜNİVERSİTESİ MÜHENDİSLİK FAKÜLTESİ
PAMUKKALE UNIVERSITY ENGINEERING COLLEGE
MÜHENDİSLİK BİLİMLERİ DERGİSİ
JOURNAL OF ENGINEERING SCIENCES
YIL : 2006
CİLT : 12
SAYI : 1
SAYFA : 87-95

NUMERICAL INVESTIGATION OF AIR FLOW INSIDE AN OFFICE ROOM UNDER VARIOUS VENTILATION CONDITIONS

Şenol BAŞKAYA, Emre EKEN
Gazi University, Faculty of Engineering and Architecture, Department of Mec. Eng., 06570/Ankara

Geliş Tarihi : 13.08.2004

ABSTRACT

Air flow characteristics inside an office room, containing one person and office furniture, was investigated numerically under various ventilation conditions. Computational Fluid Dynamics (CFD) was used for the solution of the steady two-dimensional conservation equations. Results were presented in the form of velocity vectors and temperature contours together with quantitative velocity and temperature distributions. Effects due to the occupants, inlet/outlet locations, inlet velocity, and winter/summer conditions on the airflow were examined. From the present numerical predictions it can be shown that occupants significantly alter the indoor air movement and hence affect comfort conditions.

Key Words : Ventilation, Air flow, CFD, Room

BİR OFİS ODASI İÇİNDEKİ HAVA AKIŞININ DEĞİŞİK HAVALANDIRMA ŞARTLARI ALTINDA SAYISAL OLARAK İNCELENMESİ

ÖZET

İçinde bir insan ve ofis mobilyaları bulunan bir ofis odası içindeki hava akış karakteristikleri değişik havalandırma şartları altında sayısal olarak araştırılmıştır. Zamandan bağımsız, iki boyutlu korunum denklemleri sayısal akışkanlar dinamiği (SAD) kullanılarak çözülmüştür. Sonuçlar hız vektör dağılımları ve sıcaklık konturlarına ilave olarak niceliksel hız ve sıcaklık dağılımları olarak sunulmuştur. Oda içindeki nesneler, giriş/çıkış konumları, giriş hızı ve kış/yaz şartlarının hava dağılımına etkileri incelenmiştir. Mevcut sayısal tahminlerden oda içinde bulunan nesnelerin önemli derecede iç hava hareketlerini değiştirdiği ve bunun neticesinde komfor şartlarını etkilediği gösterilebilir.

Anahtar Kelimeler : Havalandırma, Hava akışı, SAD, Oda

1. INTRODUCTION

In order to obtain maximum comfort, it is necessary to equip offices with mechanical ventilation and air conditioning systems. Because supply air properties and its distribution play a vital role in the determination of room conditions, correct estimation of the air distribution inside the occupied space is vital for the design of an affective air distribution system. Airflow patterns can be predicted experimentally or by using numerical simulation. Since the 1970’s, Computational Fluid Dynamics (CFD) has been a reliable tool for the estimation of indoor air movement and indoor thermal comfort evaluation.

Numerous studies can be found in the literature regarding indoor air distributions. A brief presentation of these investigations relevant to the present study is given below. Thermal comfort analysis of a room with a cooled roof was made by Niu and Kooi (1994). Yamamoto et al. (1994) investigated the air distribution inside an enclosure with an inlet from the ceiling and an outlet from the bottom wall. Chow and Wong (1999) presented air velocity data collected from occupied waiting halls of seven train stations, which were mechanically ventilated. A two-dimensional k-ε turbulence model was used by Xue and Shu (1999) for estimation of air velocity, temperature, and turbulent kinetic energy distributions inside a room with an air inlet from the ceiling. Different opening configurations of a room were studied using CFD by Ayad (1999). The mixed convection airflow obtained by two wall jets at different temperatures was investigated numerically and experimentally by Costa et al. (1999). Natural convection from heated room surfaces was studied experimentally by Awbi and Hatton (2000). Gan (2000) developed a numerical method for the determination of the effective depth of fresh air distribution in rooms. Velocity and temperature distributions were investigated by Sinha et al. (2000) inside a heated room for various different inlet and outlet locations. A CFD computer program was used by Lam and Chan (2001) for the investigation of velocity and temperature distributions inside a gymnasium with one cold air supply and four different exit locations. A heated mannequin, desk, and other heat sources were used by Xing et al. (2001), in their experimental measurements inside a ventilated room. They also performed numerical simulations on the same configuration. As can be seen, research in the literature has been mainly on empty spaces, situations with occupants have not attracted much attention.

2. NUMERICAL MODEL

Figure 1 shows a two-dimensional schematic drawing of the office room investigated. All necessary information regarding the room is clearly indicated on this drawing.

As can be seen from this figure the office room is equipped with standard office material. This has been kept to a minimum with one person, a chair, a desk, and a shelf. Numerous different alternatives are possible, however, the choice made is sufficient for the purpose of this investigation.

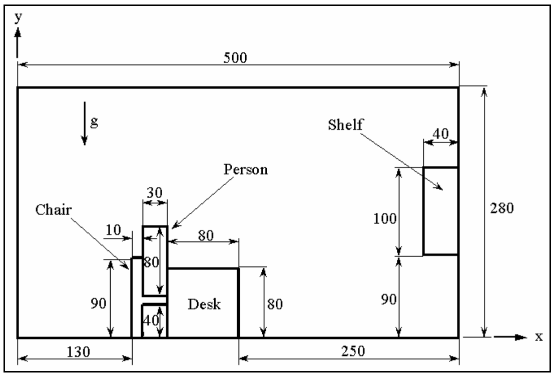


Figure 1. Schematic illustration of the room (measures in cm).

2. 1. Governing Equations

Airflow and heat transfer inside the office room is assumed to be, steady-state, two-dimensional, turbulent, and a mixed convection problem. Hence, the problem was formulated with two-dimensional equations of conservation of mass, momentum, energy, turbulent kinetic energy and its dissipation rate. For a steady, incompressible, two-dimensional flow, the conservation of mass can be expressed in the following form.

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

The turbulent momentum equation can be formulated as given below.

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial P}{\partial x_i} - \frac{\partial}{\partial x_j} \left[\mu_{ef} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - g_i (\rho - \rho_{ref}) \tag{2}$$

Here, g is the gravitational acceleration, ρ_{ref} is the reference density, μ_{ef} is the effective dynamic viscosity. In the above equation, the $g(\rho - \rho_{ref})$ term is the buoyancy force.

The turbulent energy equation can be formulated as:

$$\frac{\partial}{\partial x_j}(\rho u_j T) = \frac{\partial}{\partial x_j}\left(\Gamma_{ef} \frac{\partial T}{\partial x_j} \right) \tag{3}$$

The turbulence kinetic energy equation is expressed as:

$$\frac{\partial}{\partial x_i}\left(\rho u_i k\right)=\frac{\partial}{\partial x_i}\left(\frac{\mu_{ef}}{\sigma_k}\frac{\partial k}{\partial x_i}\right)+G_K+G_B-\rho\varepsilon \quad (4)$$

Here, σ_k is a turbulence model constant, G_K is the rate of shear production of k and G_B is the rate of buoyancy production of k. G_K and G_B are defined as given below:

$$G_K=\mu_t\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$$
$$G_B=-g_i\frac{\mu_t}{\sigma_t}\frac{1}{\rho}\frac{\partial \rho_i}{\partial x_i} \quad (5)$$

The dissipation rate of turbulence kinetic energy is given as:

$$\frac{\partial}{\partial x_i}\left(\rho u_i \varepsilon\right)=\frac{\partial}{\partial x_i}\left(\frac{\mu_{ef}}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right)$$
$$+\frac{\varepsilon}{k}\left(C_{1\varepsilon}G_K+C_{3\varepsilon}G_B-\rho C_{2\varepsilon}\varepsilon\right) \quad (6)$$

Where, $\sigma_\varepsilon, C_{1\varepsilon}, C_{2\varepsilon}, C_{3\varepsilon}$ are turbulence model constants. These definitions were made according to the standard k-ε turbulence model. In this model the turbulence model quantities are given as (Costa et al., 1999):

$$\mu_{ef}=\mu_t+\mu\,;\ \mu_t=\rho C_\mu\frac{k^2}{\varepsilon}\,;\ \Gamma_{ef}=\frac{\mu_t}{\sigma_t} \quad (7)$$

Where, μ_t is the turbulent viscosity, ρ fluid density, C_μ turbulence model constant and Γ_{ef} the effective exchange coefficient. Values of the turbulence model constants are given below.

$$\sigma_t=1.00\,;\ C_\mu=0.09\,;\ \sigma_k=1.00\,;\ \sigma_\varepsilon=1.314\,;$$
$$C_{1\varepsilon}=1.44\,;\ C_{2\varepsilon}=1.92\,;\ C_{3\varepsilon}=1.00 \quad (8)$$

Here, $C_\mu, \sigma_k, \sigma_\varepsilon, C_{1\varepsilon}, C_{2\varepsilon}$, $C_{3\varepsilon}$ are turbulence model constants. σ_t is the turbulent Prandtl number.

2. 2. Solution Method

The finite volume based PHOENICS code was used for the solution of the transport equations defining the problem. PHOENICS is a commercial code used for simulations of heat and mass transfer, fluid mechanics, chemical reaction and similar physical phenomena (Rosten and Spalding, 1987). This program provides numerical iterative approximations to the solution of non-linear partial differential equations. The numerical solution procedure applied is an improved version of the widely used SIMPLE algorithm. For the discretization of convective-diffusive transport, the hybrid scheme is used. The discretized equations are solved by the TDMA (Tri-Diagonal-Matrix-Algorithm). Distribution of the computational grid used is shown in Figure 2.

Trial solutions were obtained with a wide range of cell number combinations for grid independency checks. The final number of cells used was 100 for the x-direction and 80 for the y-direction. Finer grid distributions are employed at locations with larger anticipated gradients. Further details about the solution procedure applied can be found in Rosten and Spalding (1987), and Patankar (1980) and similar publications.

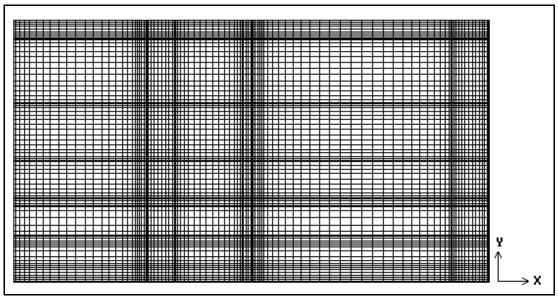


Figure 2. Distribution of the computational grid.

2. 3. Boundary Conditions

Velocities at the walls are zero because of the no-slip condition. Supply air inlet velocities u_{in} and v_{in}, inlet temperatures T_{in} and wall temperatures T_w were taken as given in Table 1. At the outlet the pressure is fixed to the ambient pressure and all variations of temperature, turbulent kinetic energy and its dissipation were taken to be zero. The no-slip boundary condition was also applied to the surfaces of the person, shelf, desk, and chair. The shelf, desk, and chair were assumed adiabatic, because at steady operation these surfaces will be at approximately the same temperature as the fluid surrounding them. The sitting person's outer surface temperature was taken to be a calculated cloth temperature of 25.4 ^{o}C. The inlet and outlet duct dimensions were taken as 0.2 m. Doors, windows, etc. are not taken into account. Simulated are only ventilation ducts, for which the second dimension is not taken into account as a result of the two-dimensional simulations.

The logarithmic law of the wall was used in regions close to the wall surfaces. Equations used for the turbulence inlet boundary conditions are given below (Costa et al., 1999).

$$k_{in} = 1.5 I_{t_{in}}^2 U_c^2$$
$$I_{t_{in}} = \left\{ \left[\left(\overline{u'^2} + \overline{v'^2} \right) / 2 \right]^{1/2} / U_c \right\} \tag{9}$$

$$\varepsilon_{in} = k_{in}^{3/2} / L_\varepsilon \; ; \qquad L_\varepsilon = d / 2 \tag{10}$$

Here, k_{in} is the turbulent kinetic energy at the inlet, I_{tin} turbulence intensity at the inlet, U_c characteristic velocity scale, u' and v' average fluctuating components of velocity, ε_{in} dissipation rate of k at the inlet, L_ε characteristic length and d is the jet slot width (Costa et al., 1999).

2. 4. Convergence and Grid Independency

The criterion of convergence of the numerical solution is based on the absolute normalized residuals of the equations that were summed for all cells in the computational domain. Convergence was considered as being achieved when these residuals become less than 10^{-3}, which was the case for most of the dependent variables. False time step relaxation for the three velocity components and temperature, and linear relaxation for pressure was used to obtain convergence that is more rapid. Relaxation factors ranged between 0.1 and 0.4. The simulations exhibited divergence if no relaxation was applied. Grid independency checks were made and the final simulations were achieved with 100x80 cell numbers in the x-y coordinate directions.

3. NUMERICAL SIMULATION RESULTS

3. 1. Verification of the CFD Method

For the verification of the problem definitions used for the present study using the PHOENICS code, results reported by Costa et al. (1999) were used. Costa et al. (1999) investigated room airflow where a horizontal cold air jet and a vertical warm air jet were used at the left wall and an exit duct at the bottom of the right wall. The velocity of the cold jet is u_{in} = 0.80 m/s and of the warm jet v_{in} = 0.87 m/s. The dimensions of the investigated closed volume are 1040 x 1040 x 700 mm. The widths of the supply inlet ducts are d = 20 mm, outlet duct 24 mm, cold jet inlet temperature is 14 °C, warm jet inlet temperature is 35 °C. All the walls of the room are at a temperature of 14 °C. Experimental results from the above mentioned paper was compared with the present CFD solution method. These verification results are reported in detail by Kuas et al. (2002). Examination of this publication shows that the results of the CFD solutions obtained from the present method are in good agreement with the experimental results by Costa et al. (1999).

3. 2. Results and Discussion

Table 1 lists ventilation alternatives studied depending on inlet/outlet configurations, summer/winter conditions and inlet velocity/temperature values.

Table 1. Conditions of the Cases Studied

Case No:	Inlet/Outlet Locations	Summer/ Winter	u_{in}, v_{in} (m/s)	T_{in} (°C)	T_w (°C)
1	Inlet 4, Outlet 5	Summer	0.50	18	30
2	Inlet 4, Outlet 5	Winter	0.75	27	10
3	Inlet 4-5, Outlet 1-3	Summer	0.50	18	30
4	Inlet 4-5, Outlet 1-3	Winter	0.75	27	10
5	Inlet 1, Outlet 3-5	Summer	0.75	18	30
6	Inlet 1, Outlet 3-5	Winter	0.75	27	10

Inlet/outlet location numbers are shown in Figure 3. The results are analyzed in detail for various profiles of the room. The profiles chosen are numbered in Figure 4. These profiles are chosen around the sitting person, because the person's comfort conditions are important.

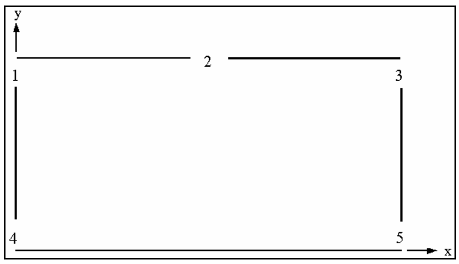


Figure 3. Inlet/outlet locations studied

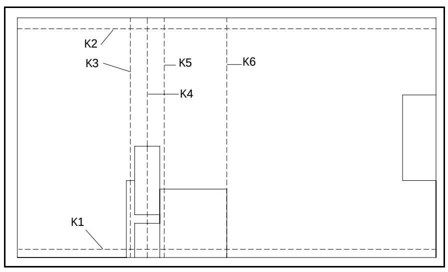
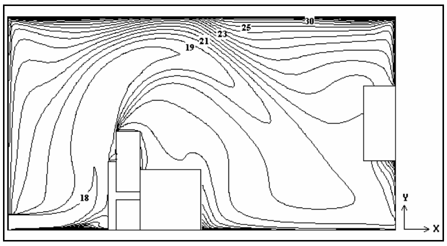
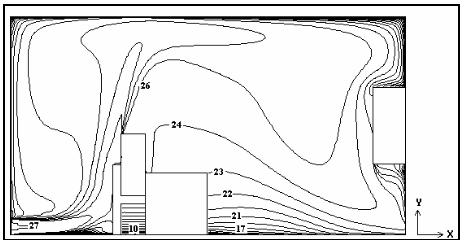


Figure 4. Profiles chosen for presentation of results

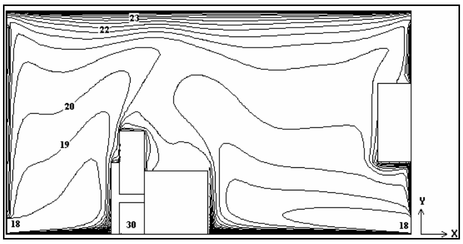
Figure 5 shows predicted temperature contours and Figure 6. shows velocity vector plots for the summer cases studied. On these plots, one can see that there is a significant effect of the occupants (see relevant literature) on the temperature distribution. In addition, changes of inlet/outlet locations significantly alter the temperature distributions.



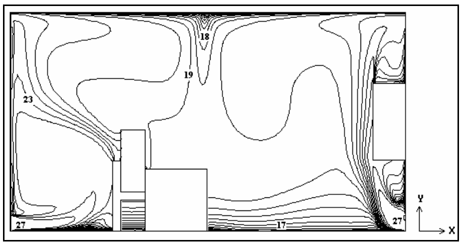
Case 1



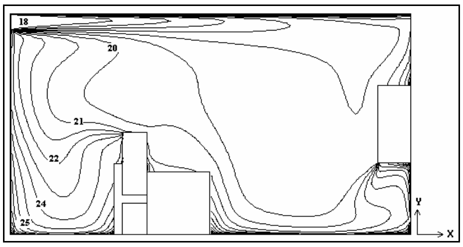
Case 2



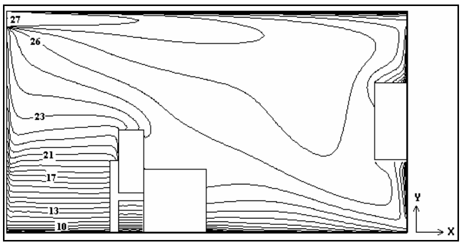
Case 3



Case 4

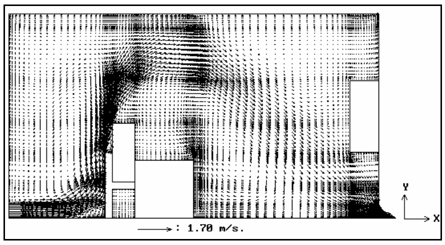


Case 5

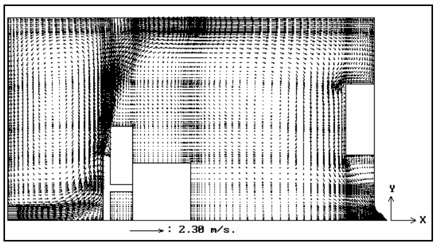


Case 6

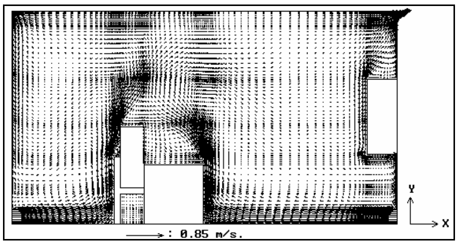
Figure 5. Temperature contour distributions for all the cases studied



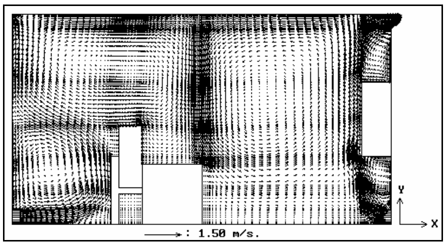
Case 1



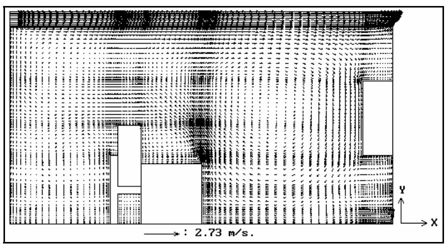
Case 2



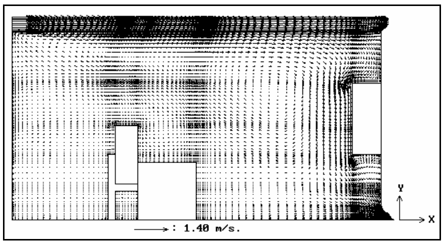
Case 3



Case 4



Case 5



Case 6

Figure 6. Velocity vector distributions for the summer cases studied

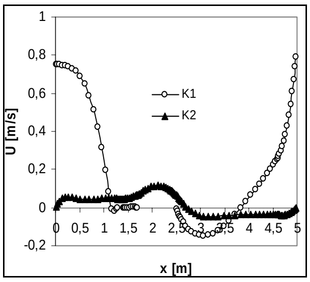
Velocity vector plots are shown in order to better make sense of the temperature distributions. From the size of the vectors, supply duct and exhaust vent locations are clearly visible on these plots (see Figs. 5-6). Overall, full recirculation in the size of the room is prevented because of the occupants. Instead, the supply air stream creates smaller recirculations at different locations (mainly around the spaces of the occupants) before being expelled at the outlet.

If Figs. 5 and 6 are analyzed jointly, one can see that the temperature distribution develops because of airflow characteristics resulting from the variation of inlet/outlet locations and the occupant’s size and location. In other words temperature and velocity distributions are coupled as a result of the convection nature of the flow. Occupants change the airflow patterns, thus, changing the temperature distribution.

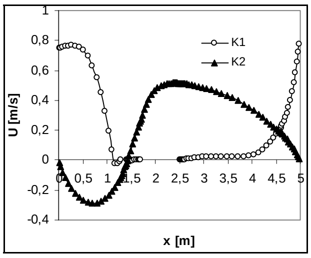
In addition, from Figs. 5 and 6, one can see that there is a substantial difference between the airflow characteristics of the cooling air distributions for the summer cases and the heating air distributions for the winter cases. This is due to the different behavior of the supply air jets, because of the temperature difference between average room temperature and supply air temperature. The cold and heavier cool air jets will drop faster than compared to the hotter and lighter heating jets which tend to rise or at least penetrate further into the room. Hence, the recirculation and air movement is wholly different in summer and winter conditions. Furthermore, some parts of the room exhibit a stratified temperature variation. The stratification occurs mostly close to the ceiling for the summer cases, and close to the floor for the winter cases.

It is possible to continue with further cross sectional plots of isotherms and velocity vectors, however, it is also important to see the actual distributions of temperature and velocity components for certain relevant profiles chosen at important locations. Hence, the next figures show temperature and velocity distributions at some profiles shown in Figure 4. These quantitative figures will enhance the conclusions drawn from the mostly qualitative contour and vector plots.

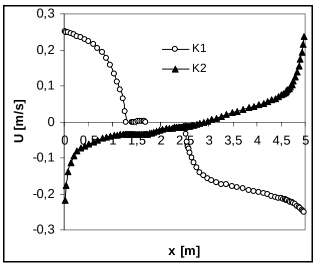
Velocity distributions for profiles K1 and K2 are shown in Figs. 7-8, and temperature distributions for profiles K3, K4 and K5 are shown in Figs. 9-10. All figures use full lengths of the room for the x and y axis. The gaps in the profiles are due to the occupants. From Figs. 7-8 the largest variations can be seen for profiles along the supply air jet and exhaust locations, hence, making these locations visible. The air jet velocity decay characteristics of the hot and cold jets differ as a result of buoyancy forces which become more dominant in regions with lower velocities. Supply air velocities decay to very small velocity values, which dominate the flow in the center region of the room. Values of velocities are also very important from a comfort-condition point of view. The inspection of values of velocities near the person shows that they all are lower than the values associated with the comfort conditions given in the literature (Anon, 1997a; b). Nevertheless, the risk of draft is present near the inlet and outlet sections.



Case 1

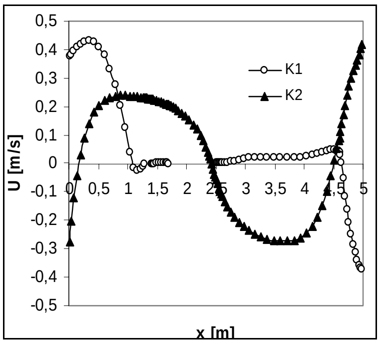


Case 2

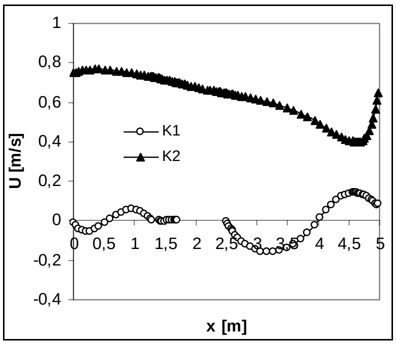


Case 3

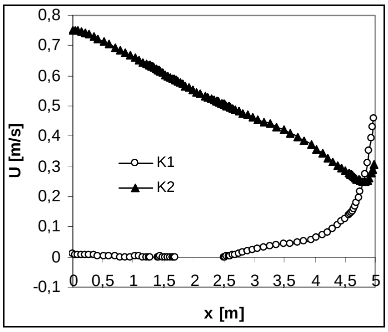
Figure 7. Velocity profiles along K1 and K2 for Cases 1, 2 and 3



Case 4



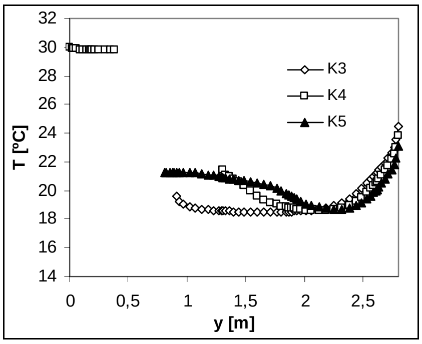
Case 5



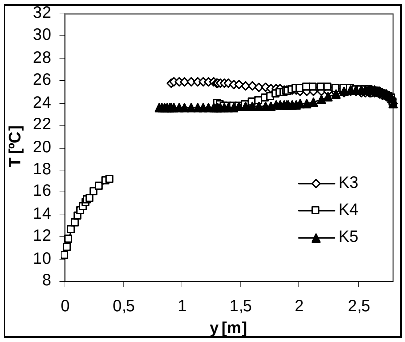
Case 6

Figure 8. Velocity profiles along K1 and K2 for Cases 4, 5 and 6

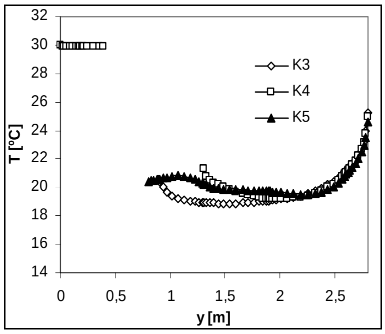
Inspection of Figs. 9-10 shows that the temperature values along the different profiles are mostly similar and close to each other. As can be seen from Figure 4, K3 to K5 are chosen around the person, and Figs. 9-10 show that temperatures do not differ much between these profiles. Effects due to the cold and hot supply air jets are also visible. The temperature differences between certain elevations above the floor are an important comfort criteria. From Figure 8 it can be seen, that overall temperature differences are not very large. However, for certain cases the values are probably exceeding comfort conditions.



Case 1

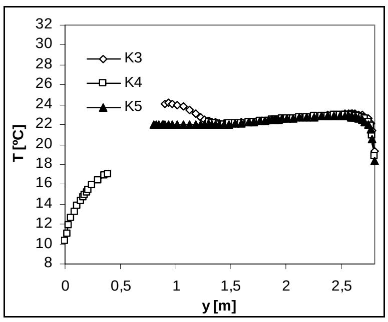


Case 2

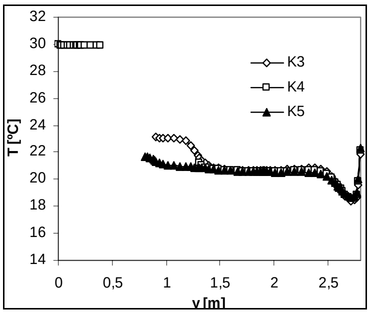


Case 3

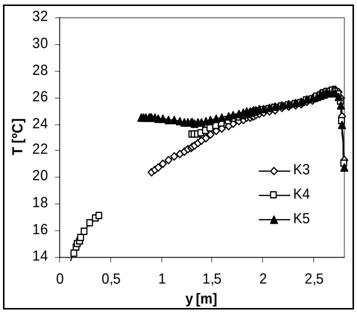
Figure 9. Temperature vs. room height along K3, K4 and K5 for Cases 1, 2 and 3



Case 4



Case 5



Case 6

Figure 10. Temperature vs. room height along K3, K4 and K5 for Cases 4, 5 and 6

4. CONCLUSIONS

Computational fluid dynamics simulations were undertaken for the determination of the characteristics of cooling/heating air sent to an office room that contains a person and other occupants. Effects of the occupants under different inlet/outlet and summer/winter configurations on the airflow have been analyzed for two different supply jet velocities. In order to determine airflow characteristics inside the room, velocity vectors, velocity profiles, temperature contours and temperature profiles for various cross sections of the room have been investigated.

Compared to studies with empty rooms, a full circulation proportional to the size of the room does not occur. The circulation is being interrupted and changed by the occupants; resulting in smaller recirculations, flow reversals and other various different flows. Furthermore, the occupants are hindering penetration of the supply air jets. These effects can result in poor mixing and ventilation. As can be understood from these comparisons, airflow in a room is highly influenced by persons and occupants present in the room space, as well as inlet/outlet configurations. In addition, appropriateness of inlet/outlet locations in terms of comfort conditions varies with cooling/heating configurations.

5. REFERENCES

Anonymous, 1997a. ASHRAE Handbook – Fundamentals. American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc., Atlanta, Georgia, U.S.A..

Anonymous, 1997b. CIBSE. Natural Ventilation in Non-Domestic Buildings, CIBSE Applications Manual AM10, London.

Ayad, S. S. 1999. Computational Study of Natural Ventilation, J. of Wind Engineering and Industrial Aerodynamics 82, 49-68.

Awbi, H. B. and Hatton, A. 2000. Mixed Convection from Heated Room Surfaces, Energy and Buildings 32, 153-166.

Chow, W. K. and Wong, L.T. 1999. Local Air Speeds Measurement in Mechanically Ventilated Spaces, Building and Environment 34, 553-563.

Costa, J. J., Oliveira, L. A. and Blay, D. 1999. Test of Several Versions for the k-ε Type Turbulence Modelling of Internal Mixed Convection Flows, Int. J. of Heat and Mass Transfer 42, 4391-4409.

Gan, G. 2000. Effective Depth of Fresh Air Distribution in Rooms with Single-Sided Natural Ventilation, Energy and Buildings 31, 65-73.

Kuas, G., Başkaya, Ş. and Sivrioğlu, M. 2002. Numerical analysis of effects of occupants on forced ventilation inside a room, **V. International HVAC+R Technology Symposium**, April 29 – May 01, İstanbul, Turkey.

Lam, J. C. and Chan, A. L. S. 2001. CFD Analysis and Energy Simulation of a Gymnasium, Building and Environment 36, 351-358.

Niu, J. and Kooi, J. 1994. Indoor Climate in Rooms with Cooled Ceiling Systems, Building and Environment 29 (3), 283-290.

Patankar, S. V. 1980. Numerical Heat Transfer and Fluid Flow, Hemisphere, New York.

Rosten, H. and Spalding, B. 1987. PHOENICS Beginners Guide, CHAM/TR100, London.

Sinha, S. L., Arora, R. C. and Roy, S. 2000. Numerical Simulation of Two-Dimensional Room Air Flow with and without Buoyancy, Energy and Buildings 32, 121-129.

Xing, H., Hatton, A. and Awbi, H. B. 2001. A Study of the Air Quality in the Breathing Zone in a Room with Displacement Ventilation, Building and Environment 36, 809-820.

Xue, H. and Shu, C. 1999. Mixing Characteristics in a Ventilated Room with Non-Isothermal Ceiling Air Supply, Building and Environment 34, 245-251.

Yamatomo, T., Ensor, D.S. and Sparks, L.E. 1994. Evaluation of Ventilation Performance for Indoor Space, Building and Environment 29 (3), 291-296.