**Proceedings of ICFDP7:**
**Seventh International Conference of Fluid Dynamics and Propulsion**
**December 18-20, 2001, Cairo, Egypt**

**ICFDP7-2001058**

# COMPUTATIONAL STUDY OF FLOW IN MECHANICALLY VENTILATED SPACES

**Basman Elhadidi**
University of Notre Dame
Notre Dame, IN, 46556
USA

**Atef Sherif**
Aerospace Engineering
Cairo University
Giza - Egypt

## ABSTRACT

A CFD model is presented to simulate flow inside ventilated spaces. Sensitivity studies are presented to examine the effect of room geometry, obstacle location in rooms and effect of supply temperature on the room flow patterns. The efficiency of two design alternatives for cooling and ventilating an office space is presented. Performance indices for the comparison are based on thermal comfort of the occupants inside the space. Computer performance shows that HVAC engineers can optimize heating/cooling and ventilation performance using three-dimensional CFD models at ease for relatively cheap cost.

## INTRODUCTION

o

on are new terms being introduced in the building community. The frequency and abundance of these terms indicates that there is a growing concern to find cleaner, safer homes that are cost effective and energy wise efficient. Recent studies indicate that 80% of our life is spent indoors and that about one third of the [1].

Development of faster computers may aid this new growing concern. CFD provides a cost effective method to predict flow field check the efficiency of their design and validate the ASHRAE/ISO standards for clean air[2]. Researchers[3] using CFD simulations in Hong Kong found that the annual cooling load in a gymnasium was overestimated by 45%. There initial stratification, which affects the ventilation and cooling effectiveness. This is an indication that the flow physics is complex and needs careful understanding and modeling, an approach feasible only by CFD techniques.

CFD developments for building simulations were done by Terai[4], Awbi *et.al*.[5-7], Davidson *et.al.*[8-11], Gan[12] and others. Awbi simulated flows in two and three-dimensional office spaces for fully turbulent flows using the standard $k$-$\varepsilon$ method. The studies showed the importance of understanding the effects of buoyant flows where instabilities may occur. Davidson compared the numerical simulations with experiments and found that for the large majority of indoor flow conditions numerical simulation predicts indoor flow correctly, except for highly buoyant flows which are unstable chaotic flows in nature (Rayleigh-Bnard instabilities). Studies by Chen *et.al.*[13], show the importance of inflow diffuser placement. Different diffuser locations affect the indoor air patterns, the ventilation effectiveness and the sensation of thermal comfort. From the computational perspective the airflow in rooms is very difficult to simulate. The flow is three-dimensional, turbulent, and buoyant and the complex geometry is not easy to simulate. However, several simplifying assumptions are usually placed to make the problem feasible. The flows are always assumed incompressible with local compressibility taken into account by the Boussinesq approximation. For most simulations the standard $k$-$\varepsilon$ method is used to model turbulence. Recently Davidson[12] used the large eddy simulation (LES) to compare it to standard $k$-$\varepsilon$ techniques. It was concluded that the LES models remain expensive, however, they will be feasible in the near future. Chen *et.al.*[14] reviewed the turbulence models used for indoor airflow and concluded that the low Reynolds $k$-$\varepsilon$ models are the most suitable for three-dimensional calculations at present.

tant design phase. For mechanical/natural ventilation two measures are used, one is the indoor air quality (IAQ) and the other is the thermal comfort. The ISO/ASHRAE fix the ventilation rates for permissible IAQ. sense, the ventilation rate may not be enough for an ineffective ventilation design and may be too much for an effective design. To assess ventilation effectiveness, contaminant simulation is needed and removal time of contaminants may be quantified by th [8,9]. Thermal comfort is quantified by experimental correlations. Fanger[15-17], derived detailed correlations correlating the percent of dissatisfied (PD) with the airflow parameters in a ventilated space. This index is the most complete up to date index and is used for all design quantifications.

This paper studies the effects of room geometry, obstacle position and supply temperature on airflow patterns inside a room. For that study the flow is assumed laminar. Two design alternatives for an office space are presented, to find the optimum ventilation technique. The design goal is to satisfy thermal comfort indices and the ISO standards. For that study the flow is assumed turbulent. A comparison of code performance on different computer platforms is presented, and finally a conclusion.

## NOMENCLATURE

### LATIN

| | |
|---|---|
| Ar | Archimedes number. |
| $A_R$ | Aspect ratio. |
| $c_p$ | Specific heat. |
| $c_{1\varepsilon}$, $c_{2\varepsilon}$, $c_\mu$ | Turbulence coefficients. |
| $f_{cl}$ | Constant. |
| g | Acceleration of gravity. |
| $G_B$ | Buoyancy production. |
| H | Room height. |
| $h_{in}$, $h_{out}$, $h_{obstacle}$ | Supply/exit height, obstacle height. |
| K | Kinetic Energy. |
| L | Room length. |
| M | Metabolic rate. |
| P | Pressure. |
| $P_K$ | Turbulence production. |
| Q | Internal heat source. |
| Re | Reynolds number. |
| T, $t_{cl}$ , $t_r$ | Temperature, clothing temperature, radiation temperature. |
| U | Velocity component. |
| W | Work |
| $w_{obstacle}$ | Obstacle width. |

### GREEK

| | |
|---|---|
| $\varepsilon$ | Dissipation. |
| $\phi$ | Variable. |
| $\Gamma$ | Diffusivity. |
| $\mu$ | Viscosity. |
| $\rho$ | Density. |
| $\sigma_k$, $\sigma_\varepsilon$, $\sigma_t$, $\sigma_\mu$ | Turbulence coefficients. |

## DESIGN ALTERNATIVES

There are four major design alternatives that may be used to achieve ventilation effectiveness. Figure 1 presents a local exhaust ventilation (LEV) technique. This is widely used in industrial applications where localized sources of contaminants are present.

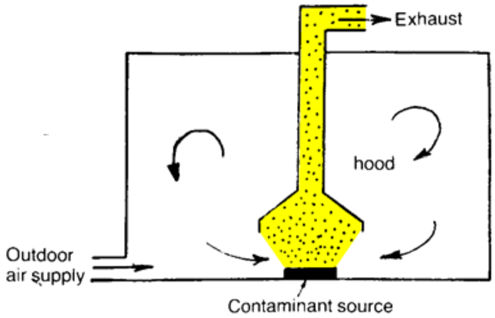

**Figure 1 Schematic representing a LEV**

Figure 2 presents the piston ventilation technique. This is

small contaminant sources by dragging them down to the exhaust; however, it is not suitable for large ventilation rates as the turbulence levels affect the efficiency.

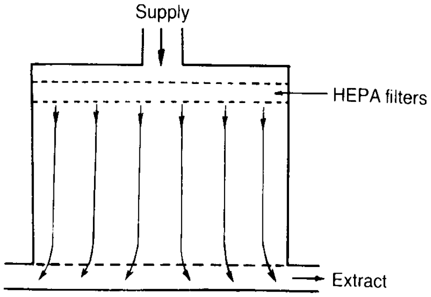

**Figure 2 Schematic representing piston ventilation**

Figure 3 presents the displacement ventilation mechanism. Cooler air is supplied at the lower level and is extracted at the ceiling. Heat loads such a
cold air and produce an upward plume that carries any contaminant produced ($CO_2$ in case of humans). This is an efficient ventilation technique and results in the best IAQ. Figure 4 presents the process of mixing ventilation. In this process a jet of air mixes in the space diluting the contaminant. The overall cooling/heating is faster than any other ventilation scheme, however it produces the worst IAQ.

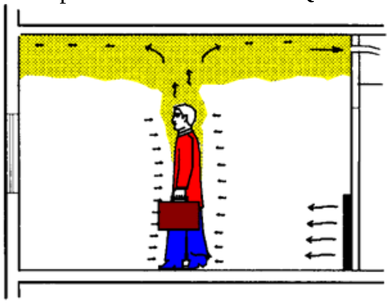

**Figure 3 Schematic representing displacement ventilation**

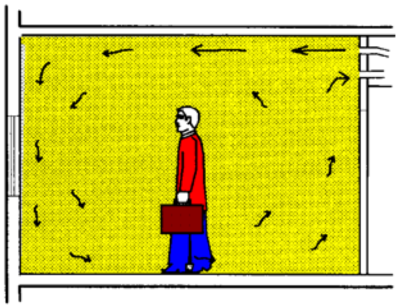

**Figure 4 Schematic representing mixing ventilation**

## THERMAL COMFORT

Thermal comfort is evaluated in terms of the predicted mean vote (PMV) and predicted percentage of dissatisfied (PPD) as proposed by Fanger[15-17]. PMV is an empirical correlation relating the air velocity, air temperature, mean radiant temperature, water vapor, clothing thermal resistance and occupant metabolic rate. This correlation was obtained after extensive experiments on human test cases from different sexes and ages[17]. The PMV is given by,

$$\mathrm{PMV}=\begin{pmatrix}0.303e^{-0.036M}+\\0.028\end{pmatrix}\begin{Bmatrix}6.937+0.556(M-W)\\-3.05\times10^{-3}P\\-1.7\times10^{-5}MP\\-0.147M-0.0014MT\\-3.96\times10^{-8}f_{cl}\begin{bmatrix}(t_{cl}+273)^4-\\(t_r+273)^4\end{bmatrix}\\-f_{cl}h_c(t_{cl}-T)\end{Bmatrix}$$

The PPD is related to the PMV by the equation,

$$\mathrm{PPD}=100-95\exp\left(-0.03353\mathrm{PMV}^4-0.2179\mathrm{PMV}^2\right) \qquad (1)$$

To satisfy ISO standards, PMV is within the range,

$$-0.5\le \mathrm{PMV}\le 0.5 \qquad (2)$$

Thermal discomfort may occur due to other factors such as turbulence, radiant temperature effects, cold/hot floors and to vertical temperature gradients. The vertical temperature gradient is mostly sensed in the case of natural and displacement ventilation designs. Figure 5, shows the effect of the vertical air temperature gradient on the percentage dissatisfied.

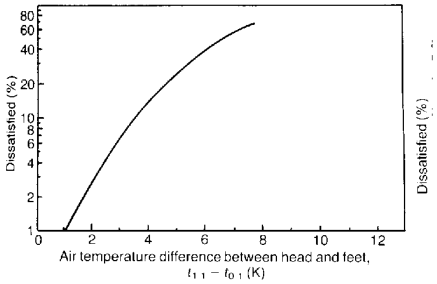

**Figure 5 Allowable temperature difference between head and feet**

## MATHEMATICAL MODEL

The mathematical description of airflow is based on the fundamental laws of physics; conservation of mass, momentum and energy. These equations can be expressed in the form,

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial}{\partial x_i}(\rho\phi u_i)=\frac{\partial}{\partial x_i}\left(\Gamma_\phi\frac{\partial\phi}{\partial x_i}\right)+S_\phi. \qquad (3)$$

Where $\phi$ is the dependent variable. Table 1 gives the expressions for the source terms $S_\phi$ for each variable that is likely to be needed in solving ventilation problems. The constants for the k-ε method are the standard constants proposed by Launder and Spalding[18].

$$\sigma_k=1.0 \quad \sigma_\varepsilon=1.3 \quad \sigma_t=0.9 \quad \sigma_\mu=0.09 \quad c_{1\varepsilon}=1.44 \quad c_{2\varepsilon}=1.92$$

The turbulent viscosity, turbulence production term and buoyancy productions terms are,

$$\begin{aligned}\mu_t&=\frac{c_\mu\rho k^2}{\varepsilon},\\P_k&=\mu_t\left[\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right]\frac{\partial u_i}{\partial x_j},\\G_B&=-\frac{\mu_t g\beta}{\sigma_t}\frac{\partial t}{\partial y}.\end{aligned} \qquad (4)$$

The SIMPLER (Semi Implicit Method for Pressure Linked Equations Revised) scheme proposed by Patankar[19] is applied. This finite volume formulation applies a staggered grid for the pressure variable to avoid pressure instabilities and checkering effect. The pressure correction equation is solved by an ADI (alternate differencing implicit) iterative scheme. This is suitable for time dependent problems. Inflow velocity, turbulence and dissipation levels, temperature and pressure. are specified. Along the boundaries the standard log-wall rule is used for turbulent flows and non-slip/impermeability condition for laminar flows.

| Equation | $\phi$ | $\Gamma_\phi$ | $S_\phi$ |
|---|---|---|---|
| Continuity | 1 | 0 | 0 |
| Momentum | $u_i$ | $\mu_e$ | $-\nabla P + \nabla \cdot \left(\mu_e \nabla \cdot \vec{U}\right) + \rho \vec{g}$ |
| Temperature | T | $\Gamma_e$ | $Q/c_p$ |
| Kinetic Energy | k | $\Gamma_k$ | $P_k - \rho\varepsilon + G_B$ |
| Dissipation rate | $\varepsilon$ | $\Gamma_\varepsilon$ | $C_{1\varepsilon}\frac{\varepsilon}{k}\left(P_k + G_B\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$ |

**Table 1 Source Terms in the Transport Equations for Turbulent Flow**

## RESULTS

Before attempting to design a ventilated space, we need to ask ourselves: What type of flow pattern should we have[20]? Each room has different ventilation requirements, and turbulence levels to maintain its IAQ goal. A surgery room for instance will either use displacement or piston ventilation systems to keep the turbulence level low and high IAQ. Other applications such as rapid cooling will require mixing ventilation systems. There are several factors that affect indoor airflow; wall heat transfer, internal heat sources, velocity at supply terminal, location of supply terminals, room geometry, obstacles and many more. Including all these effects together is difficult, but a thorough understanding of a few is essential.

### Effect of room geometry

It is convenient to introduce the aspect ratio for two-dimensional spaces by,

$$A_R = L/H. \tag{5}$$

For normal offices the $A_R$ is $1<A_R<3$. In corridors the value of $A_R$ may exceed three. In older buildings the values of $A_R$ may be even less than unity; mainly because the volume above the occupied zone was used as a buffer zone for contaminants rising from people and other sources by convective currents. Figure 6a, 6b and 6c present three different simulations for $A_R$ equal to 0.5, 1 and 2, for a fixed Re=1000 based on diffuser height. For all cases $h_{in}/H=h_{out}/H=0.2$. Comparing the plots we see that for the lowest $A_R$ the flow acts more like a piston ventilation system. The flow is forced to turn upwards due to the narrow geometry. For the $A_R$ =2, it is more like displacement ventilation with horizontal stratification of the flow. This simple geometry change is an indication that correct choice of terminal supply is related to room geometry, and different ventilation designs may arise if this consideration is not taken into account.

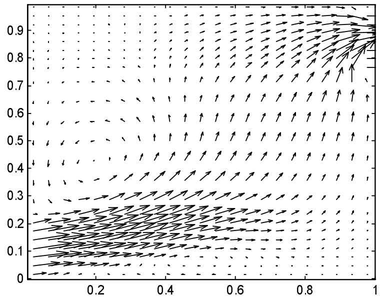

**Figure 6a Flow in a space with $h_{in}=h_{out}=h/5$, Re=1000, $A_R=1$**

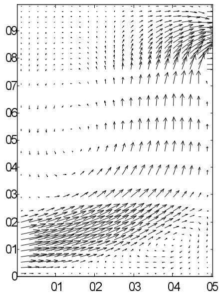

**Figure 6b Flow in a space with $h_{in}=h_{out}=h/5$, Re=1000, $A_R=1/2$**

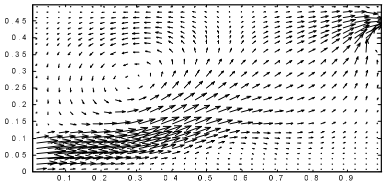

**Figure 6c Flow in a space with $h_{in}=h_{out}=h/5$, Re=1000, $A_R=2$**

### Effect of Obstacles

Objects, which protrude into the primary air stream, may deflect the jet path and strongly affect the air motion. There are various architectural obstacles such as beams and columns, which are unavoidable. Serious draught risks may appear, since the volume is decreased forcing the velocity to increase in the occupied zone. Figure 7a and 7b show an obstacle placed in the flow field. The obstacle dimensions are $w_{obstacle}/L=0.5$, $h_{obstacle}/H=0.05$. For case (a) the obstacle was placed against the edge opposing the inlet location. The flow beneath became a stagnant zone. Moving the obstacle slightly left as shown in case (b) high velocities are formed at the sides of the object. In real situations these velocities may be the cause of high draught currents. The simulations show that an obstacle that may act as a partition in a room (viewed from top) may prevent the ventilation of one of the partitions in the room creating bad IAQ. For spaces with partitions supply terminals are best placed on the ceiling to create a uniform distribution, however, this may cause down draughts.

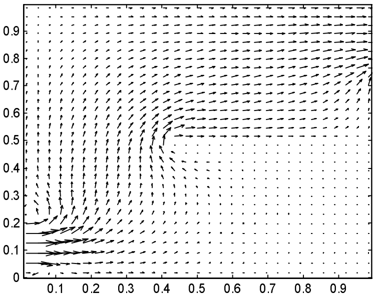

**Figure 7a Object Placed adjacent to wall**

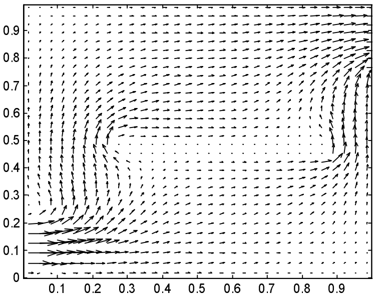

**Figure 7b Object moved slightly left**

## Effect of supply temperature

The supply of air is both a source of buoyancy and momentum. The momentum is nondimensionalized and characterized by the Reynolds number (Re), the buoyancy force is characterized by the Archimedes number (Ar). The Ar number is based on the temperature difference, ΔT, between the extract and supply and is given by,

$$Ar = \frac{g\Delta T}{T}\frac{L}{u_{\infty}^{2}} . \qquad (6)$$

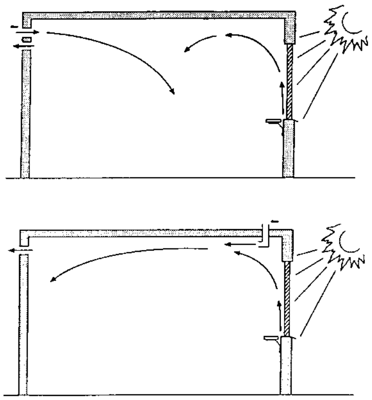

**Figure 8 Upper figure presents case of opposing flow, lower figure presents case of aiding flows**

In addition to the Archimedes number the location of the heating and cooling loads within the room are important parameters, because the flow set up by sources may either counteract or assist the flow set up by the jet, as seen in the Figure 8.

## Supply of warm air

With the supply of *warm air* it is possible that the air will only circulate in the upper part of the room which leads to stratified flow. Figure 9a, 9b show the velocity vectors and temperature contours when the buoyancy is neglected. Increasing the temperature difference and taking the effect of buoyancy into account is seen in figure 9c and 9d. An upward plume is seen, with a large recirculation vortex. This vortex mixes the air in the room and entrains little heat from the plume. This is an example of an inefficient form of heating, where the heating energy is lost without effective heating to the room. The IAQ would be low as well; any contaminant in the room will remain and recirculate.

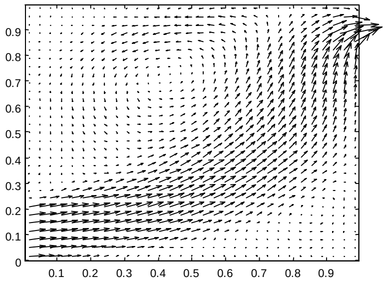

**Figure 9a** Re=1000, Ar=0. Velocity vectors.

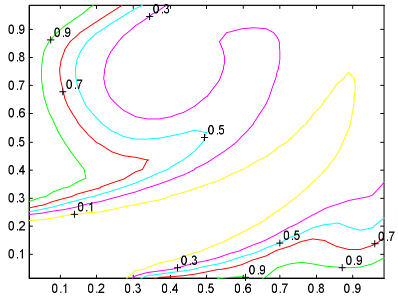

**Figure 9b** Re=1000, Ar=0. Temperature contours.

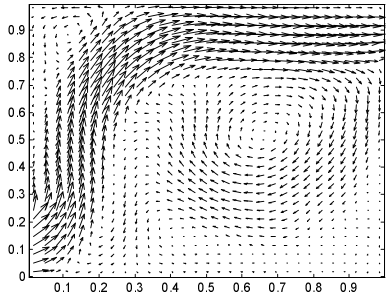

**Figure 9c** Re=1000, Ar=5. Velocity vectors.

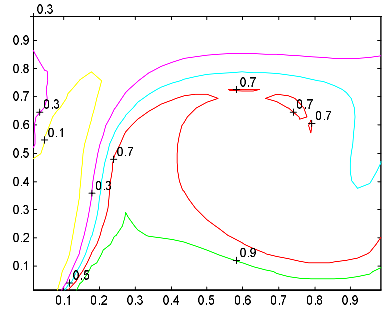

**Figure 9d** Re=1000, Ar=5. Temperature contours.

## Supply of cold air

With the supply of *cold air* it is possible for the jet to drop into the occupied zone and cause unacceptable high velocities. As seen in Figure 10a, 10c the effect of the cold air is to deflect the flow downwards decreas

may lead to thermal discomfort for the occupants under the jet and bad IAQ were other mixing occurs.

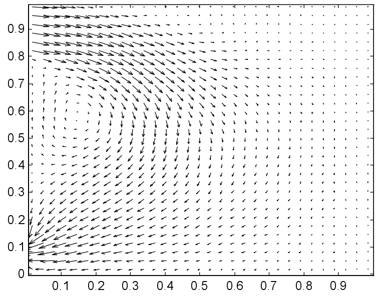

**Figure 10a** Re=1000, Ar=0. Velocity vectors.

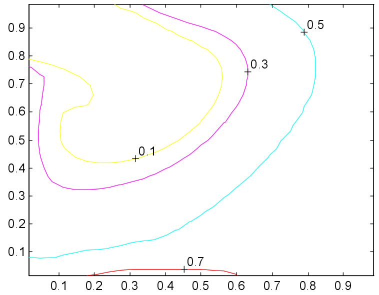

**Figure 10b** Re=1000, Ar=0. Temperature contours.

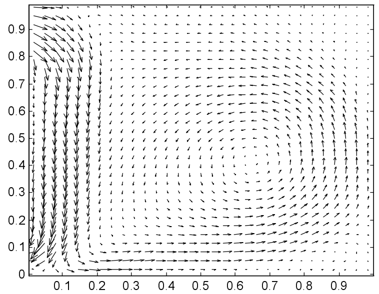

**Figure 10c** Re=1000, Ar=-5. Velocity vectors.

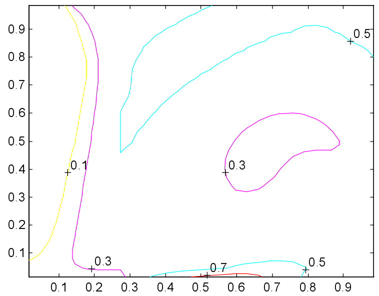

**Figure 10b** Re=1000, Ar=-5. Temperature contours.

It is important to note that heating/cooling effects may

flow path and lead to strong plumes forming upwards or downwards, with large circulation zones. This eventually leads to bad IAQ and unacceptable thermal conditions. Other effects of heating can be seen in figure 11. Mixing and displacement ventilation are used to cool a computer workstation, or heat source. The mixing system results in an unstable plume that bursts out heat into the room at moments. The displacement system produces a well-stratified environment inside the space. Both thermal comfort and IAQ for the displacement system would be better.

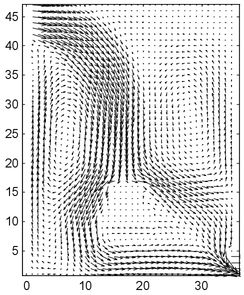

**Figure 11a** Computed Velocity Vectors (Mixing system)

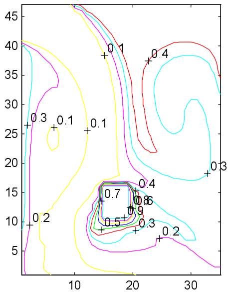

**Figure 11b** Computed Temperature Contours (Mixing system)

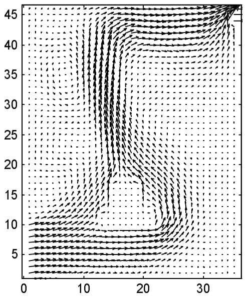

**Figure 11c** Computed Velocity Vectors (Displacement system)

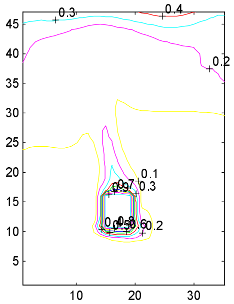

**Figure 11d** Computed Temperature Contours (displacement system)

## Design of an office space

An office space (3x4x3 m), having an inlet and outlet area equal to 25 cm with two occupants is analyzed. According to the ISO standards a flow rate of 27 l/s is required to ventilate this space effectively, if one of the workers is a smoker. Two design alternatives are studied, one is a displacement system and the other is a mixing system.

## Displacement ventilation

For the given flow rate, and diffuser area the Reynolds number is 10,000 (based on room length, 810 based on diffuser height) and the Archimedes number is 60. The supply temperature is fixed at 25°C, and the room temperature, was initially fixed at 30°C respectively.

Figure 12 presents the results for that case. It is seen from figures 12c and 12d, that the flow pattern is acceptable since the condition $-0.5 < PMV < 0.5$ is satisfied in most of the domain. The velocity vectors and temperature contours are showing stratified flows, which are characteristics of displacement ventilation systems.

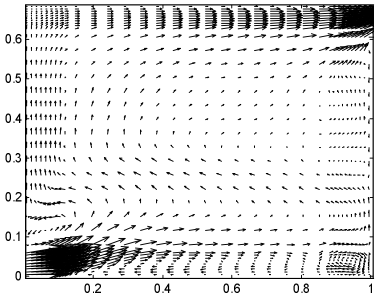

**Figure 12a** Computed Velocity Vectors

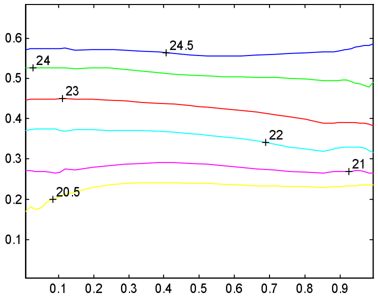

**Figure 12b** Computed Temperature Contours

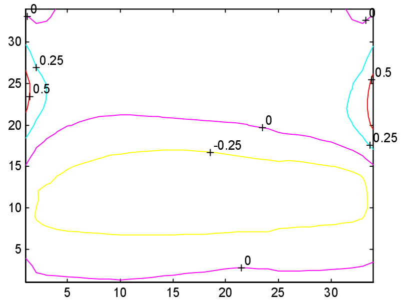

**Figure 12c** PMV contour values

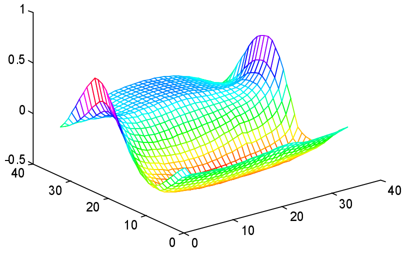

**Figure 12d** PMV in domain

## Mixing ventilation

Applying mixing ventilation is shown below for the same problem. For the given flow rate, and diffuser area the Reynolds number is 40,000 (based on room length, 3600 based on diffuser height) and the Archimedes number is 1.8. The supply temperature is fixed at 25$^{o}$C and the room temperature, is fixed initially at 30$^{o}$C.

Figure 13 shows the results for this case. It is seen from figures 13c and 13d, that the flow pattern is slightly acceptable since the condition -0.5 < PMV < 0.5 is satisfied in most of the domain. The jet throw is easily seen to be equal to approximately one half of the room length. The mixing zones to the left and right of the jet are characteristics of mixing ventilation.

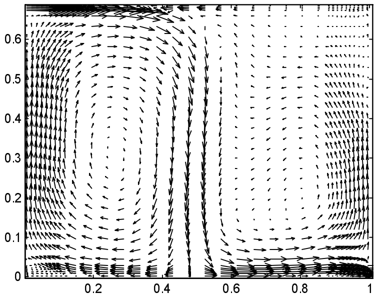

**Figure 13a** Computed Velocity Vectors

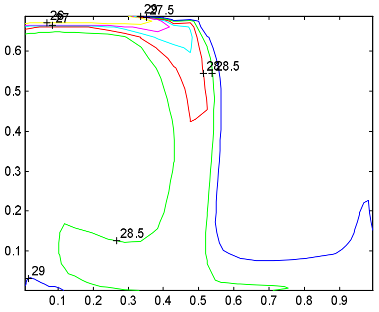

**Figure 13b** Computed Temperature Contours

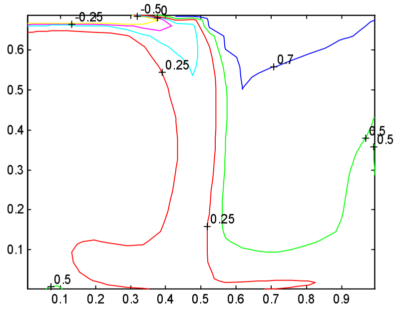

**Figure 13c** PMV contour values

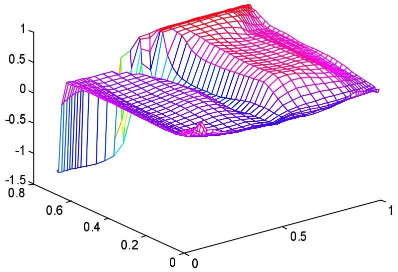

**Figure 13d** PMV values in domain

Even though both designs are thermally acceptable, the displacement system performs better. The PMV is much more uniform, with no real cold spots or hot spots in the room. The IAQ is much better, since less mixing and turbulence exists. Based on that, for an office space, displacement systems act more favorably.

## Three-dimensional effects

Three-dimensional effects are significant and may alter the flow patterns drastically. For the same office dimensions presented above simulations were performed for mixing and displacement ventilation systems. Figure 14a and 14b show the velocity vectors in the xz plane and the temperature distribution in the xy plane for the mixing case. The inlet and outlet diffusers have an area of 0.25 and 0.4 m$^{2}$. Figure 15a and 15b show the velocity vectors in the xz plane and the temperature distribution in the xy plane for the displacement ventilation case. The inlet and outlet diffusers are as long as the room and are 0.25 m wide. The temperature differential between the inlet and outlet was 3$^{o}$C, and there is an obstacle in the middle of the room.

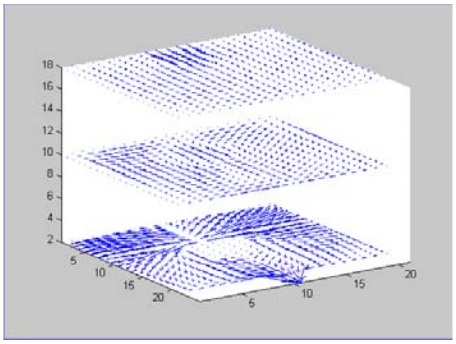

**Figure 14a** Flow velocity in xz-plane.

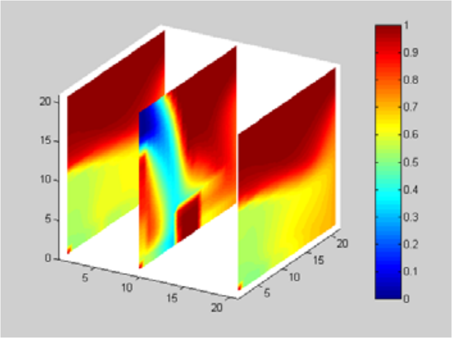

**Figure 14b** Temperature distribution in xy-plane.

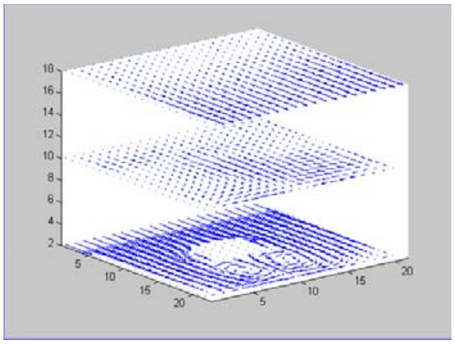

**Figure 15a** Flow velocity in xz-plane.

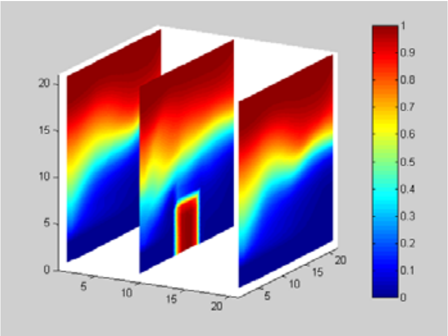

**Figure 15b** Temperature distribution in xy-plane.

As seen from the three-dimensional simulations the flow patterns are complex. When obstacles are simulated in two-dimensional flows they produce stagnant zones behind them, whereas in three-dimensional flows there is separated flows as seen in figure 15a. The temperature distribution and thermal comfort will depend on the correct flow field prediction. These complicated flow patterns induce draught risks and should be taken into account while computing the thermal comfort indices. This extra computational cost is now feasible and a full three-dimensional solution may be obtained in the order of hours.

## Computer time

With the advent of the new Pentium IV processors, the simulation time needed to perform a three-dimensional calculation has dropped significantly orders of magnitude. Shown in figure 16 is the current computer performance compared to older Pentium chipsets. The processors used for this comparison were, Pentium II, Pentium III 1000 MHz, Pentium IV 1500 MHz, and finally a Sun Blade 1000 machine with a 600 MHz UltraSparc III processor. With this technological speedup it is expected to be able to solve fully three-dimensional turbulent flow within the order of hours.

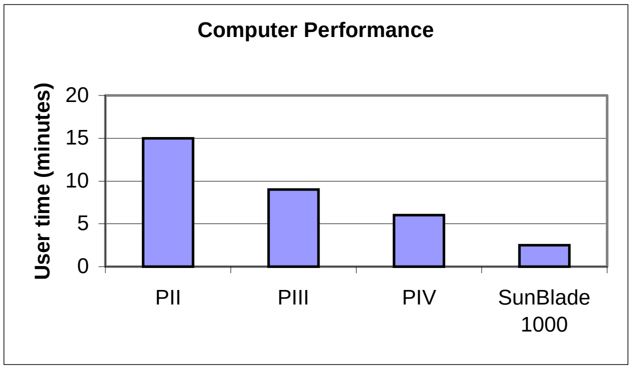

**Figure 16** Computer Performances.

## CONCLUSION

CFD is an important tool for all HVAC engineers. The model presents the complexity of the flow patterns that evolve

the effects of obstacles, geometry change and buoyant forces. Obstacles were shown to create stagnant zones, which affects the ventilation and reduces the IAQ. Buoyant plumes have be

exhaust. Instabilities due to heat sources can cause hot/cold spot feelings.

To test the model an office space was designed using a displacement and mixing ventilation system. Both systems are acceptable according to the ISO standard, and from the thermal comfort standpoint. The displacement system however had less turbulence, mixing and a stratified thermal environment. This is noticed by the uniformity of the PMV distribution inside the space.

The model is computationally efficient and may be used by HVAC designers to get a first insight on ventilation unit placements and ventilation effectiveness. With the advent of the Pentium IV processors the computational time is of the order of hours for three-dimensional simulations and minutes for two-dimensional simulations.

Three-dimensional models (not fully presented) add insight and depth to the flow patterns. The same effects of obstacles, buoyancy exists, besides to the complications that arise from confined spaces, such as flow turning and reversal at walls.

## REFERENCES

1.
simulation: an overview of developments and information
*Building and Environment*, Vol.35, p.347-361, 2000.
2. Lu, W.,
Airflow and Temperature Field in a Room with

*Building and Environment*, Vol.32, No.6, p.541-550, 1997.

3. *Building and Environment*, Vol.36,, p.351-358, 2001.
4. *Architectural Institute of Japan*, Japan, p.63, 1959.
5. *Building and Environment*, Vol.24, 1989, p.73-84.
6. *Procedure of CFD for Environmental and Building Services Engineer*, 1991, p.67-79, Institution of Mechanical Engineers, London
7. ll *Proc of Sixth International Conference on Indoor Air Quality and Climate (Indoor Air'93)*, Vol.5, 1993, p.283-288, Helsinki.
8. of the Local Age and the Local Purging Flow Rate in Two- , 10-12 June 1987, Session 3, p.1-14.
9. *Building and Environment*, Vol.22, No.2, 1987, p.111-127.
10. Davidson - Dimensional Room- *Building and Environment*, Vol.24, No.4, 1989, p.363-372.
11. of the Flow in a Three-Dimensional Ventilated Room *th International Conference on Air Distribution in Rooms,* , Yokohama, Japan, Vol.2, July 17-19 1996, p.161-168.
12. *Energy and Buildings*, Vol.23, p.73-81, 1995.
13. Chen, Q. and Moser, A. and Suter, P., of Indoor Air Quality and Thermal Comfort Under Six *ASHRAE,* Vol.98, Part 1, 1992, p.203-217.
14. Predicting Room Ai *ASHRAE*, Vol.98, Part 1 1992.
15. *ASHRAE Journal*, February 1986.
16. Fanger, P.E., and Melikov, H.K. and Hanzana, H. and *ASHRAE Journal*, April 1989.
17. t Equation for Indoor Air *ASHRAE Journal*, October 1989.
18. *Computer Methods in Applied Mechanics and Engineering*, Vol.#, p.269-289, 1974.
19. ansfer and Fluid
20. *Transactions of the Department of Mechanics/Applied Computational Fluid Dynamics*, Royal Institute of Technology, S-100 44 Stockholm, Sweden, August 1992.