Slot Ventilated Room with Heated Obstruction

Dr. Waheed S. Mohammed, **Dr. Wissam A. Mohammed and ***Asst. Lect. AbdulJabbar Muttair Ahmed

Received on:22/3/2007
Accepted on: 9/12/2007

Abstract

A numerical study of two-dimensional turbulent buoyant rcirculating flow within mechanically ventilated rooms is reported. The study involves the solution of elliptic partial differential equations for the conservation of mass, momentum, energy, turbulent energy and its dissipation rate in a finite difference form. These equations were solved together with algebraic expressions for the turbulent viscosity and heat diffusivity, using (K-ε) turbulence model. A modified version of a two-dimensional elliptic computer code was used to simulate the complex flows inside a slot-ventilated room. The present study demonstrates that the flow behavior depends on several parameters, such as airflow rate, size and temperature of the heated obstruction. Each of these parameters was modeled separately to understand their affects on the airflow characteristic inside ventilated room.

التهوية بفتحة خطية لغرفة بوجود مصدر حراري

الخلاصة

تم انجاز دراسة حاسوبية لجريانات دورانية مضطربة طفوية ثنائية البعد داخل غرفة يتم تهويتها قسريا. تتضمن الدراسة حل المعادلات الجزئية للقطع الناقص المتمثلة بحفظ الكتلة، الزخم، الطاقة و الطاقة المضطربة ومعدل ضياعها باستخدام تقنية الفروقات المحددة (Finite-Difference Technique)، ولقد حلت هذه المعادلات سوية مع الصيغ الجبرية للزوجة المضطربة (Turbulent Viscosity) وانتشارية الحرارة (Heat diffusivity) بأعتماد مود يل الاضطراب المسمى (K- ε). تم تطوير برنامج حاسوبي اهليليجي ثنائي البعد لحساب هذه الجريانات المعقدة داخل غرفة. تم دراسة غرفة يتم تهويتها عن طريق فتحة خطية مع وجود مصدر حراري بأحجام مختلفة. أن الدراسة الحالية قد أوضحت أن الجريان الداخلي يعتمد على عدة عوامل مثل سرعة الهواء المجهز، حجم ودرجة حرارة المصدر الحراري لقد تم دراسة هذه العوامل بصورة منفصلة لكل منها لمعرفة تأثيرها على خصائص توزيع الهواء داخل الغرفة المعنية.

* Mechanical Engineering Department.
** Technical Education Department.
*** Technical Education Department.

Symbols

a_E ,a_W , a_N , a_P a_S,	Coefficient of general Finite-difference equation
A_e,$A_w$$A_n$,$A_s$	Area of finite-difference Cell boundaries.
B	The source term in general finite-difference equation.
C_1, C_2, C_D	Coefficientin turbulence Model.
D_e,D_w,D_n,D_s	Diffusion terms.
E	Empirical constant.
F_e,F_w,F_n, F_s	Convection coefficients.
G_K	Kinetic energy generation by shear.
G_B	Kinetic energy generation by buoyancy.
g_I	Tensor notation for Gravitational Acceleration.
H	Height of heated obstruction.
H	Room height.
h_I	Height of inlet opening.
K	Turbulence kinetic energy.
L	Room length.
P	Static pressure.
Pe	Finite-difference cell Peclet number.
P_J	Jaytillaka's P-parameter.
R_Φ	Residual source.
S_Φ	General source term.
T	Temperature.
U_i,U_j	Tensor notation for Velocity components.
X_i, X_j	Tensor notation for Principle directions.
X, Y	Cartesian coordinates.
Y	Normal distance from wall.
Y^+	Dimensionless distance From the wall.

Greek symbols

B	Thermal expansion Coefficient.
Φ	The general dependent Variable.
ε	The dissipation rate of Turbulence.
σ_K , σ_ε	Turbulence Prandtl number for Diffusion of (K) and (ε).
σ, σ_t	Laminar and turbulent Prandtl number.
ν_t	Turbulent viscosity.
ν	Kinematic viscosity.
τ_w	Wall shear stress.
Γ	Diffusion coefficient.
∇	Gradient (vector operator).
δ_x,δ_y	Inter-nodes distance.
K	Von Karman constant.
Λ	Convergence criteria.
M	Dynamic viscosity.
P	Density.

Subscripts

Eff	Effective values.
In	Inlet value.
L	Laminar flow condition.
P,E,W,N,S	Central node and its Neighbors.
Ref	Reference value.
t	Turbulent flow condition.
τ	Friction velocity

Superscripts

*	Previous iteration value.
´	Correction to starred Values
+	Dimensionless value

1. Introduction

The air conditioning of a room serves several purposes to remove or supply heat in order to maintain a comfortable temperature level and to supply the room with a given amount of fresh air. Therefore, the practice of creating a controlled climate in indoor space (e.g. in home, schools, hospitals, hotels, commercial offices and even factories) is no longer luxury, but an essential part of the modern living and working. The four important factors for human comfort are temperature, humidity, purity and motion of the air [1]. In recent years a number of new ventilation systems and strategies have been introduced to improve air quality, thermal comfort and energy efficiency. This has changed the nature of current airflow problem [2]. Ventilation is defined as "controlled supply and exhaust of air to improve the indoor air quality". There are four different ventilation principles used to control the air distribution within ventilation rooms (unidirectional flow ventilation, mixing ventilation, displacement ventilation, and local exhaust ventilation)[3].

Recent studies have emphasized the importance of inter zone air movement in building and demonstrated the need for better understanding of this movement in any attempt to predict the air distribution system efficiency or thermal performance of the building. The environmental conditions such as airflow velocity and temperature distribution within a conditioner space may be obtained by flow visualization and measurements respectively. These kinds of investigations are usually elaborate, expensive to undertake, and sometimes inaccurate. Recent research turned toward an alternative approach based on the numerical solution of the governing flow equations for mass, momentum, and scalar properties. This is because recent advances in numerical calculation procedures for fluid flow and heat transfer have brought computer usage within reasonable cost limits for practical design problems. **Hjertager and Magnussen** [4] used the Patanker and Spalding algorithm [5] to solve the finite difference equations for velocity components and continuity. They also used the (K-ε) turbulence model. This numerical study is performed to predict unsteady, turbulent flow in a ventilated box-shaped empty room. **Nielsen et al.** [6] carried out a study to assess the accuracy of the numerical procedure in predicating two-dimensional isothermal **Awbi** [7] applied a numerical solution to predict the air movement in square plan room ventilated by a continuous slot diffuser across the width of the ceiling and at a distance 1.2 m from a wall discharging towards the far wall. Two-dimensional non-isothermal solutions were used and this produced realistic predictions of the vertical velocity and temperature profiles when compared with measurements in the room. **Davidson** [8] investigated displacement ventilation room using finite difference numerical procedure together with a hybrid scheme. Comparison was made using experiment conducted in a water box model. The Boussinsque approximation was used for the buoyancy effect in the vertical component of the momentum. He concluded that the flow was well predicated in comparison with experiment.

The present work was aimed to study the behavior of buoyant flow in ventilated room in the presence of a heated obstruction. The effect of flow rate, size and temperature of heated obstruction on such behavior will be included. Such flows are turbulent, recirculating and elliptic in nature. The study will be carried by numerical simulation of the governing

equations (i.e. mass, momentum, energy, turbulence kinetic energy and its dissipation rate) using a finite difference technique with a hybrid differencing scheme to solve these equations. The (K-ε) model of turbulence was employed in these simulations. Momentum and temperature field will be numerically computed for such flows using a procedure similar to the SIMPLE algorithm of Patanker and Spalding. The computational procedure will then be modified to simulate the present problem. These modifications may be summarized by the geometrical shape, size of a heat obstruction, buoyancy effect and the boundary conditions. The results of the flow field, temperature distribution will then be presented accordance with human comfort inside a ventilated room.

3. Mathematical Formulation

The present problem governs by the principles of conservation of mass, momentum and energy. These principle can be expressed in terms of (PDE′S) for incompressible, two dimensional, turbulent flow as follow [9].

i. Continuity Equation

$$\frac{\partial U_i}{\partial X_i} = 0$$

ii. Momentum Equation

$$\frac{\partial}{\partial X_j}(U_i U_j) = -\frac{\partial}{\partial X_i}\left(\frac{p}{\rho}\right) + g_i + \frac{\partial}{\partial X_j}\left(\nu_{eff}\left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right)\right)$$

iii. Energy Equation

$$\frac{\partial}{\partial X_j}(U_i T) = -\frac{\partial}{\partial X_i}\left(\Gamma_{eff}\frac{\partial T}{\partial X_j}\right)$$

here ν_{eff} is the effective viscosity

$$\nu_{eff} = \nu + \nu_t$$

The main difficulty in solving these equation is the determination of the turbulent viscosity (ν_t), therefore it can be expressed by "turbulence model"

3.1 Turbulence model

In present study the (K-ε) turbulence model has been used. The (PDE) for the turbulent kinetic energy (K) and its dissipation rate (ε) are as follow [9]:

$$\frac{\partial}{\partial X_i}\left[U_i K - \left(\nu + \frac{\nu_t}{\sigma_K}\right)\frac{\partial K}{\partial X_i}\right] = G_K + G_B - \varepsilon$$

$$\frac{\partial}{\partial X_i}\left[U_i \varepsilon - \left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial X_i}\right] = \frac{\varepsilon}{K}\left[C_1(G_K + G_B) - C_2\varepsilon\right]$$

where

$\sigma_K, \sigma_\varepsilon$ are turbulent Prandtl number for (K) and (ε).

$$G_K = \nu_t\left[\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right]\frac{\partial U_i}{\partial X_j} \equiv \text{Kinetic energy generation by shear}$$

$$G_B = -\frac{\nu_t}{\sigma_t}\frac{g\beta}{}\frac{\partial T}{\partial y} \equiv \text{Kinetic energy generation by buoyancy}$$

The turbulent viscosity (ν_t) can now be

C_D	C_1	C_2	σ_K	σ_ε
0.09	1.44	1.92	1.0	1.3

calculated from the values of (K) and (ε)

$$\nu_t = C_D\frac{K^2}{\varepsilon}$$

where

C_1, C_2, C_D are the turbulence model constant and given by [9]:

3.2 Wall Function

The wall function formulation of Lunder and Spalding [10] which is used to provide predictions of the flow within the viscous layer adjacent to the wall, assumes that the flow in this region behaves locally as a one- dimensional Couette flow. The variation of the resultant velocity (U_P) in the near wall region is assumed to follow a logarithmic law. For a smooth wall this law can be expressed in the form :

$$U_P^+ = \frac{U_p}{U_\tau} = \frac{1}{\kappa}\ln\left(E\ Y^+\right)$$

where:

$$U_\tau = \frac{\tau_w}{\rho\, C_D^{1/4} K^{1/2}}$$

and

$$Y^+ = \frac{C_D^{1/4}\rho\, K^{1/2}\ Y_p}{\mu}$$

Large temperature gradients are frequently present in the near wall region of turbulent flows. The distribution of temperature of this region is assumed to be analogous to that for velocity (U_P^+). It may be written in the form :

$$T_p^+ = \sigma_t (U_p^+ + P_J)$$

where (P_J)is the viscous sublayer resistance, which is given by [11]

$$P_J = 9.0\frac{\left[\frac{\sigma}{\sigma_t} - 1\right]}{\left[\frac{\sigma}{\sigma_t}\right]^{1/4}}$$

where σ and σ_t are the laminar and turbulent Prandtl number respectively.

3.3 The General Form of the (PDE′S)

The equations to be solved could be expressed in the general form

$$\frac{\partial}{\partial X_i}(\rho\Phi U_i) = \frac{\partial}{\partial X_i}\left(\Gamma_\Phi \frac{\partial \Phi}{\partial X_i}\right) + S_\Phi$$

where:

$$\frac{\partial}{\partial X_i}(\rho\Phi U_i) \equiv \text{Convection term}$$

$$\frac{\partial}{\partial X_i}\left(\Gamma_\Phi \frac{\partial \Phi}{\partial X_i}\right) \equiv \text{Diffusion term}$$

$$S_\Phi \equiv \text{Source term}$$

Table (1) diffusion coefficients and source for each variable

Equation	Φ	Γ_Φ	S_Φ
Continuity	1	0	0
x-direction momentum	U	v_{eff}	$-p_x + \nabla.(v_{eff}\nabla.U) + g_x$
y-direction momentum	V	v_{eff}	$-p_y + \nabla.(v_{eff}\nabla.U) + g_y\beta(T - T_{ref})$
Temperature	T	$\frac{v}{\sigma} + \frac{v_t}{\sigma_t}$	
Turbulence energy	K	$\frac{v_{eff}}{\sigma_K}$	$G_K - \varepsilon + G_B$
Dissipation Rate	ε	$\frac{v_{eff}}{\sigma_\varepsilon}$	$C_1\frac{\varepsilon}{K}(G_K + G_B) - C_2\frac{\varepsilon^2}{K}$

4. Numerical Solution Procedure

The numerical integration technique used in the present study is "Finite Difference Technique" which includes (dividing the flow domain into many small regions called the "control volumes" and integration of the general (PDE) over these volumes to obtain general (FDE) of the variables which is:

$$a_P\Phi_P = \left(\sum_{E,W,N,S} a\Phi\right) + b$$

where (b) constant term in (FDE) and a_N, a_S, a_E, a_w are neighbor coefficients which represent the convection and diffusion flux at the control volume faces. To formulate the equations that describe the neighbor coefficients a "Differencing Scheme" must be used. The hybrid scheme is adopted in the present study where the coefficient equations are as follow

$$a_E = \text{Max}(-F_e, D_e - F_e/2, 0)$$
$$a_W = \text{Max}(F_w, D_w + F_w/2, 0)$$

The adoption of a staggered grid for the velocity components ensures the calculation of the pressure differences acting on the staggered central volumes. So the momentum equations become

$$a_P U_P = \left(\sum_{E,W,N,S} a\ U\right) + b + A_e(P_P - P_E)$$

$$a_P V_P = \left(\sum_{E,W,N,S} a\ V\right) + b + A_n(P_P - P_N)$$

The resulting velocity components will not satisfy the continuity equation so, the

pressure-correction equation is used which can be written as

$$a_P\ P'_P = \left(\sum_{E,W,N,S} a\ P' \right) + b$$

where

$$b = \left(\rho A U^*\right)_w - \left(\rho A U^*\right)_e + \left(\rho A V^*\right)_s - \left(\rho A V^*\right)_n$$

(b) represent the residual in the continuity equation (mass source). when (b = 0) that means the velocity components satisfy the continuity equation.

The (PDE′S) must be solved to obtain (U, V, T, K, ε, P′) for all the grid nodes in the field because of non-linearity and strongly interlink. Therefore, SIMPLE, iterative procedure of Patankar and Spalding is used to solve these equations. Line by line technique with block-adjustment procedure and (TDMA) is used for solving (PDE′S).

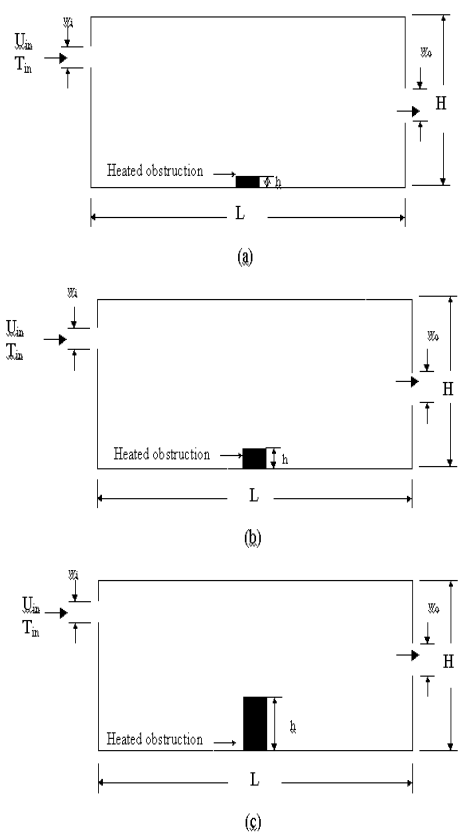


Fig.(1) Schematic diagram of a slot-ventilated room with heated obstruction (a) h/H = 0.11 (b) h/H = 0.13 (c) h/H = 0.4

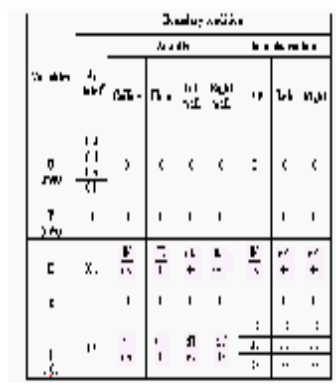
4.1 Convergence of the Numerical Solution

The convergence criterion for the numerical solution applied in the present study is based on the residual R_Φ of any variable (Φ) at node (P). The numerical solution reaches the convergence when $\sum_{142\,43} |R_\Phi| \le \lambda$.

for all field

(λ convergence criteria). In the present study $\lambda = 10^{-3}$.

Table (2) Boundary conditions for the slot ventilated room with heated obstruction.



5. Computations

One of the most important aims for ventilating a space with air is to maintain an acceptable air motion and temperature in the working of occupation zone. A slot-ventilated room has been widely used

to give such conditions. This room with heated obstruction is shown in Figure (1). The flow generated in such a geometry is a two dimensional turbulent recirculation buoyant one. The effect of ventilation flow rate on the flow characteristic in the occupation zone was studied first, followed by the effect of the size and temperature of the heated obstruction.

5.1 The Geometry Considered and Its Boundary Conditions

The geometrical arrangement that has been tested in the present work has a dimensions of (5.5 m) wide and (2.4 m) high with inlet opening [w_i = 0.1 m] and outlet opening [w_o = 0.3 m]. The values for velocity component, temperature, turbulent kinetic energy and its dissipation rate at all the solid boundaries are given in table (2).

Figure (2) shows the computational grid employed for the present solution of slot-ventilated room with a heated obstruction utilized (28 × 27) grids with an expansion between adjacent nodes on both the x and y directions.

5.2 Result and Discussion

There are many factors that affect the flow characteristic and temperature distribution inside the occupation zone of a slot-ventilated room. The results of the effect of some of these factors are discussed below.

5.2.1 The Effect of Ventilation Rate

The effect of ventilation rate has been studied for several flow rates (ie 5,10,16 and 21) air changes per hour (ACH). Figure (3a-b) shows the results of both 5 ACH and 10 ACH. The cold air is directly drops down and leads to high velocity near the heated obstruction. Figure (4a) shown an acceptable flow pattern since the vertical temperature gradient is less than 3 ˚C as specified by ASHRAE [12]. The results of the other two air changes (16 and 21) ACH are shows in Figure (5a-b). The increasing of ACH to 16 and beyond that causes the cold air to attach to the ceiling and cause a wall-jet flow. Figure (6a-b) shows that the flow pattern is not acceptable in this case since the vertical temperature gradient is greater than 3˚C. Therefore as the ventilation rate is increased, the flow pattern and flow characteristic give an unacceptable level of comfort.

5.2.2 Effect of the Size of the Heated Obstruction

The results of the effect of the size of the heated obstruction represented as (h/H = 0.11, 0.13 and 0.4) with airflow rate equal to 5-ACH are shown in Figure (7). Figure (7a) shows the results of (h/H = 0.11) in which the jet is attached to the ceiling and flow as a wall jet. This physically means that the cold jet will exit directly to the outlet with a little heat exchange inside the comfort zone. Figure (8a) shows the temperature contour of this flow pattern and indicates that a high temperature gradient exists in the comfort zone.

Figure (7b) shows the result of [h/H = 0.13]. In this figure the jet is dropped down attaching the left wall. The increasing effect of the heated obstruction size causes the flow above the heated obstruction to rise upward making two recirculation zones inside the comfort region. This effect is clearly shown in the temperature contour of Figure (8b).

Figures (7c & 8c) show the increasing effect of heated obstruction size of [h/H = 0.4]. The heated obstruction in these figures works as divides the flow into two regions. The temperature distribution

within the comfort zone is acceptable in both zones.

5.2.3 Effect of Temperature of the Heated Obstruction

The effect of temperature of the heated obstruction on the jet characteristic and temperature distribution inside the ventilation is presented in Figures (9-10). The values of heated obstruction temperature considered were (17, 25 and 35 ˚C). This means that a temperature difference of (2, 10 and 20 ˚C) between the temperature of the heated obstruction and the inlet flow. The heated obstruction size is taken (h/H = 0.11) in these figure which gives ACH of 5. It is clear from Figure (9) that as the temperature of the heated obstruction increases the flow pattern changes from the wall jet to a two-region recirculation zone due to buoyancy effect.

6. Conclusions

A numerical study was conducted to predict the influence of ventilation flow rate on the flow characteristic in the occupation zone and the effects of the size and temperature of a heated obstruction on a slot ventilated room. The hybrid differencing scheme was used in formulating the relevant finite-difference coefficient for all variables. The line by line technique was adopted for this case. The computed flow are presented in graphical form as flow pattern and temperature contour to show the effect of ventilation rate on the air movement and temperature distribution inside the room. The study also include the effect of the size and temperature of the heated obstruction.

The following remarks are deduced.

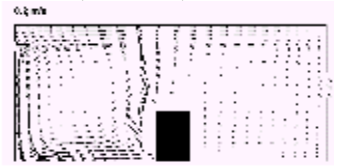
i. Increasing the airflow rate beyond 16-ACH will produce an uncomfortable condition inside the room.
ii. The presence of heat obstruction affects the flow pattern inside the room. For the same inlet velocity and temperature difference the best size of heated obstruction is h/H (0.4) which gives acceptable air movement and temperature distribution inside the comfort zone.
iii. The most critical effects are those of temperature differences. When the size of the heated obstruction is h/H (0.11) and flow rate equal to (5 ACH), the best temperature difference between the inlet and heated obstruction was found to be 20 ˚C.
iv. Finally, the present computational procedure can be regarded as a good tool to simulate complex buoyant flow inside the room with different sizes of the heated obstruction.

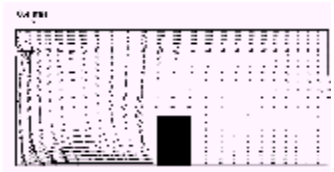
References

1. Khurmi, R. S. and Gupta, J. K. " Text Book of Refrigeration and Air Conditioning", Eurasia Publishing House, New Delhi, 1987.
2. Rutten, P. G. S, Wisse, J. A, Cheen, Q. "The Measurement and Simulation of Indoor Air Flow" Research is partially funded by a grant from the Dutch Origination for Scientific Research (Now-Stimulans), December 1998.
3. Zou Yue. "Air Jet Velocity Decay in Ventilation Applications" Building Services Engineering Royal Institute of Technology (KTH) Stockholm, Sweden, August 1999.
4. Hjertager B. H. and Magnussen B. F., "Numerical Prediction of Three Dimensional Turbulent Buoyant

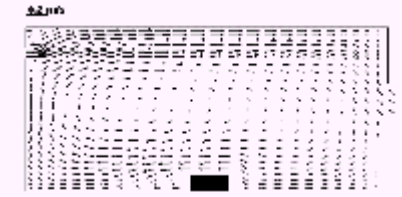
Flow" Heat Transfer and Turbulent Buoyant Convection, Hemisphere, Washington Vol. 2, PP 429-441, 1972.

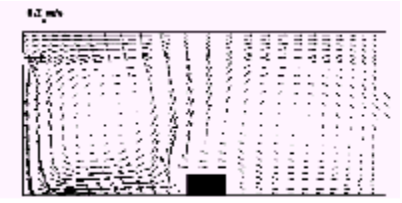
5. Patankar S. V. and Spalding D.B., " A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flow", Int. Journal of Heat and Mass Transfer, Vol.15, pp.1787-1806, 1972.
6. Nielsen P. V., Restivo A. and Whiteklaw J. H., "The Velocity Characteristics of Ventilated Room", ASME, J. Fluid Eng., Vol. 100, 1978, PP291-298.
7. Awbi, H. B. "Application of computational fluid dynamics in room ventilation", Building and Environmental. Vol 24, No.1, PP 73-84,1989.
8. Davidson, L., "Ventilation By Displacement in A Three Dimensional Room A numerical Study", Building and Environmental, Vol, 24 No.4, 1989, PP (363-372).



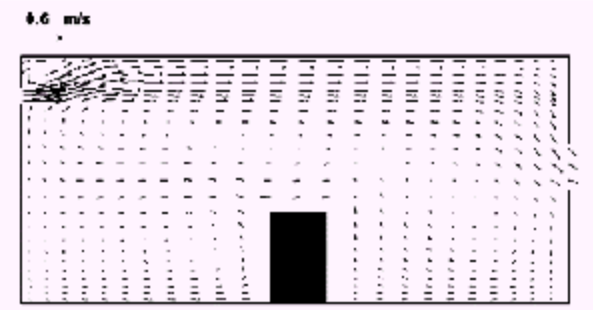


9. Jane W. Z Lu, Isaac Y. F Lun, Joseph C Lam "CFD Investigation of Airflow Around Objects With Different Shapes" Department of Building and Construction , City University of Hong Kong, 83 Tat Chee Avenue, Kowloon Tong, Kowloon , Hong Kong, 2004
10. Launder B.E and Spalding D.B " The Numerical Computation of Turbulent Flows" Imperial College of Science and Technology, Department of Mechanical Engineering, London,1973, UK.
11. Awbi H. B. "Ventilation of Buildings" Senior Lecturer Department of Contraction Management and Engineering First published, by E & FN spon 1991.
12. Basman Mohamed Nabil El Hadidi " Computational Study of Flow in Mechanically Ventilated Space" Seventh International Conference of fluid dynamic and propulsion. December 19.21, 2001, Sharm El. Sheikh, Egypt.
13. James W. Weather " A Study of Computational Fluid Dynamics Applied to Room Air Flow" MSc. Thesis-Oklahoma University Of Science and Art, 1990.

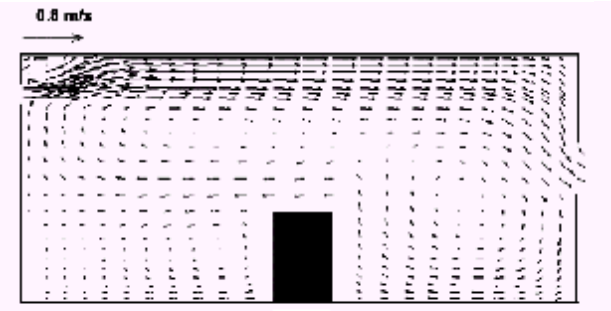




(b)

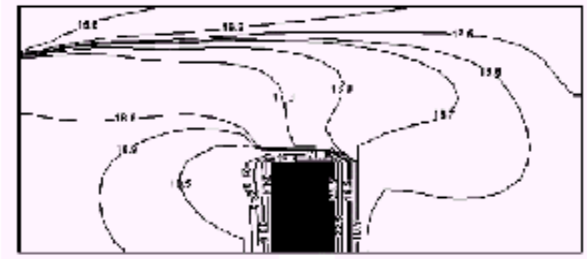


(a)

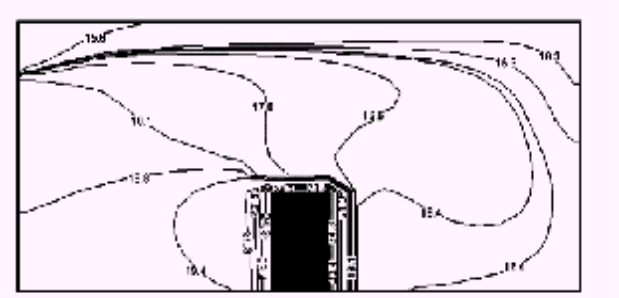


(b)

Fig.(5) Flow pattern of buoyant flow for (a) 16-ACH (b) 21-ACH

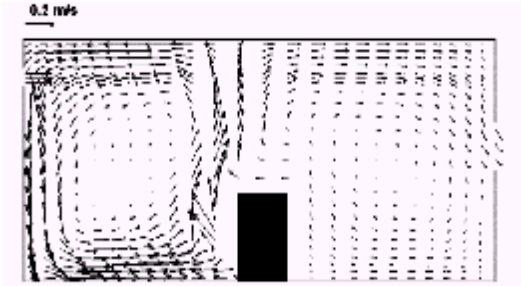


(a)



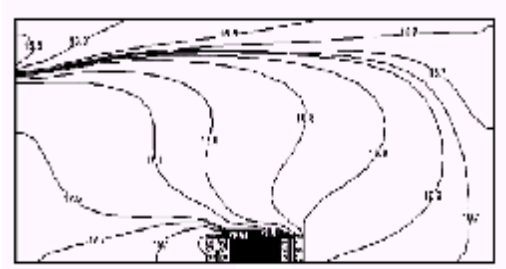
(b)

(6) Temperature contours of buoyant flow for (a) 16-ACH (b) 21-ACH

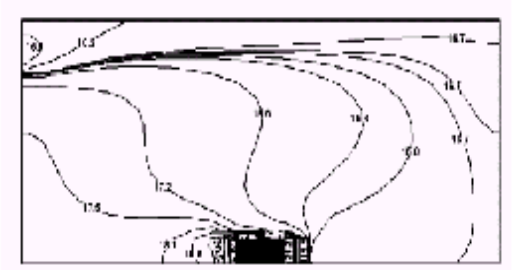


(c)

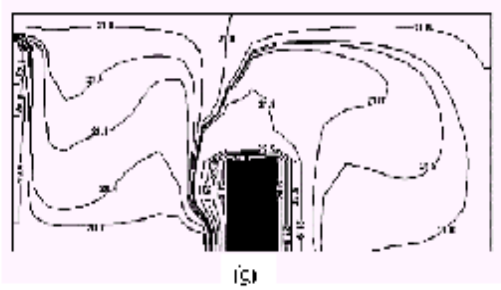
Fig.(7) Flow pattern of buoyant flow (different size of the heat obstruction h/H) for 5-ACH (a) h/H = 0.11 (b) h/H = 0.13 (c) h/b = 0.4



(a)



(b)



(c)

Fig.(8) Temperature contours of buoyant flow (different size of the heated obstruction h/H) for 5-ACH (a) h/H = 0.11 (b) h/H = 0.13 (c) h/H = 0.4

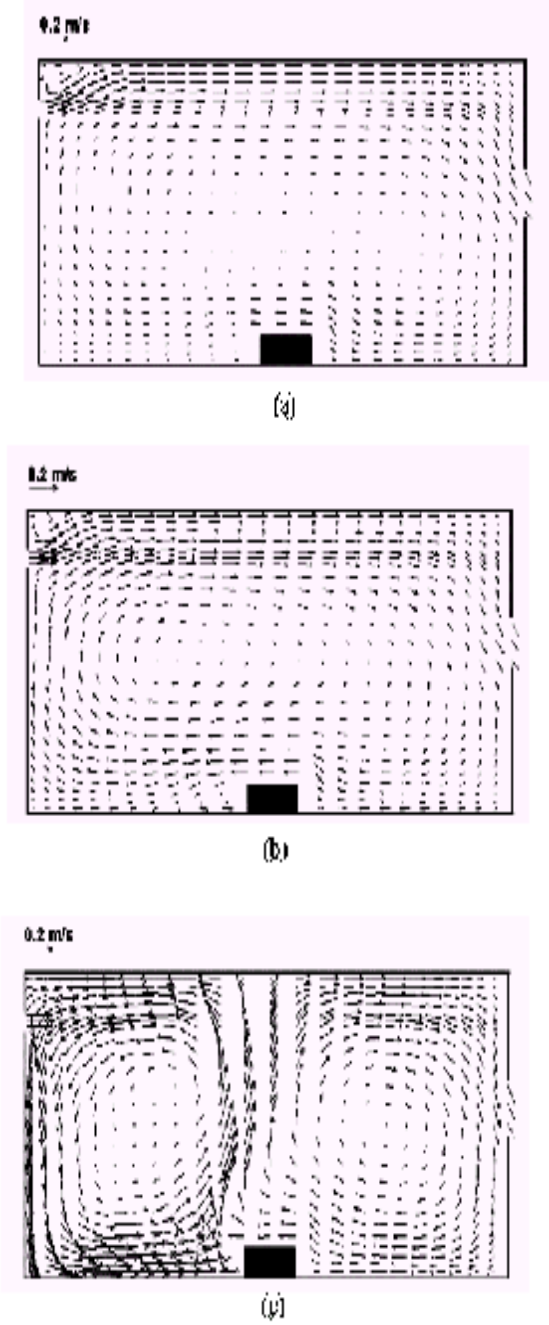


Fig.(9) Flow pattern for different heat obstruction temperatures of 5-ACH and heated obstruction size h/H = 0.11 (a) 17 °C (b) 25 °C (c) 35 °C

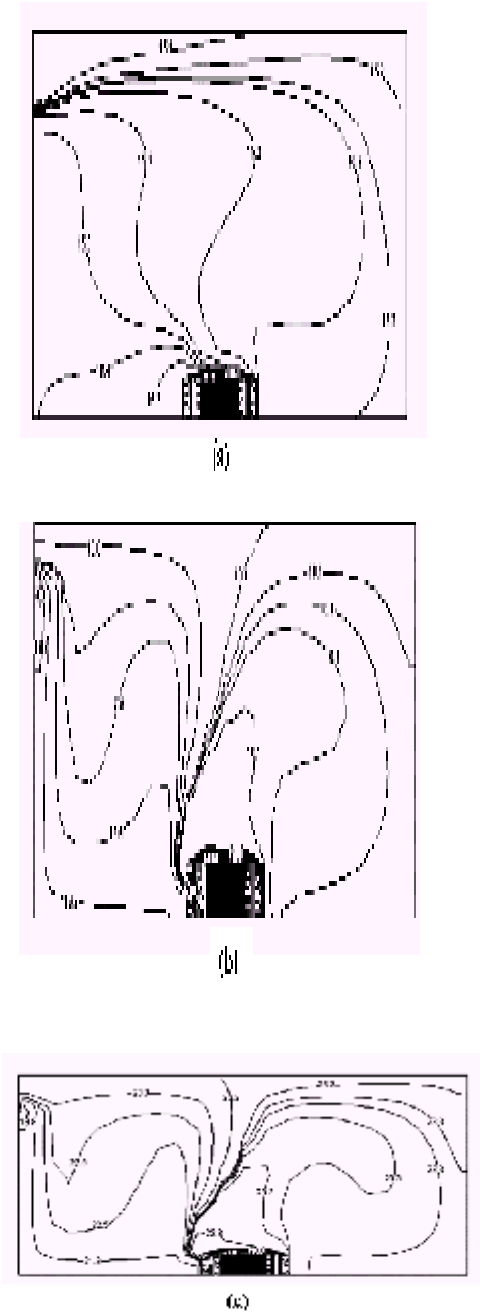


Fig.(10) Temperature contours for different heated obstruction temperatures of 5-ACH and heated obstruction size h/H = 0.11 (a) 17 °C (b) 25 °C (c) 35 °C