

## Numerical Study of Flow In Mechanically Ventilated Rooms Under Non- Isothermal Conditions

Prof. Dr. Jalal M. Jalil\*, Asst.Prof. Dr. Talib K. Murtadha\* and Hanaa M. Mohammed Ali\*

Received on: 5/4/2005

Accepted on: 22/1/2007

### Abstract

A computational procedure was conducted in (3x3x3 m) rooms and in a full-scale model building to study the effect of supply temperature on the room flow patterns in mechanically ventilated spaces. The procedure is based on the solution in finite-volume form of 3D equation for the conservation of mass, momentum, energy, kinetic energy and dissipation rate. Effect of buoyancy on the turbulence model has been included. The results are shown to be in reasonable agreement with experimental data.

### الخلاصة

لقد اجريت حسابات عددية لدراسة تجهيز هواء بدرجات حرارية مختلفة لغرف بمقياس كامل (3, 3, 3) متر بصورة ميكانيكية. اعتمد تكنيك الحسابات على حل معادلات حفظ الكتلة والزخم و الطاقة وبالأبعاد الثلاثة مع معادلات (k, e) للجريان الاضطرابي. وقد شمل البحث تأثير الطفو على موديل الاضطراب. التوافق بين النتائج العددية و البيانات التجريبية كان معقولا.

### NOMENCLATURE

$C_{1e}, C_{2e}, C_{\mu}$  = Turbulent Coefficients.  
 $g$  = Acceleration of gravity.  
 $K$  = Kinetic energy.  
 $P$  = Pressure.  
 $G_k$  = Turbulent Production.  
 $T$  = Temperature.  
 $U$  = Velocity Component.

### GREEK

$\varepsilon$  = Dissipation.  
 $\Gamma$  = Diffusivity.  
 $\mu$  = Viscosity.  
 $\rho$  = Density.  
 $\sigma_k, \sigma_e, \sigma_{\mu}, \sigma_t$  = Turbulent Coefficients.

### Introduction

The major aim of ventilating spaces for human occupancy is find cleaner, safer homes that are cost effective and energy wise efficient [1]. Recent studies indicate that 80% of our life is present indoors and that about one

third of the world's energy is utilized for heating and cooling [2]. Predication of the distribution of velocity, temperature in a ventilated space is important for designing air ventilation system, by studies of building thermal analysis, indoor air

\* Dept. of Technical Education, UOT., Baghdad-IRAQ.

quality and thermal comfort [1]. CFD provides a cost effective method to predict flow field in buildings. Developments of CFD for building simulations were done by Awbi et al. [3,4], Haghighat [1], Davidson [5] and others. Awbi simulated flows in two and three- dimensional office spaces for turbulent flows using the standard (k-ε) method. The studies show the importance of understanding effects of buoyant flows where instabilities may occur.

Haghighat [1] simulated the pattern of isothermal flow caused by infiltration and ventilation in three- dimensional multi-zones by numerical simulation and that zones were separated by a partition with a door opening. The result shows that the location of the door not only guides the direction of the air movement but also affects the strength of the air circulation. Batzanas and Kittas [6] studied the effect of different ventilation types (sizes and position) on ventilation rates, air flow temperature distribution in tunnel greenhouse and conclude that the highest ventilation rates are not always the best solution to cool the green house. In this paper, the study of the effect of supply temperature on airflow patterns inside a room was performed using (k-ε) turbulent model. Comparison of code performance with available measurements is presented.

#### MATHEMATICAL MODEL

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

$$\frac{\partial}{\partial x_i}(\rho f u_i) = \frac{\partial}{\partial x_i} \left[ \Gamma_f \frac{\partial f}{\partial x_i} \right] + S_f \quad (1)$$

where  $\phi$  is the dependent variable. Table (1) gives the expression for the source terms  $S_\phi$  for each variable that is likely to be needed in solving problems. The constants for the (k-ε) method are the standard constant proposed by Launder and Spalding [7].

$$\begin{aligned} \sigma k &= 1.0 & \sigma \varepsilon &= 1.3 \\ \sigma \mu &= 0.09 & \sigma t &= 0.9 \\ C_{1\varepsilon} &= 1.44 & C_{2\varepsilon} &= 1.92 \end{aligned}$$

The turbulent viscosity and turbulence production terms are,

$$\mu_t = \frac{c_\mu \rho k^2}{\varepsilon} \quad (2)$$

$$G_k = \mu_t \frac{\partial u_i}{\partial x_j} \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \quad (3)$$

The simple (Semi Implicit Method for Pressure Linked Equation) scheme proposed by Patankar [8] is applied. This finite volume formulation applies staggered grid for the pressure variable to avoid pressure instabilities. The iterative sequence was carried out using a point-by-point technique, with correction of the velocity components, inflow velocity, turbulence and dissipation levels, temperature and pressure are specified. Along the boundaries the standard wall- function is used [9].

**Table 1**  
**Source Terms in the Transport Equations**

Equation	$\phi$	$\Gamma_\phi$	$S_\phi$
Continuity	1	0	0
Momentum	$u_i$	$\mu_e$	$-p_i + \nabla \cdot (\mu_e \nabla u) + \rho g_i$
Temperature	T	$\Gamma_e$	$\overline{G_k - \rho \epsilon}$
Kinetic energy	$\kappa$	$\Gamma_k$	
Dissipation rate	$\epsilon$	$\Gamma_\epsilon$	$C_{1\epsilon} \frac{\epsilon}{\kappa} (G_k) - C_{2\epsilon} \rho \frac{\epsilon^2}{\kappa}$

### CODE VALIDATION

To validate the code program for turbulent non-isothermal flow, flow through a rectangular shaped room was used. Figure (1) shows velocity vectors for fine grid systems, it is also show the comparison of the present results with that obtained by Weather [10], the figure shows acceptable agreement.

### RESULTS

A room (3 x 3x 3 m), having an inlet and outlet area equal to (0.5 x 0.5 m) with two windows is used. Effect of supply temperature is studied, one is a supply of warm air and the other is a supply of cold air.

#### Supply of warm air

For ventilation rate 15 air changes per hour (ACH), the supply temperature is fixed at 30°C and the room temperature was initially fixed at 20°C respectively. Figure 2 shows the velocity vectors and temperature contours for that case.

#### Supply of cold air

For ventilation rate 8 (ACH), the supply temperature is fixed at 20°C and the room temperature, was

initially fixed at 30°C and 25°C respectively. Figures 3 and 4 show the velocity vectors and temperature contours for that case.

### DISCUSSION

For 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. Shows Figure 2, an upward plum, with large recirculation vortex. This vortex mixes the air in the room. This is an example of an inefficient form of heating, where the heating energy is lost with out effective heating to the room. The effect of buoyancy is great forcing the flow to deflect upwards sharply, with a large primary vortex, circulating in the space. We can conclude that that effect of velocity can not be taken alone to state whether mixing flow pattern is present or not.

For the supply of *cold air* it is possible for the jet to drop into the occupied and caused unacceptable high velocities. As seen in Figure 3 the effect of the cold air is to deflect the flow downwards decreasing the “jet throw”. This may lead to thermal discomfort for the occupants under the jet. It is seen from Figure 4, that

the velocity vectors and temperature contours showing stratified flows. This case is acceptable according to the ASHRAE, and from the thermal comfort standpoint. It may be conclude that negative buoyancy may lead to high velocities in the flow field. The air tends to drop down, and force the main flow to deflect. In Figures 3 and 4, increasing the velocity and decreasing the buoyancy leads to more mixing. The size of the primary vortex varies greatly inside the space and it is not possible to conclude whether increasing negative buoyancy helps in more mixing or not, because changing the outlet side may change the result.

It is important to note that heating, cooling effects may change the flow path. Buoyancy effects may "short circuit" the flow path and lead to plumes forming upwards or downwards, with large circulation zones.

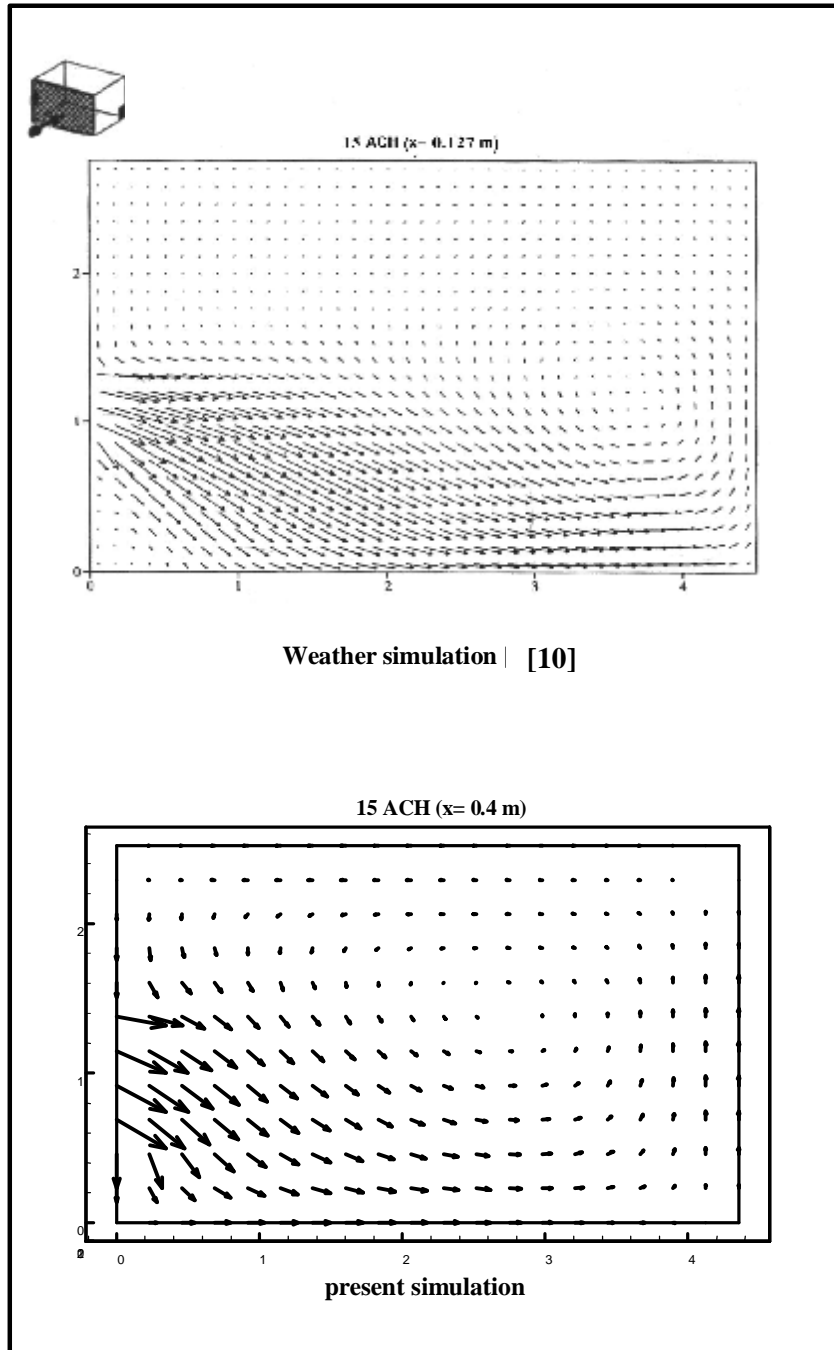
## REFERENCES

1. Haghighat F., Wang J.C.Y. and Jiangz., "Three-Dimensional Analysis of Air Pattern and Contaminant Dispersion in Ventilated Two-Zone Enclosure", ASHRAE Trnsaction, Vol.96 Pt1, 1990.
2. Tianzhen Hong H., Chou S. K. and Bong T.Y., "Building Simulation : an over view of developments and information sources", Building and Environment, Vol.35, P.347-361, 2000.
3. Awbi H.B. "Application of Computational Fluid Dynamics in Room Ventilation", Building and Environment, Vol.24, PP (37-84), 1989.
4. Awbi H.B. and Gan G., "Computational Fluid Dynamics in Ventilation", Procedure of CFD for Environmental and Building Services Engineer, PP (67-79), Institution of Mechanical Engineers, London, 1991.
5. Davidson L. "Ventilation By Displacement in a Three-Dimensional Room-A numerical Study", Building and Environment, Vol. 24, No 4, PP (363-372), 1989.
6. Bartzanas T. and Kittas C.. "Optimization of Green House Ventilation Performance with Computational Fluid Dynamics", Lab. Of Agriculture Constructions and Environmental Control, University of Thessaly.
7. Launder, B. and Spalding B., "The Numerical Computation of Turbulent Flows", Computer Methods in Applied Mechanics and Engineering, PP (269-289), 1974.
8. Patankar S.V., "Numerical Heat Transfer and Fluid Flow,", Hemisphere Publishing Corporation, 1980.

9. Murakami S., Kato S. and  
Ishida Y. "3-D Numerical  
Simulation of Turbulent  
Airflow in and Around  
Building Based on the (k- $\epsilon$ )  
Model with Generalized  
Curvilinear Coordinates".  
ASHRAE Transaction.

Vol.95. Pt.2, PP(30-53),  
1989.

10. Weather J. W., "Study of  
Computational Fluid Dynamics  
Applied to Room Airflow",  
MSc. Thesis-Oklahoma  
University of Science and  
Art, 1990.



**Figure ( 1) Comparisons of velocity vectors of  
present simulation and weather [10]**

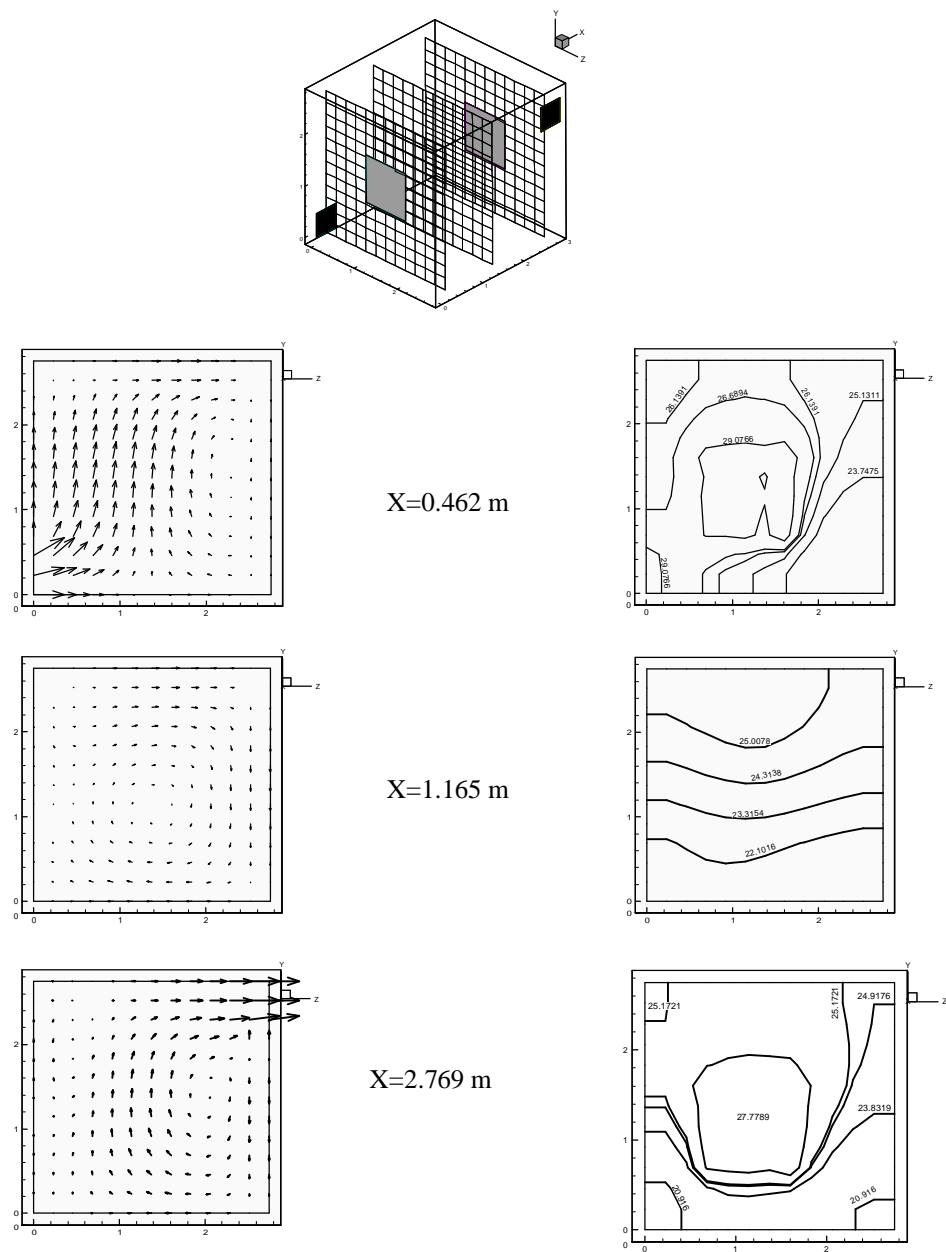


Figure (2) Flow pattern and temperature distribution of (warm air supply) for 15 ACH ( $x=0.462$ ,  $1.165$  and  $2.769$  m)

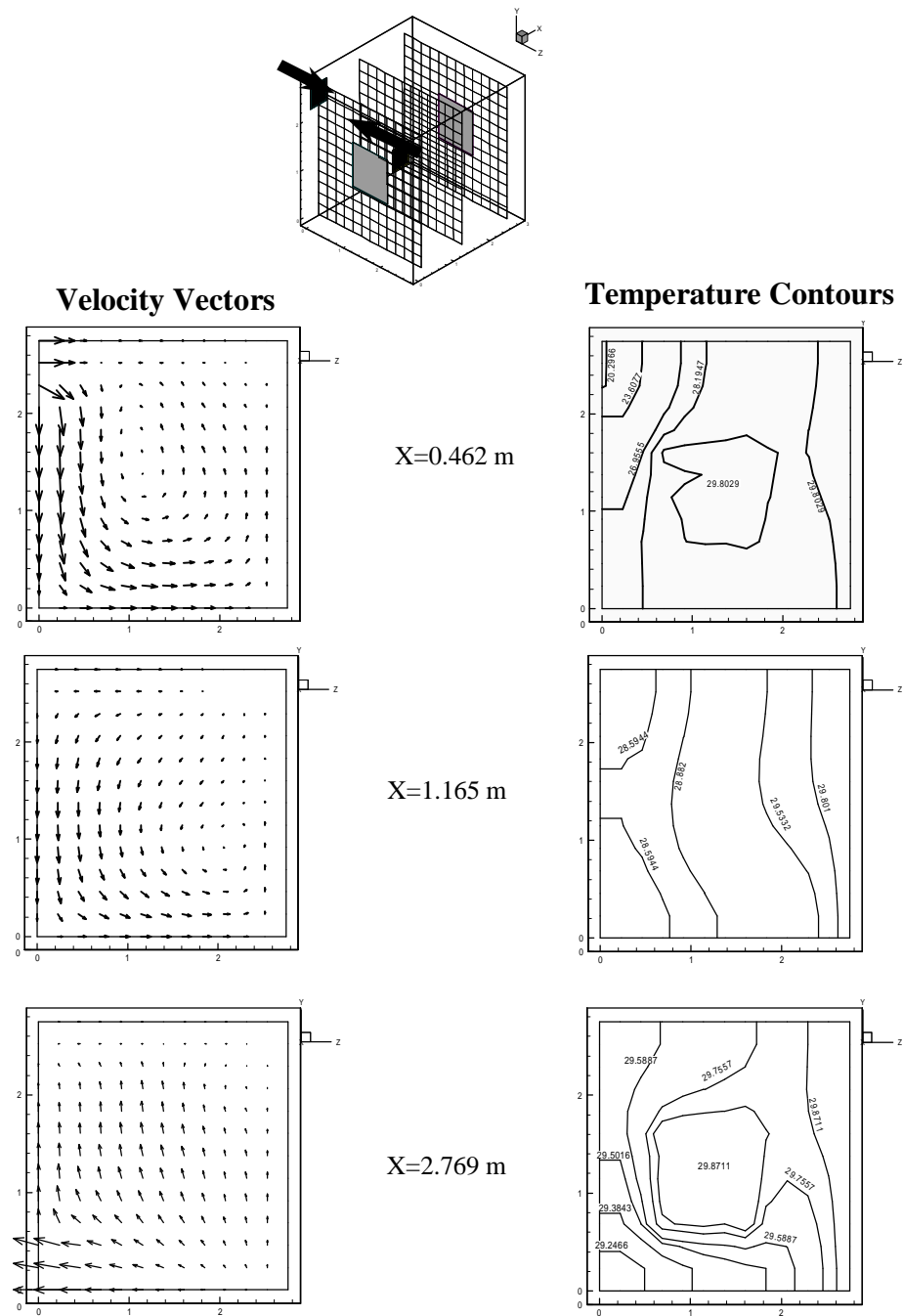


Figure (3) Flow pattern and temperature distribution of ( cold air supply)  
for 8 ACH ( $x=0.462, 1.165$  and  $2.769\text{ m}$ )



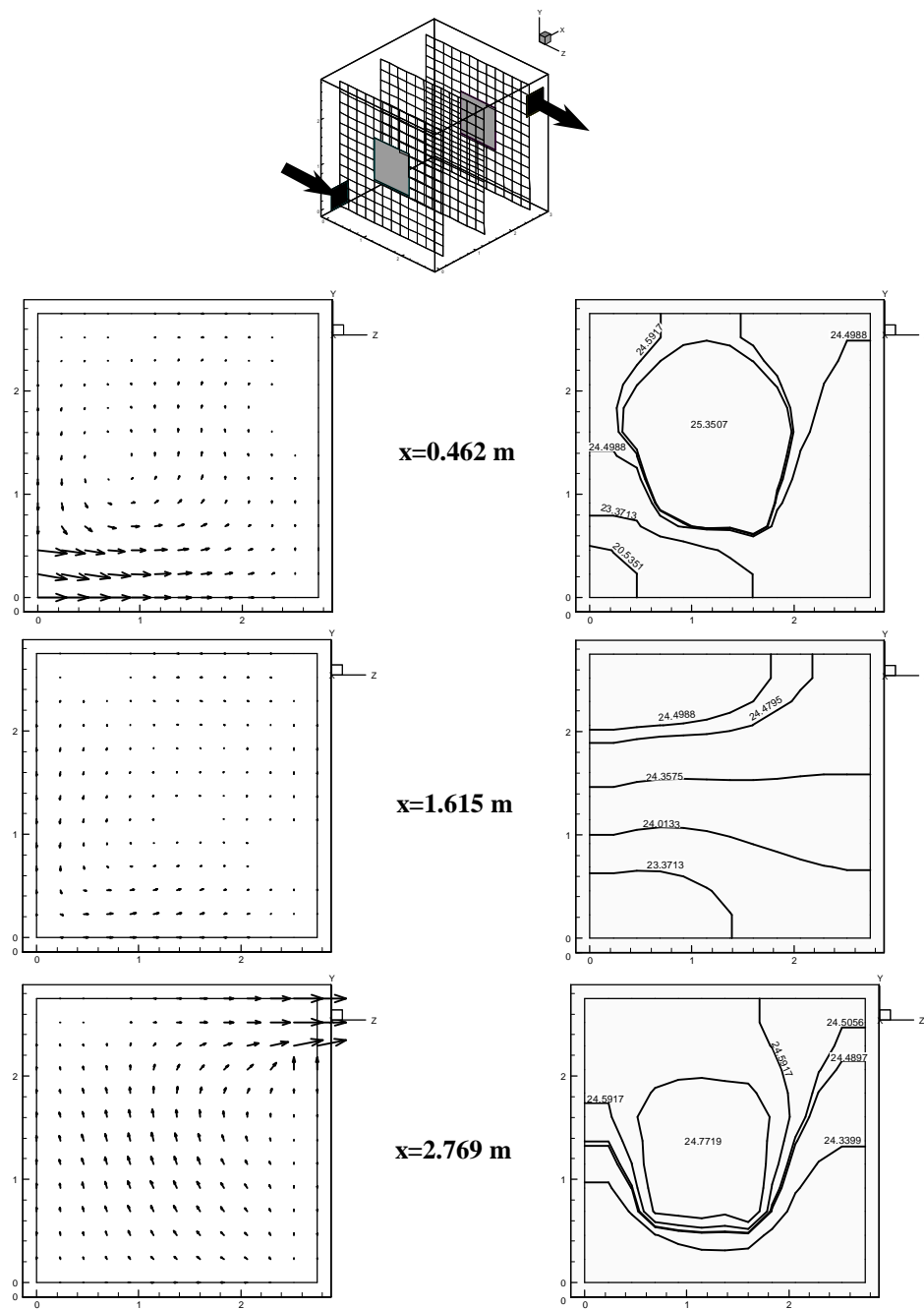


Figure (4) Flow pattern and temperature distribution of (cold air supply) for 8 ACH  
( $x=0.462$ ,  $1.165$  and  $2.769$  m)