A study of the Effect of Air Inlet and Exit on Indoor Air Movement

Abdual Hadi Nema Khalifa
Assistant Professor
Technical College-Baghdad
email:ahaddi58@yahoo.com

Ahmed Jasim Hamad
Lecturer
Technical College-Baghdad
email:ahmed.elhamad@yahoo.com

Johain Joudet Faraj
Lecturer
Technical College-Baghdad
mail:Johaintech@yahoo.com

Ahmed Qasim Ahmed
Assistant Lecturer
Technical College-Baghdad
email:en_ahmed82@yahoo.com

ABSTRACT

This work consists of a numerical simulation to predict the velocity and temperature distributions, and an experimental work to visualize the air flow in a room model. The numerical work is based on non-isothermal, incompressible, three dimensional, $(k - \varepsilon)$ turbulence model, and solved using a computational fluid dynamic (CFD) approach, involving finite volume technique to solve continuity, momentum and energy equations, that governs the room's turbulent flow domain. The experimental study was performed using (1/5) scaled room model of the actual dimensions of the room to simulate room air flow and visualize the flow pattern using smoke generated from burnt herbs and collected in a smoke generator to delivered through the prototype room. The numerical results were compared with those obtained from experiments, the correspondence between numerical and experimental was fairly good and also, a fair comparison was found with other workers in this field.

Key words: visualization, air flow, indoor air, climate, numerical

دراسة تأثير مدخل ومخرج الهواء على حركة الهواء داخل غرفه

أحمد جاسم حمد
مدرس
الكلية التقنية ـ بغداد

عبد الهادي نعمه خليفه
استاذ مساعد
الكلية التقنية ـ بغداد

أحمد قاسم أحمد
مدرس مساعد
الكلية التقنية ـ بغداد

جهين جودت فرج
مدرس
الكلية التقنية ـ بغداد

الخلاصة

يهدف هذا البحث الى تحسين المعرفة ببعض الوسائل التي يمكن ان تستخدم في حساب توزيع السرع ودرجات الحرارة واظهار هيئه جريان الهواء خلال الفضاء المكيف للغرفه وهذه الخطوه لايمكن الاستغناء عنها عند تصميم انظمة تكييف بنوعيه جيدة. تضمنت الدراسه اجراء محاكاة عدديه لحساب توزيعات السرع ودرجات الحراره وعمل تجريبي لاظهار هيئه جريان الهواء في انموذج الغرفه. المحاكاة العدديه للجريان اعتمدت على انموذج الجريان ثلاثي الابعاد المضطرب غير المنضغط وبدرجات حراره مختلفه وباسلوب المعادلتين $\varepsilon - k$ واستخدم في الحل احدى طرائق ديناميك الموائع الحسابي. اجريت الدراسه التجريبيه على انموذج مصغر لغرفه صمم ونفذ خلال هذه الدراسه وفق معامل قياس مناسب 5\1 لمحاكاة جريان الهواء للفضاء الحقيقي (الغرفه) لاظهار هيئه الجريان باستخدام دخان منتج من احتراق حبوب نباتيه يجمع في مولد دخان و يوجه الى انموذج الغرفه. مقارنة النتائج العدديه مع النتائج التجريبيه ومع نتائج باحثين اخرين اظهرت توافق مقبول.

1. INTRODUCTION

The major objective of the heating, ventilation and air-conditioning system is to provide comfort and suitable indoor air quality within the occupied zone of building. An important step in the process is to furnish air to each space, in such a way that any natural air current or irradiative effect within the space are counteracted, and assure that temperature, humidity and air velocities within the occupied zones, are held at acceptable condition. The challenge is to provide good mixing without creating uncomfortable drafts, and assure that there is reasonable uniformity, without unacceptable changes in the room conditions as the load requirements of the room change. **Awbi, 1989.** studied the results of computer programs developed for solving 2-D and 3-D ventilation problems, the problems were solved by finite difference form, steady state conservation equations of mass, momentum, and thermal energy. Preservation of fluctuating velocity components is made using the $(k - \varepsilon)$ turbulence model. Predicted result of air velocity and temperature distribution in the room, were corrected by experimental measurements, a CFD method has been described and applied to predict the air flow and heat transfer in 2-D enclosure and the 3-D flow of wall jet over surface with mounted obstacles. The CFD solution produced reasonably good predictions of the air velocity and temperature distribution in the test room. **Bartak , et el.,2001.** studied the room with mixing ventilation, focused on the local mean age of air. The measurements were performed, using the tracer gas concentration decay method. The numerical predications were obtained from CFD model. Two numerical grids, coarse and fine, have been used to compare the accuracy of the results. **Hanibuchi, and Hokoi,2001.** studied the distribution properties of room air temperature, velocity and heat loss, which were investigated experimentally and analytically. Measurements under heating condition were made in a full-scale model room, in order to investigate the influence of air conditioner location on the velocity and temperature fields in steady state, as well as unsteady heating load. The following results were obtained, the influence of the direction in which the jet was blown from the air conditioner is clearly seen in regions of different temperature distribution adjacent to the floor; the heat loss calculated from those experimental results showed several difference characterized by the direction of the blown jet, and the difference between the maximum and minimum heat loss obtained from the experimental results is roughly 15%, and, results from numerical analysis agree with the experimental results. **Zhang,** and **Chen,2005**. studied the particle transport and other distributions in ventilated rooms, by both experimental investigation and numerical simulation. The numerical simulation was achieved by $(k - \varepsilon)$ turbulence model, the Lagrangian particle tracked method, and the particle source in cell scheme. The Lagrangian method was introduced, evaluated and then used to analyze particle dispersion in rooms, with three different ventilation systems. The Lagrangian method introduced uncertainty in the particle concentration calculation. The uncertainly level is associated with Lagranging sample size, number of computational cells for concentration, and even particle source position in the turbulent flow. **Nielsen, 2006.** conducted the experiments with room air distribution that is generated by a radial ceiling-mounted diffuser and a diffuser generating flow with swirl then compared with the air distribution obtained by mixing ventilation from a wall-mounted diffuser, vertical ventilation, and displacement ventilation. The air distribution generated by a radial diffuser is partly controlled by the momentum flow from the diffusers and partly from gravity forces where the thermal load and the temperature difference between room air and supply air deflect the radial wall jet down into the occupied zone. The ceiling diffuser with swirling flow generates a flow pattern in the room that is rather uninfluenced by the thermal load. The flow is highly mixed above the occupied zone, and the air movement penetrates the occupied zone close to the walls. All systems were tested in the same room with a load consisting of two manikins, each sitting at a desk with two PCs and two desk lamps, producing a total heat load of 480 W. In all five cases, the design of the air distribution system was based on flow elements from the diffuser, a maximum velocity assumption, and a critical vertical temperature gradient in the room. **Chen,** and **Zhang,2008.**

introduced two categories of flow models, Reynolds Average Navier stoke equation model (RANS modeling), and Large Eddy Simulation (LES), as well as two popular particle models (Lagrangian and Eulerian). Both RANS and LES modeling have been used to predict air flow in rooms. The performance of RANS modeling was similar to that of LES for the case studied. However, LES provide more detailed information than RANS modeling and used only one or no empirical coefficient. LES is considered as a powerful tool for multiple scale studies. **Kwang, et al., 2008**. studied the effect of the airflow discharge rate of the ceiling type air-conditioner on ventilation performance in a lecture room with mixing ventilation. Experiments and CFD were conducted for analyzing ventilation performance. The concepts of mean air age and indoor CO_2 concentration were used for evaluating ventilation performance. A CO_2 generation model was used in the simulation and calculation a lot of cases with respect to the airflow rate of air-conditioner and the mechanical ventilation rate. A selected experimental measurements were performed in the lecture room of the same layout as the numerical one for verifying simulation results. Mean air age is gradually increased, but CO_2 concentration is oppositely decreased in the occupied zone with the increment of the discharge airflow rate of the ceiling type air-conditioner. This result shows that both mean air age and residual life time must be considered for evaluating ventilation performance when the contaminants are generated indoors. And the increment of discharge airflow of the ceiling type air-conditioner can induce the piston effect and push the contaminants out of the occupied zone. From this result, it is found out that ventilation performance can be increased when the momentum source like an air-conditioner is used in the room with the mixing ventilation.

2. ROOM AIR MODELING

The successful prediction of room air flow or air movements in enclosure has recently been based on two approaches. The first approach depends on the measurements of air velocity, and temperature distribution in a physical model of building in a laboratory, for pre-design evaluation or actual site measurements for post design investigation. The second approach depends on numerical predication by considering the room air flow simulation model solved using computational fluid dynamic (CFD) techniques. The two approaches mentioned above were used in this work.

2.1 Numerical Solution

The equations that describe the flow of a fluid and heat within an enclosure are all based on **Awbi, 1989**, and assume that the numerical simulation is based on non-isothermal, incompressible three- dimensional ($k - \varepsilon$) two - equations turbulent flow model:

2.1.1 Conservation of mass

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0 \qquad (1)$$

2.1.2 Conservation of momentum
For x direction, U momentum is:

$$\frac{\partial}{\partial t}(\rho u) + \frac{\partial}{\partial x}(\rho uu) + \frac{\partial}{\partial y}(\rho uv) + \frac{\partial}{\partial z}(\rho uw) = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(\mu \frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu \frac{\partial u}{\partial y}\right) + \frac{\partial}{\partial z}\left(\mu \frac{\partial u}{\partial z}\right) +$$

$$\frac{1}{3}\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)\right] + \frac{\partial}{\partial x}(-\rho \overline{u'u'}) + \frac{\partial}{\partial x}(-\rho \overline{u'v'}) + \frac{\partial}{\partial x}(-\rho \overline{u'w'}) + \rho g_x \qquad (2)$$

Similar equations are used for momentum in y and z directions.

2.1.3 Conservation of energy

$$\frac{\partial}{\partial t}(\rho T)+\frac{\partial}{\partial x}(\rho u T)+\frac{\partial}{\partial y}(\rho v T)+\frac{\partial}{\partial z}(\rho w T)=\frac{\partial}{\partial x}\left(\Gamma\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\Gamma\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\Gamma\frac{\partial T}{\partial z}\right)$$

$$+\frac{\partial}{\partial x}\left(-\rho\overline{u'T'}\right)+\frac{\partial}{\partial x}\left(-\rho\overline{v'T'}\right)+\frac{\partial}{\partial x}\left(-\rho\overline{w'T'}\right)+S_T \quad (3)$$

Where: $\Gamma=\dfrac{\mu}{\sigma}, \ and \quad \sigma=\dfrac{\mu.cp}{\lambda} \quad and \ T=T+T'$

The term S_T is a source term allowing for the rate of thermal energy production.

2.1.4 $k-\varepsilon$ Turbulence model

Two equations $k-\varepsilon$ turbulent model is considered to simulate the turbulent flow numerically. The transport equation for turbulence kinetic energy (k) is given by [**Awbi, 1989**.]:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x}(\rho u k)+\frac{\partial}{\partial y}(\rho v k)+\frac{\partial}{\partial z}(\rho w k)=\frac{\partial}{\partial x}\left(\Gamma_k\frac{\partial k}{\partial x}\right)+\frac{\partial}{\partial y}\left(\Gamma_k\frac{\partial k}{\partial y}\right)+\frac{\partial}{\partial z}\left(\Gamma_k\frac{\partial k}{\partial z}\right)+$$

$$\mu_t\{2[(\frac{\partial u}{\partial x})^2+(\frac{\partial v}{\partial y})^2+(\frac{\partial w}{\partial z})^2]+(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x})^2+(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x})^2+(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z})^2\}$$

$$-c_u\rho\frac{k^{1.5}}{L}+\beta g\frac{\mu_t}{\sigma_t}\frac{\partial T}{\partial y} \quad (4)$$

Where:

$$k=\frac{1}{2}\left[\left(\overline{u'}\right)^2+\left(\overline{v'}\right)^2+\left(\overline{w'}\right)^2\right] \quad (5)$$

is the turbulent Prandtl number (0.5 to 0.9) **Awbi,1989**, and c_u is constant $\Gamma_k=\dfrac{\mu_e}{\sigma_k}$, $\sigma_k\approx1$, σ_t

≈ 0.09, the last term of equation 4 represents the effect of buoyancy.

The body force in x and z direction $\rho g_x \ and \ \rho g_z$ were omitted.

$$\frac{\partial}{\partial t}(\rho u)+\frac{\partial}{\partial x}(\rho u u)+\frac{\partial}{\partial y}(\rho u v)+\frac{\partial}{\partial z}(\rho u w)=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial u}{\partial z}\right)+$$

$$\frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial w}{\partial x}\right) \quad (6)$$

The body force in the y direction has been written as a buoyancy force, where is the density at the flow reference temperature.

$$\frac{\partial}{\partial t}(\rho v)+\frac{\partial}{\partial x}(\rho u v)+\frac{\partial}{\partial y}(\rho v v)+\frac{\partial}{\partial z}(\rho v w)=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(\mu_e\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial v}{\partial z}\right)+$$

$$\frac{\partial}{\partial x}\left(\mu_e\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial y}\left(\mu_e\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left(\mu_e\frac{\partial w}{\partial y}\right)-g(\rho-\rho_o) \quad (7)$$

The transport equation for ε is given as:

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x}(\rho u\varepsilon)+\frac{\partial}{\partial y}(\rho v\varepsilon)+\frac{\partial}{\partial z}(\rho w\varepsilon)=\frac{\partial}{\partial x}\left(\Gamma_\varepsilon\frac{\partial\varepsilon}{\partial x}\right)+\frac{\partial}{\partial y}\left(\Gamma_\varepsilon\frac{\partial\varepsilon k}{\partial y}\right)+\frac{\partial}{\partial z}\left(\Gamma_\varepsilon\frac{\partial\varepsilon k}{\partial z}\right)+$$

$$C_1\frac{\varepsilon}{k}\mu_t\{2[(\frac{\partial u}{\partial x})^2+(\frac{\partial v}{\partial y})^2+(\frac{\partial w}{\partial z})^2]+(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x})^2+(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x})^2+(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z})^2\}$$

$$-c_2\rho\frac{\varepsilon^2}{k}+C_1\beta g\frac{\varepsilon}{k}\Gamma_t\frac{\partial T}{\partial y}\qquad(8)$$

Where:

$$\Gamma_k=\frac{\mu_e}{\sigma_\varepsilon},\sigma_k \text{ :is a constant equal to 1.22, } C_1=1.44, C_2=1.92.$$

2.1.5 Boundary conditions
2.1.5.1 Inlet boundary conditions
The flow is assumed uniform at room inlet as shown in **Fig.1**. The tangential velocity is set to be zero, the normal velocity component (U_n) is determined depending on the desired air change per hour (ACH) as follows **Kwang, 2008**:

$$U_{jet}=\frac{ACH.V_r}{3600.A_{in}}\qquad(9)$$

Where:

V_r =Room volume

A_{in} =Supply outlet area
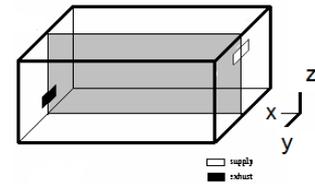
Figure1.

The kinetic energy and dissipation rate at inlet are determined using the following equations **Awbi, 1989**:

$$k_{in}=\frac{2}{3}I_u^2.U_{jet}^2\qquad(10)$$

$$\varepsilon_{in}=\frac{k_{in}^{\frac{2}{3}}}{\lambda_H}\qquad(11)$$

Where $\lambda=0.005$ and $I_u^2=0.14$

2.1.5.2 Outlet boundary conditions
A uniform velocity distribution is assumed over the room outlet. Tangential velocity is considered to equal zero ($U_i=0$), and the normal velocity is computed from the continuity equation.

$$U_{out}=U_{in}=\frac{(\rho.A)_{in}}{(\rho.A)_{out}}\qquad(12)$$

Exit temperature T$_{out}$ is obtained from the energy equation for whole flow field. Due to the assumption of a uniform distribution of the forces in the exit plane, the gradient of k, ε and all scalar properties are zero **Kwang,2008**, hence

$$\frac{\partial T}{\partial x_n} = 0, \quad \frac{\partial k}{\partial x_n} = 0, \quad \frac{\partial \varepsilon}{\partial x_n} = 0, \qquad (13)$$

2.1.5.3 Solid boundary conditions

The normal velocity U$_n$ is considered equal to be zero at the wall. The simplest way of imposing the tangential velocity U$_i$ is

$$\left.\frac{\partial U_i}{\partial x_n}\right|_{wall} = \frac{mU_i}{y_n} \qquad (14)$$

Where m is a constant equals to $\frac{1}{7}$.

Boundary conditions for turbulent energy may be imposed using the following equation **Nielsen, 2006**:

$$\varepsilon = \frac{(c_u^{\frac{1}{2}} \cdot k)^{\frac{3}{2}}}{k\,y_n} \qquad (15)$$

Where c_u is constant and equal to 0.09.

To impose the temperature B.C. on the wall, a heat balance concept is adopted, by assuming the heat conduction through the wall is equal to the heat convected a way from the wall.

$$k\frac{\partial T}{\partial x_n} = hc(T_2 - T_w) \qquad (16)$$

2.1.6 Initial conditions

The initial values for all three velocities and pressure at every node inside the computational flow domain are set to zero, while, the initial value of temperature is assumed some suitable value depending of each case, and the initial values for the turbulent kinetic energy and turbulent dissipation rate are set at some percentage of the inlet values.

2.2 Numerical Procedure

    A finite volume method has been used in discretization scheme with staggered grids and Semi Implicit Method for Pressure Linked Equation (SIMPLE) procedure to link the velocity filed to pressure filed **Shuzo, 1989** . The computational domain is divided into number of non-overlapping control volumes.

3. EXPERIMENTAL WORK

    The experimental test rig consists of a typical prototype room (5m × 4m × 3m), and a suitably scaled (1/5) model room (1m × 0.8m × 0.6m). The minimum air changes per hour (ACH), for a room with side wall supply air, was calculated using the relation given by **Kwang,2008**:

$$N = 7.843 \sqrt[3]{\frac{qB}{(3+H) \times L^2}} \qquad (17)$$

Where :

B,H,L= room dimension in m.

q: total heat load in W.

To produce N=6.913 ACH for the prototype room. Accordingly, other prototype related

parameters were calculated. The kinematical similarity, between the prototype and model rooms, was achieved by equating the Reynolds number, resulting in:

$U_m = 5 \ U_p = 4.5$ m/s

Similarly, other model related parameters were calculated.

The reduced scale model was made from an aluminum frame with 4mm thick sliding plastic walls and ceiling. The ducting system were made from galvanized steel plates (gauge 24), with different dimensions. Two centrifugal fans (2825 rpm) were used; one to deliver the smoked air to the room and the other is for the exhaust. The supply and return air velocities were measured using digital anemometers (Model DA40). The smoke was traced out using Sony digital camera. **Fig.3** shows the complete test rig with the model room mounted.

4. RESULT AND DISCUSSIONS

The flow inside a room is affected by many parameters, mechanical and thermal. The amount of air flow, inlet temperature, supply velocity, location of opening, room geometry and the presence of obstacles are all important parameters. To add to the complexity, the buoyancy has many sources like computers, human, lights to name a few. Almost, all of the above parameters were considered in this work. **Fig.4** pictures the air distribution through the prototyped room, the swirl is clearly visible. The numerical results come close to the experimental outcome as shown in **Fig.5** for the same case of one supply opening and one outlet opening. The swirl persists even near side walls as shown in **Fig.6**.The swirl becomes more pronounced with the two supply openings and one outlet opening all located on the same wall as shown in **Fig.7**. While **Fig.8** shows that the pattern becomes more complicated for the three supply opening on the front wall and one outlet opening on the side wall. The flow pattern generated by air diffusers is simpler as can be seen from f**igures 9** and **10** for one and two diffusers respectively. When the ceiling contains an obstacle, stagnant zones will be generated behind them, also separation and reattachments of the air stream and the accompanying recirculation can be recognized. These flow patterns vary in strength depending on the proximity of these obstacles from the supply openings, as shown in **figures 11, 12** and **13** for (1/4), (1/2) and (3/4) room length proximity respectively. **Table 1** gives the summary of the variables under study. **Table 2** shows the properties of air and other numerical parameters used in simulation model.

5. VALIDITY OF CURRENT WORK

The results of the air velocity and temperature distribution that's obtained by using numerical solution method inside the room are validated with that obtained from experimental work. The comparison between the numerical and experimental work showed a good agreement, as shown in **Fig.2**. Also the numerical and experimental results obtained in current work are validated by

comparing with similar experimental and numerical work **Shuzo, 1989** and the comparison also showed a good agreement, as shown in **Fig.2**.

6. CONCLUSIONS

When inlets discharge horizontally near the ceiling, the warmest air in the room mixed immediately with cool air. Ceiling diffusers that designed for vertical downward air projection produce stagnant regions near the ceiling. When the supply jet encounters ceiling obstacles, it either completely separates from the ceiling or it may reattach downstream the obstacle.

Table1. Summary of variables that are studied in the present work.

Parameter	variation
Supply velocity	0.6 , 0.9 , 1.5 (m/s)
Flow type	Turbulent with all function
Room dimensions	$4 \times 5 \times 3$ (m)
Inlet air velocity profile	Uniform for all cases
Grid resolution	Uniform $40 \times 50 \times 30$

Table2. Fluid properties in simulation model.

Parameter	ρ kg/m^3	v m^2/s	k W/mK	C_p J/kgK	T_{in} $°C$	T_w $°C$
Value	1.19	1.15×10^{-5}	0.026	1003.6	17, 25, 35	30
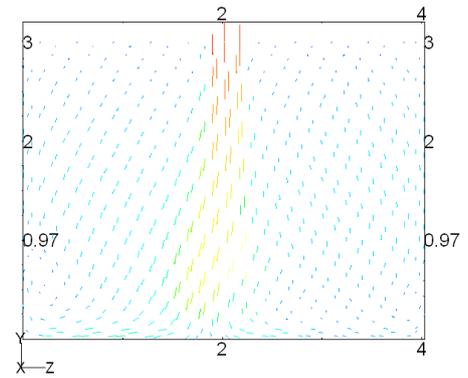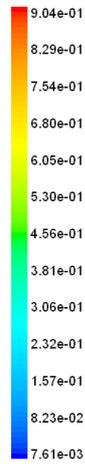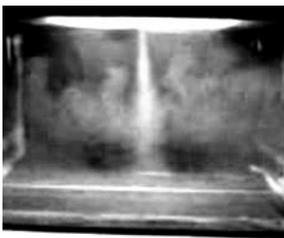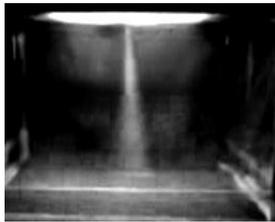
Visualization current work

Numerical current work
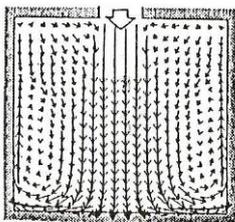
Numerical simulation ,**Shuzo, 1989**.
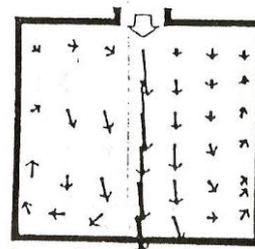
Experimental work, **Shuzo, 1989.**

Figure 2. Validity of current work**.**

117
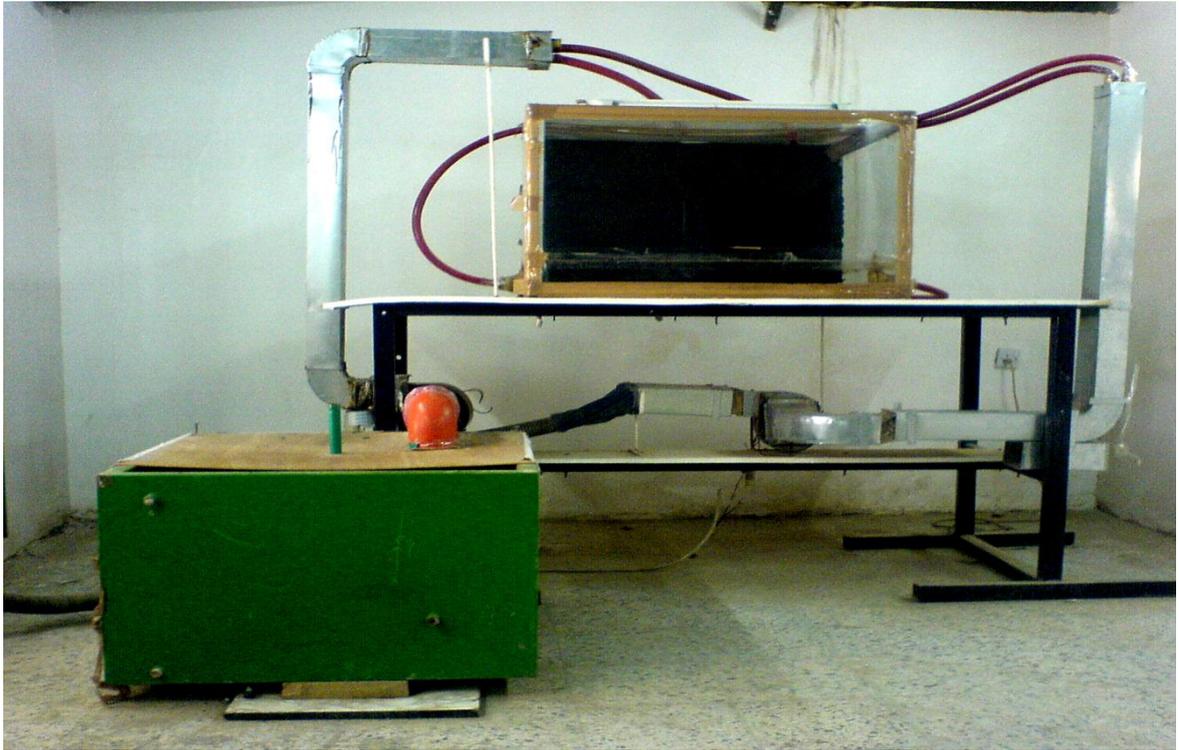
Figure3. Complete test rig with the model room mounted.

Figure4. Air distribution through the prototyped room.
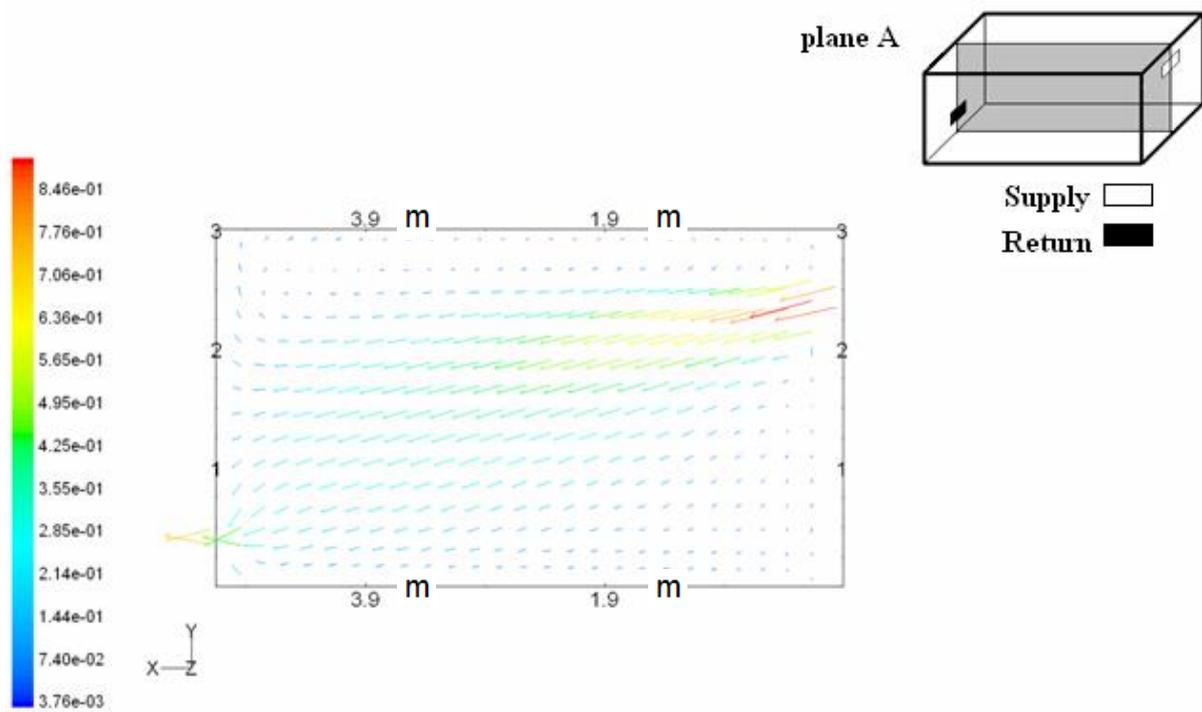
Figure 5. Velocity distribution in a room with single supply and single extract opening.
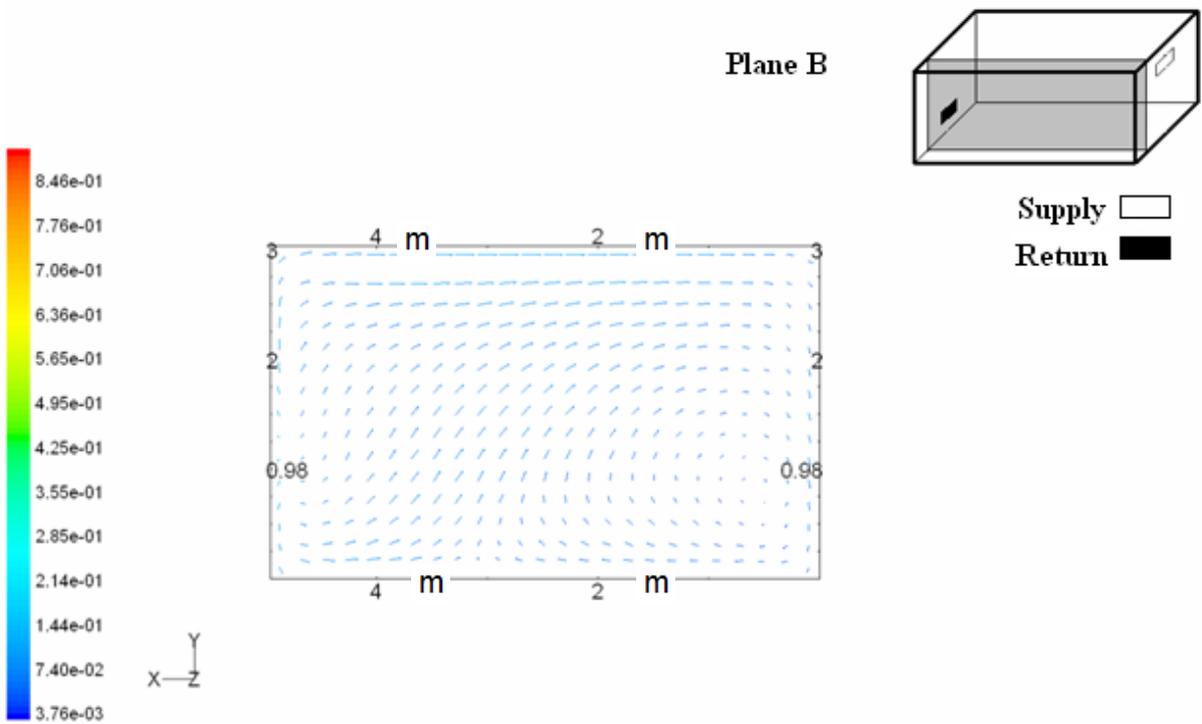
Figure 6. Velocity distribution in a room with single supply and single extract opening.
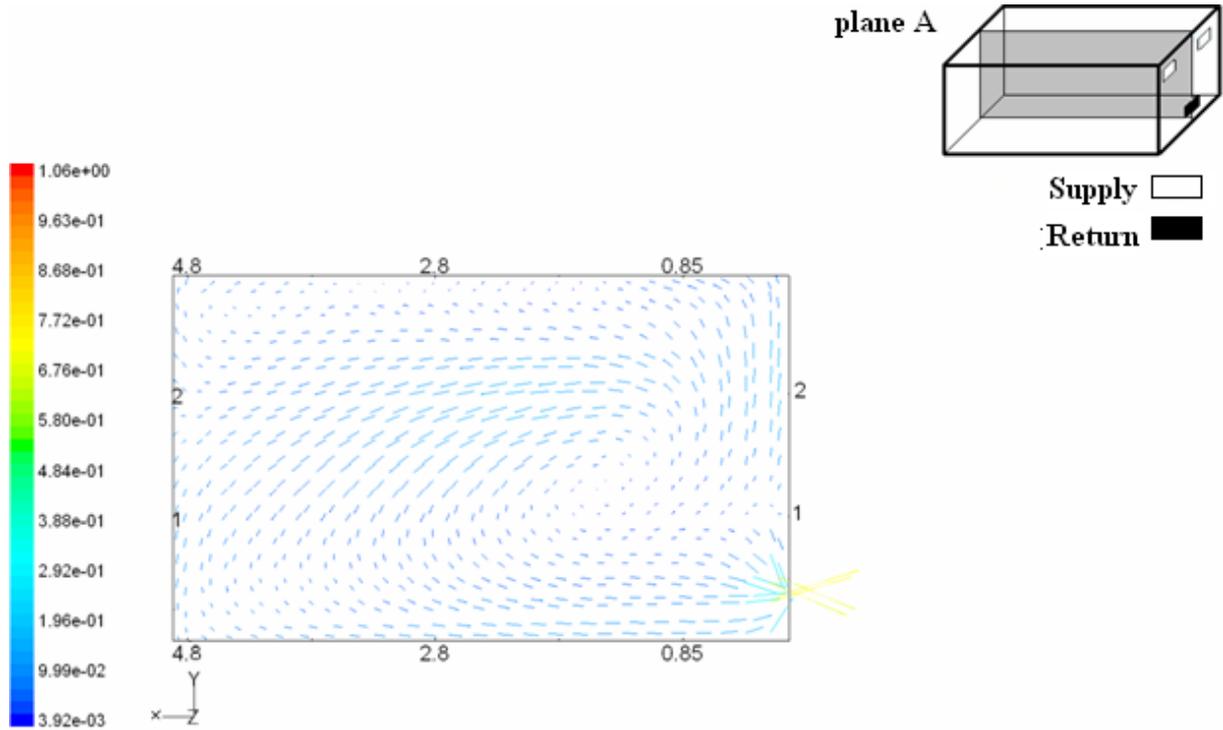
Figure7. Velocity (m/sec) distribution in a room with two supply and single extract opening.
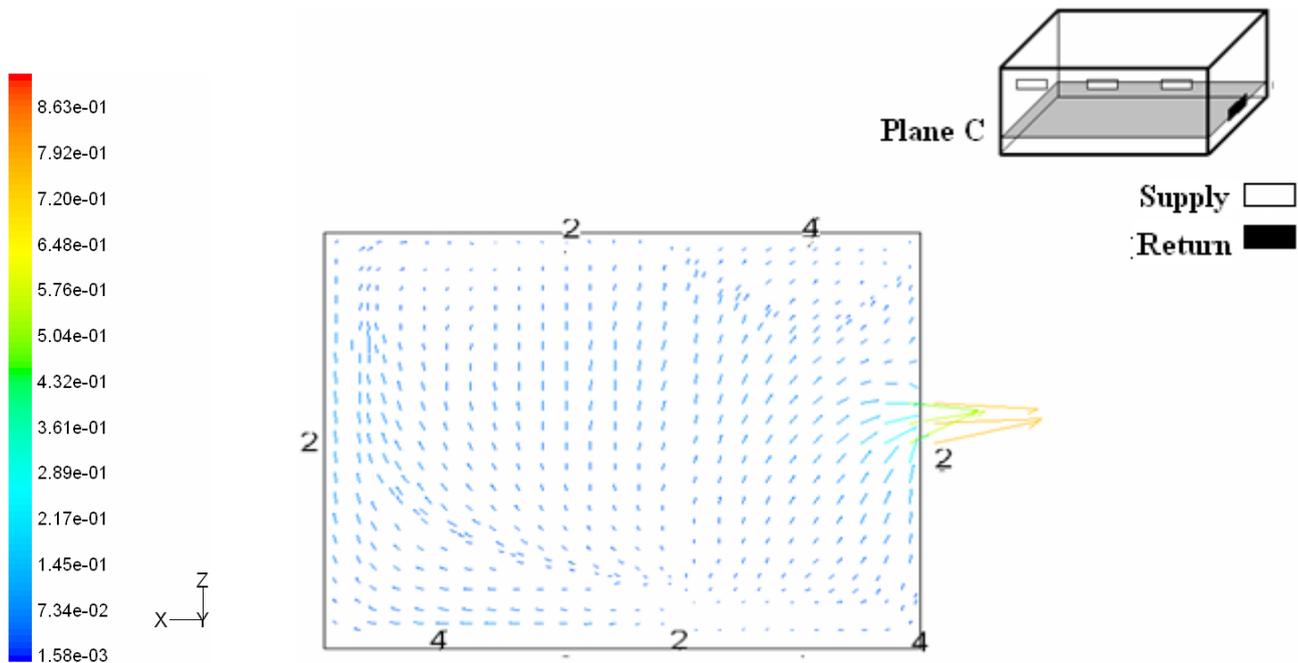
Figure 8. Velocity (m/sec) distribution in a room with three supply and single extract opening.
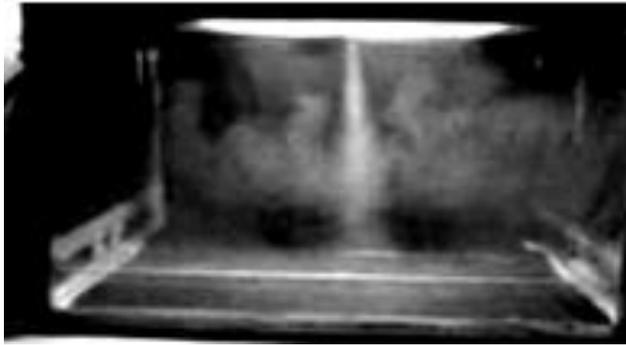
Figure 9. Pattern generated by single diffuser.   **Figure10.** Pattern generated by two diffusers.
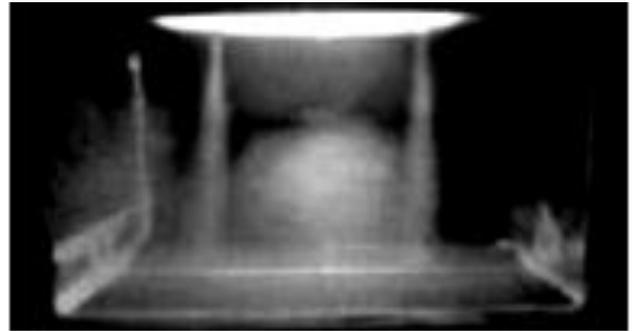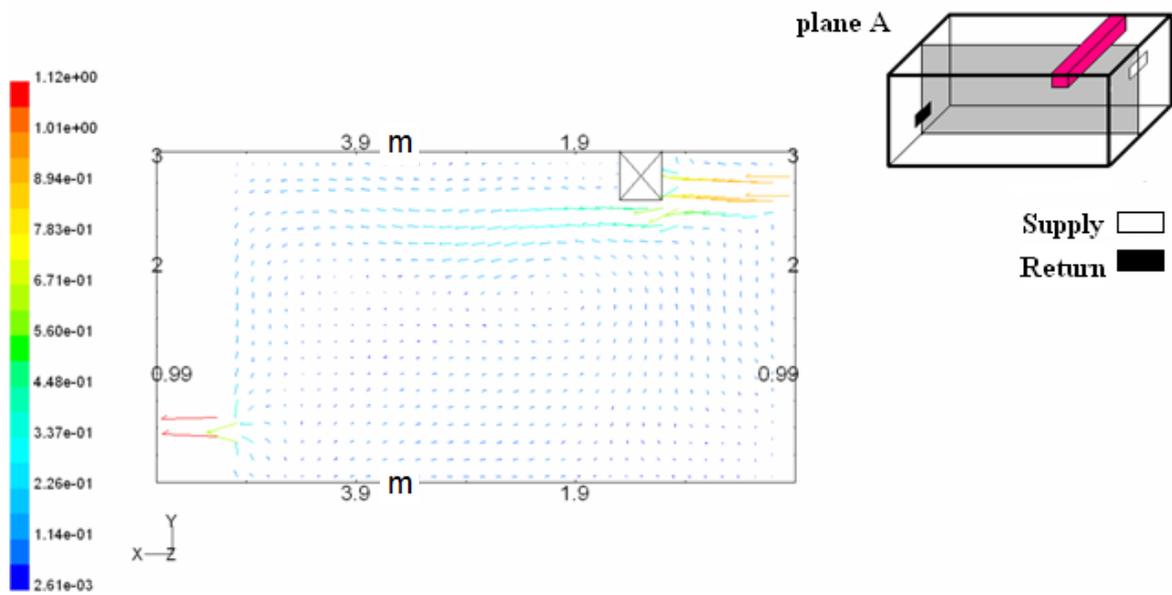
Figure11. Velocity distribution with velocity (U supply= 0.9 m/s) and constant supply temperature of (T=17 $^{\circ}C$) with ceiling obstacle at (1/4) of the room length.
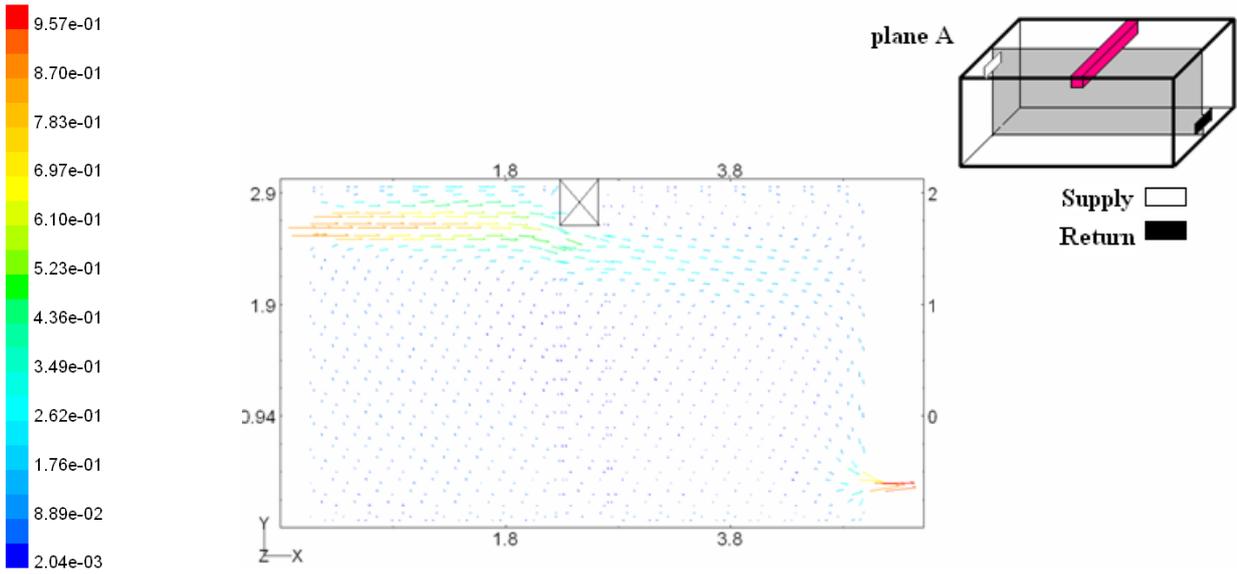
Figure 12. Velocity distribution with velocity (U supply=0.9 m/s) and constant supply temperature of (T=17$^\circ C$) with ceiling obstacle at (1/2) of the room length.
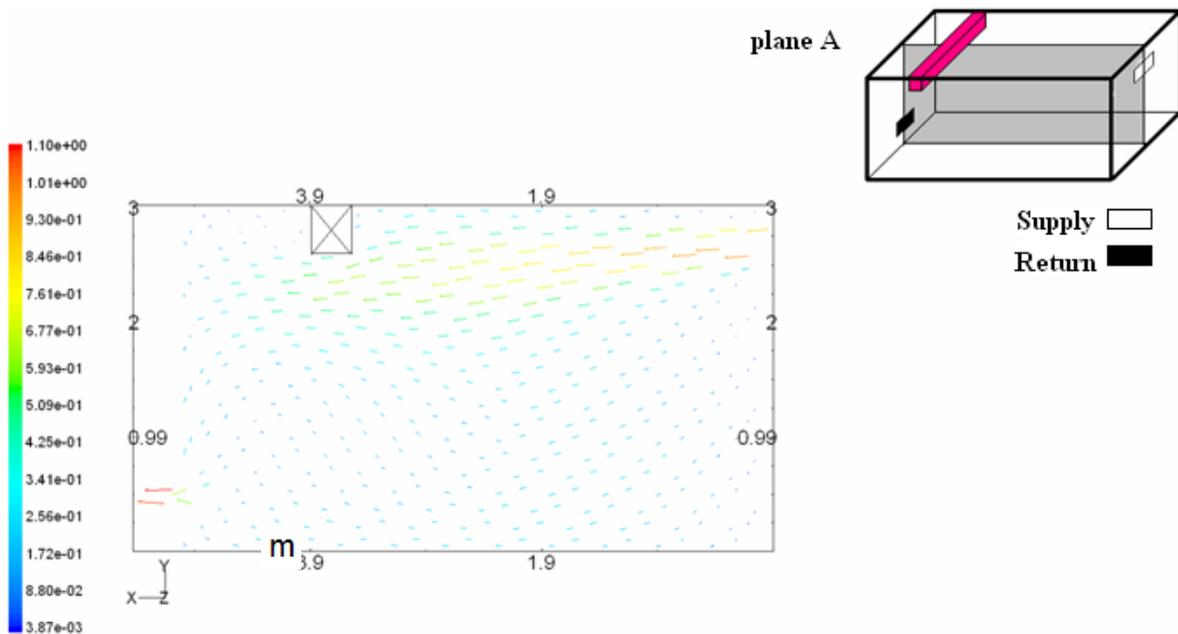
Figure 13. Velocity distribution with velocity (U supply=0.9 m/s) and constant supply temperature of (T=17$^\circ C$) with ceiling obstacle at (3/4) of the room length.

122

REFERENCES

Awbi, H.B., 1989, *Application of Computational Fluid Dynamics in Room Ventilation,* Building and environment, Vol.24, No.1, PP.73-84.

Bartak, M., Cermak, M., Clarke, J. A., Denev, J., and Drkal, F., 2001, *Experimental and Numerical Study of Local Mean Age*, Seventh International IBPSA Conference, Rio de Janeiro, Brazil, PP. 773-780.

Chen, Q., and Zhang, Z., 2005, *Prediction of Particle Transport in Enclosed Environment*, China Particuology, Vol.3, No.6, PP. 364-372.

Hanibuchi, H., and Hokoi, S., 2001, *Experimental and Numerical Prediction of the Velocity and Temperature Field in a Room Heated by a Room Air-conditioner* , Ph.D. thesis- Kyoto University, Japan, PP. 606.

Kwang, C. N., Chang, W.H., and Myung, D. O., 2008, *Effect of the Airflow Rate of a Ceiling Type Air-Conditioner on Ventilation Effectiveness in a Lecture Room*, International Journal of Refrigeration Vol. 31, Issue 2, PP. 180-188.

Nielsen, P., 2006, *Air Distribution In A Room With Ceiling-Mounted Diffusers : Comparison With Wall-Mounted Diffuser, Vertical Ventilation, and Displacement Ventilation*, American Society of Heating, Refrigerating and Air-Conditioning Engineers. Annual meeting, Chicago, Vol. 112, No.2, PP. 498-504.

Shuzo, M., and Shinsuke, K., 1989,*Numerical and Experimental Study on Room Air Flow 3-D Predictions using the (K-E) Model*, Building and environment.Vol.24, No.1, PP 85-97.

Zhang, Z., and Chen, Q., 2006, *Experimental Measurements and Numerical Simulations of Particle Transport and Distribution in Ventilated Rooms*, Atmospheric Environment, Vol.40, No.18, PP. 3396-3408.

NOMENCLATURE

Symbols	Definition	Unit
a	Coefficient value	dimensionless
A_{in}, A_{out}	Inlet and outlet cross- section area	m^2
g_x, g_y, g_z	Gravitational acceleration	m/s^2
H	Room height	m
J	Flux due to both diffusion and convection	dimensionless
k	Thermal conductivity	$W/m.K$
k	Turbulent kinetic energy	m^2/s^2
k_{in}	Turbulent energy at room inlet	m^2/s^2
$k_{i,j,k}$	Turbulent energy at cell	m^2/s^2
p	Pressure	N/m^2
Pe	Peclet number	dimensionless
Re	Reynolds number	dimensionless
$T_\circ$	Reference temperature	K
T	Temperature	K
T'	Fluctuation temperature	K
$T_{i,j,k}$	Temperature at cell center	K
U, V, W	Velocity components in x ,y and z-direction	m/s
U_{jet}	Inlet velocity	m/s
U_n	Normal velocity	m/s
μ	Dynamic viscosity	$kg/m.s$
α	Diffusivity	m^2/s
β	Volumetric expansion coefficient	$1/K$

ε	Turbulent energy dissipation	m^2/s^3
ϕ	Generic scalar quantity	dimensionless
ρ	Density	kg/m^3
μ_e	Effective viscosity coefficient	$kg/m.s$
μ_t	Turbulent viscosity	$kg/m.s$
υ	Kinematics viscosity	m^2/s
υ_e	Effective kinematics viscosity	m^2/s
υ_t	Turbulent viscosity	m^2/s
$\sigma_k, \sigma_\varepsilon$	Constants for $k-\varepsilon$ model	dimensionless
$\Delta x, \Delta y, \Delta z$	Cell dimensions	m
Γ	Diffusion coefficient	dimensionless
Γ_t	Turbulent diffusion coefficient	dimensionless
μ_e, μ_t	Coefficients of effective viscosity and turbulent viscosity	$kg/m.s$