AIR TEMPERATURE DISTRBUSION AND CONTAMMINATE CONCENTRATION IN LABORATORY UNDER IRAQI CLIMATE

Dr. Alaa Abbas Mahdi   Zahraa Hassan Jasim
alaa.abbas59@yahoo.com
University Of Babylon , Colleg Of Engineering / Mechanical Engineering Department

ABSTRACT :-

Ventilation is the main performance requirement in laboratory design as it has to guarantee a safe and comfortable indoor environment. standards and guidelines on laboratory ventilation often impose high ventilation rates, increasing the energy need for ventilation. This research focuses on the steady-state distribution of temperature and contaminate concentrations in a real environment which was a function of several factors such as the position of air exhausts. Here several different exhaust locations were investigated to determine the optimum exhaust positions. In this study, CO_2 is used as an indicator of the concentration of pollutants inside tested room.

Room concentration patterns for a laboratory were simulated with computational fluid dynamics (CFD) simulations by using (FLUINT6.3.26) and (GAMBIT V.2.4.6) computer programs for various exhaust locations. The computational results were validated with design data due to Iraqi cooling code (setup temperature and temperature difference from head to foot level). The numerical results showed that the exhaust grills located near the ceiling resulted in lower pollutant concentrations than the corresponding exhausts near the floor.

KEYWORDS: Mixing ventilation, Indoor air quality, air distribution performance index, Numerical model, Iraqi cooling code.

تأثير مواقع السحب على توزيع درجات الحرارة وتركيز الملوثات داخل حيز مختبر عند ظروف المناخ العراقية

د.علاء عباس مهدي   زهراء حسن جاسم
جامعة بابل/ كلية الهندسة/ قسم الهندسة الميكانيكية

الخلاصة :-

تعتبر التهوية احد المتطلبات الرئيسية عند تكييف الهواء في تصميم المختبرات لضمان بيئة داخلية أمنة ومريحة. غالبا ما تفترض المعايير والتوجيهات الحالية وجوب تجهيز معدلات تهوية عالية في المختبر مما يؤدي الى زيادة الطاقة المستخدمة للتهوية. في هذا البحث تم التركيز على توزيع درجات الحرارة والملوثات في الحالة المستقرة داخل بيئة حقيقية معتمدة على عدة عوامل ومنها موقع منافذ سحب الهواء. تم دراسة مواقع مختلفة لمنافذ سحب الهواء لتحديد موقع السحب المثالي . كذلك استخدم غاز ثاني اوكسيد الكربون كمؤشر لتركيز الملوثات داخل حيز الاختبار .

استخدم برامج الحاسوب (FLUINT6.3.26) و (GAMBIT2.4.6) لمحاكاة وتحليل انماط توزيع الملوث لمختلف مواقع السحب .تم التحقق من صحة النتائج من خلال مقارنتها مع القيم التصميمية الموجودة في المدونة العراقية للتبريد (الحرارة التصميمية، فرق درجات الحرارة بين مستوى الراس والقدم).اظهرت النتائج العددية ان مواقع السحب بالقرب من السقف تعطي توزيع ملوثات اقل تركيزا مما لو كانت بالقرب من مستوى أرضية الحيز.

Received:28-1-2016
Accepted:13-4-2016

Nomenclature

Sym.	Description	Sym.	Description
A	Surface area for wall. (m^2)	LM	Corrector latitude and month
C_p	Specific heat of the air at constant pressure. (KJ/Kg.K)	N	Total number of draft temperature points measured in occupied zone
DR	Daily Rang for outlet temperature. (^oC)	Q	Heat transfer through the wall. (W)
q_{ex}	Cooling load for the heat conduction through the walls and transmitted solar radiation. (W)	T_x	local temperature. (^oC)
q_l	Cooling load for the overhead lighting. (W)	ΔT_{hf}	Temperature difference from head to foot level. (^oC)
q_{oe}	Cooling load for occupants, desk lamps and equipment. (W)	T_{av}	Average room temperature. (◦C)
Q_r	Radiation heat transfer. (W)	T_r	Total mean temperature. (^oC)
h_o	Convection heat transfer coefficient for outside air. $W/m^2.k$	T_e	Exhaust air temperature (^oC)
h_i	Convection heat transfer coefficient for inside air. ($W/m^2.k$)	T_m	outlet design temperature. (^oC)
K	Color factor correct (Dark = 1.0, Med = 0.83, Light = 0.65)	T_o	outlet temperature. (^oC)
K_1	Conduction heat transfer coefficient for first layer of wall. (W/m.k)	T_{sp}	Setup (design)temperature. (^oC)
K_n	Conduction heat transfer coefficient for final layer of wall. (W/m.k)	U	Total heat transfer coefficient $W/m^2.K$

Abbreviations

ADPI	**Air Distribution Performance Index**
ASHRAE	American Society for Heating, Refrigeration, and Air Conditioning Engineering
CFD	Computational Fluid Dynamics
CLF	Cooling load factor.
CLTD	Cooling load temperature difference, depend on type of wall.
CLTDc	Correct cooling load temperature difference
MV	Mixing Ventilation
EDT	Effective Draft Temperature
SC	Shadow coefficient.
SHG	Solar heat gain.

Sub-Script

Av	**average**	**o**	**outside**
Ex	External	oe	Occupants and equipment
Hf	head to foot level.	r	Radiation
I	Inside	sp	Setup
L	Overhead light	x	local

Greek letter

ρ	Air density	kg/m^3

INTRODUCTION

The past decade, interest in energy efficient building design has increased rapidly due to climate change issues. The raise energy use in research laboratories gives rise to a large potential for energy saving strategies for these facilities. In search of energy efficient laboratories, ventilation plays a main role as it is one of the largest factors in the total energy use of laboratory facilities. Ventilation of laboratory is responsible for 25 % of the total primary energy use of the university, (Glenn Reynders and Dirk Saelens, 2011).

Ventilation is the method of providing fresh air to an enclosed space to refresh / remove / replace the existing atmosphere. Ventilation is usually used to remove contaminates such as gasses, dust occurring as particles or vapors and provide a healthy and safe working environment. It can be either natural or mechanical ventilation. It is not just adequate to remove heat and contaminated air, it is also necessary to distribute and control the air movement in the tested room in such a way that thermal comfort is achieved in the occupied zone, (Gery Einberg, 2005). This paper presents a numerical study of a computer laboratory in Babylon university for predicting the indoor airflow and thermal comfort by adopting a mixing ventilation system with different locations of exhaust grills as shown in **Fig.(1-a)** and **Fig.(1-b)**. Mixing ventilation means that the supply air and room air are mixed well by the actions of the supply jet momentum and buoyancy. The numerical results compared with the standard value due to Iraqi cooling code and ASHRAE standard . **Fig.(1-c)**, **Fig.(1-d)** and **Fig.(1-e)** shows the tested room when the exhaust grill locate on the north wall (case-I) and when the exhaust grill locate on the west wall (case-II) and when the exhaust grill locate on the east wall (case-III) respectively .

THEORETICAL ANALYSES

The numerical simulations were done using the FLUENT version (3.2.26) and GAMBIT with RNG turbulence model to create and grid the rooms geometries and then to simulate the room air distribution and concentration in enclosures. The CFD program used the Re-Normalization Group (RNG) k–ε model because it gives more accurate fitting and closer to reality than the other turbulent model for indoor airflow simulations (Lim et. al, 2005). The test chamber of the laboratory case with mixing system has the inner dimension (length x width x height): 12 x 10 x 3.5 m, which is equivalent to a typical (33) persons (0.4 x 1.1 x 0.35) m, (33) computers, (5) tables and one T.V show, all dimension and heat convective from components given in **table 1**. The heat transfer through the walls have been calculated depending on the Iraqi conditions. In Iraqi buildings the walls mainly formed from multi- materials as (cement, common brick and gypsum) as shown in **Fig.(2-a)** and floor materials shown in **Fig.(2-b)**. South and west walls exposed to outside conditions while other room sides are partitions between rooms at same design inside air temperature (24ºC Iraqi standard for cooling tables). Eight vertical poles (1,2,3,4,5,6,7 and 8) used with two horizontal lines (10,11), mid vertical plane (9) and horizontal plane at y=0.4m. Each vertical pole (2 m) high and have (101) points used to investigate temperature and CO_2 concentration in the tested room. The horizontal lines have (42) points used to calculated air distribution performance index (ADPI) inside occupied zone as shown in **Fig.3**. The environmental chamber can provide controlled air supply and inlet temperatures depending on the total cooling load. Depending on tables listed in (Iraqi cooling code, 2012), **table 2** presents the values which can be used to determine the heat transfer through the walls for present case study. The supply air temperature and velocity depend on the overall load as shown below:

Step one: Determination of heat transfer from outside to inside,(Iraqi cooling code, 2012) :
Heat transfer through internal walls due to conduction .

$Q = UA\Delta T$ (1)
$U = 1/Rt$ (2)

$$R_t = \frac{1}{h_i} + \frac{x_1}{k_1} + \dots\dots + \frac{1}{h_o} + \frac{x_n}{k_n} \quad (3)$$

Heat transfer through external walls (exposed to outside conditions) due to conduction.

$$Q = U * A * CLTDc \quad (4)$$

CLTDc: Correct cooling load temperature difference, its calculate as:
For walls:

$$CLTDc = (CLTD + LM) \times K + (25.5 - T_i) + (T_m - 29.4) \quad (5)$$

Where:
LM: Corrector latitude and month,
T_i : inside room temperature and
T_o : outlet design temperature.

For doors and windows:

$$CLTDc = CLTD + (25.5 - T_i) + (T_m - 29.4) \quad (6)$$

$T_m = T_o - D_R/2$
T_m : outlet design temperature. (^oC)
The heating load transition through the window generated by solar radiation is evaluated as :

$$Q_r = A * SC * SHG * CLF \quad (7)$$

Step two: Determine the supply air temperature (T_s) and air flow rate. There are some steps are based on standards of ASHRAE Research's (Chen, 1999 & Chen, 2003) to find ventilation rate and supply air temperature (T_s).

$$Q = (0.295q_{oe} + 0.132q_l + 0.185q_{ex})/(\rho\, cp\, \Delta T_{hf}) \quad (8)$$

$$Ts = T_{sp} - \Delta T_{hf} - [(A_z.q_t) / (0.584Q^2 + 1.208A.Q_s)] \quad (9)$$

$$q_t = q_{oe} + q_l + q_{ex} \quad 10)$$

Then assume ΔT_{hf}=3^{o}C (ANSI/ASHRAE Standard, 2001), T_{sp}=24, T_o=47^{o}C (Iraqi cooling code, 2012), the values of flow rate (Q) and air temperature supply (T_s) can be determine as list in **table 3**.

The air change per hour (ACH)

$$ACH = (Q/V_{Room}) * 3600 \quad (11)$$

INDOOR AIR QUALITY PARAMETERS-:

Ventilation systems are designed and operated for the purpose of maintaining good thermal comfort and indoor air quality and their performance could be assessed based on a number of specific indices describing how efficiently the system achieves these goals. Heat removal effectiveness (ε_t) and contaminate removal effectiveness (ε_c), are defined to represent the ability of a ventilation system of removing heat and internally generated pollutants from a ventilated space, expressed by (Awbi, 2003, Novoselac, A. et al. 2002)

$$\varepsilon_t = \frac{T_o - T_i}{T_m - T_i} \quad (12)$$

$$\varepsilon_c = \frac{C_o - C_i}{C_m - C_i} \quad (13)$$

In equations (10 & 11), T is the air temperature (°C), C is the contaminate concentration in part per million (ppm), subscripts $_o$, $_i$ and $_m$ denote outlet, inlet and mean values in the occupied zone.

AIR DISTRIBUTION PERFORMANCE INDEX (ADPI)

The air distribution performance index (ADPI) is a percentage that is defined by the number of points measured in an occupied zone where EDT is within the set limit (> -1.7°C and < 1.1°C) over the total number of points measured in it (Awbi, 2003). ADPI of 60 to 69 is considered as unsatisfactory, 70 to 79 as satisfactory and 80 and above as good air distribution. The air distribution performance index rating of an air diffusion system depends on a number of factors: **1.** Outlet type **2.** Room load
3. Room dimensions and diffuser layout **4.** Outlet throw

When properly selected, most outlets can achieve an acceptable ADPI rating. The higher the ADPI rating, the higher the quality of room air diffusion within the space. Generally an ADPI of 80 is considered acceptable. The effective draft temperature, is given as (Awbi, 2003):

$$EDT = (T_x - T_r) - 8(V_x - 0.15) \quad (14)$$

Vx : is the local velocity and the maximum air velocity was taken as 0.35m/s.

NUMERICAL SIMULATION

The airflow transport can be described by the following time-averaged Navier– Stokes equation:

$$\mathrm{div}\left(\rho V \emptyset - \Gamma_{\varphi,\mathrm{eff}}\, \mathrm{grad}\, \emptyset\right) = S_{\emptyset} \quad (15)$$

where ρ is the air density (kg/m^3), $\Gamma_{\varphi,\mathrm{eff}}$ is the effective diffusion coefficient (kg/m.s), V is the air velocity vectors (m/s), S is the source term of the general flow property, and $\emptyset$ is any one of the components shown in **table 4**. When $\emptyset = 1$,the general equation becomes the continuity equation. The effective diffusion coefficients and the source terms for different $\emptyset$ are listed in **table 4**. In the table, the effective viscosity, μ_{eff}, is the sum of molecular viscosity and turbulent viscosity.

Boundary Conditions and Numerical Solution

Exact simulation of airflow in a room by CFD depends on proper specification of the boundary conditions by the user. The surface temperatures on the ceiling, floor, side walls, window are measured and listed in **table 3** to use as boundary conditions for the walls and heated objects in CFD simulations. The CO_2 concentration at the supply air is assumed to be 350 ppm in the CFD study and the indoor CO_2 sources are from the occupants. According to the (ASHRAE Standard, 2001),the CO_2 concentration should be less than 1000 ppm in the room. Velocity inlet boundary conditions for both air inlet and contaminate source were considered uniform and the flow was normal to inlet section. One advantage of taking the density of both constituents to be equal and constant is the possibility of making use of the Boussinesq approximation to model buoyancy effects. The standard FLUENT wall function (no slip, smooth and no diffusive flux of the species) was used. For discretization, the body force weighted scheme for pressure, the SIMPLEC scheme for pressure-velocity coupling, and the second-order upwind scheme for momentum, turbulence and species were chosen, (Fluent Inc. 2006). Under relaxation factors were manipulated to get quick convergence and solution was assumed to converge when the residuals for all scalars were less than or equal to 10^{-3} and 10^{-6}.

RESULTS AND DISCUSSION

Numerical simulations was conducted to analyze the temperature and CO_2 concentration distributions with the change of exhaust grills location. The simulated results were compared with design data (setup temperature and temperature difference from head to foot level) to validate the CFD model.

Temperature Distribution

Fig.4 shows the air temperature distribution in each poles for the three cases with convergent distribution except in the remote regions where a clear increase appears on the temperature distribution with poles (7&8) for the three cases, reaching temperatures in the first case, to (35.5ºC) near the western wall in the lower areas, this is due to the location of this poles between the heat source (persons) in addition to the position near the window. While the temperatures dropped with other cases and reach to maximum value (34ºC) in same region for case-II and reach to (26.1ºC) for case-III as shown in **Fig. 4**, poles 4&6 recorded the increment in temperature with case-II due to location near the exhaust grill. Average temperatures throughout for all room was (35.3ºC) for the first case while (30.443ºC) for second case and (28.42ºC) for third case and this leads to raising the thermal efficiency and increase the air performance distribution index, on the other hand the temperature difference from head to foot level was larger for the first case as shown in **table 5**. For the above cases, the air temperature changes between the head and foot level of a sedentary occupant are less than 3 K as stipulated in ASHRAE.

Figs. 5&6 show the distribution of air temperature contours for three cases at x=5m and at y=1.8m respectively. For all cases, the relatively cold air approximately 18ºC from the ceiling diffusers first sprays down to the floor driving by its initial momentum as well as density difference. Regular distribution of temperature which appears in the third case leads to increase the thermal efficiency as shown in table 4. where It can be seen that a region of medium temperature (23^{o}C - 27^{o}C) exists in the activity space of the occupants (occupied zone), while the temperature at the south wall is equal 33.3^{o}C because this wall exhibition to high external temperatures higher than other walls. Exhausts located on the same wall as the location of the inlet are always better than their respective exhausts on the far wall; for example, case-II is better than case-I. This is because when the exhausts are in the opposite wall, the residence time for the ventilation air is small, and it gets less chance to be mixed with the room air to carry it out of the tested room. On the other hand, when the exhausts are on the same wall, air jet disperses, hits the opposite wall, disperses much more, and is thus recirculate and mixed well with the room air to push them out of the room.

CO_2 Concentration Distribution

Fig. 7 show the CO_2 concentration with each poles for the three cases. The highest percentage concentration has been observed at the pole-7 about (500 ppm) with (case-I) due to large distance from supply diffuser (remote areas that are not up to it fresh air), (Zhijian, Angui, Li, and Xin , 2010) show the same result where it was found that the higher pollutant concentration in the corner of the room. In (case-II) the contaminate decreasing in the whole room for the same reason with the temperature distribution, while case-III was the better than the other cases because of the air supply and exhaust grills are located at the same high on the opposite walls for this reason lowering the contaminate level too high. High velocity removing more contaminate of the tested room, (Jiaqing and Chang, 2010) notes that high airflow velocity allows a better removal of contaminate comparison with the other case.

In general, exhaust locations at the top of the wall (near the roof) are better than the exhausts near the floor, this is because inlet jet can pick up the upward moving contaminate and carry the contaminate out with it; for example case-III is better than case-II, because it is directly opposite to the inlet jet in which case the inlet jet is short-circuited and allows fresh air to blow out of the room before mixing with the contaminate.

Figs. 8&9 show the CO_2 concentration in breathing zone at y= 0.4m and at y=1.8m above the floor. The ambient CO_2 level is 350 ppm. The indoor CO_2 are from the occupants, and the occupants are the heat sources as well. The heat produces updraft clouds that can bring the CO_2 to the upper zone. Concentration of most of the place is about (341–728) ppm for case-I while about (351-556) ppm for case-II and (335-492) ppm for case-III, this can maintain a better IAQ, especially for breathing zone and increasing pollution efficiency as shown in **table 5**.

Table 5 shows the value of the ventilation indices given by equations (12) to (14),obtained from the CFD calculations for comparison between the three cases.

CONCLUSIONS

Exhaustive computational investigation was performed to study physical factors that affect distribution of temperature and contaminate concentration in a laboratory.

The following conclusions are obtained from this study:

- The exhausts location is very important to take into account to reach human comfort conditions, where higher heat and contaminate removal effectiveness efficiency obtained when the exhausts located near the roof than the location near the floor (for example, case-III is better than case-I and case-II) which was due to short circuiting of fresh inlet air.
- Mechanical ventilation should be employed in the laboratory, especially, when there is large population density in laboratory. Variation of supply air velocity could avoid stagnation zone.
- To allow an optimal ventilation design, more specific performance requirements need to be specified such as type of supply, thermal requirements, influence of laboratory equipment, influence of personnel, etc.

Table 1: Dimensions and heat convective from components

Items	**No.**	**Dimensions, m**			**Heat flux (Iraqi cooling code,2012) W**
		X	Y	Z	
Persons	33	0.4	1.1	0.35	75 per person
Table	5	0.8	0.75	2.5	0
T.v. show	1	0.1	0.6	1	80
Supply	3	0	1.08	0.28	-
Exhaust	1	0	0.3	0.2	-
Lights	31	0.1	0.1	0.25	40
Lamps	5	0.1	0.05	1.3	36
Computers	33	0.37	0.35	0.4	60
Door	1	0.2	1	1.5	-

Table 2: Heat transfer details through walls and window at 1/ September,12:00 o'clock in the city of AL- Hilla, Latitude 32.°

Walls	CLTD	LM	K	SHG	SC	CLF
West wall	8 (group -B)	0.5	-1.1	-	-	-
south wall	6 (group -B)	0.5	3.8	-	-	-
Window	5	0	1	710	0.94	0.14
h_i(inside room)= 8.29 w/m^2.k & h_o(outside room) = 22.7 w/m2.k						

Table 3 : Values of air flow rate, supply temperatures and heat transfer through the walls.

			Heat transfer (W)	
			window	
Q L/s	T_s	ACH	**conduction**	**Radiation**
686	17 ºC	6	300.38	1008

Table 4: Terms, coefficients and constants in Eq. (15)

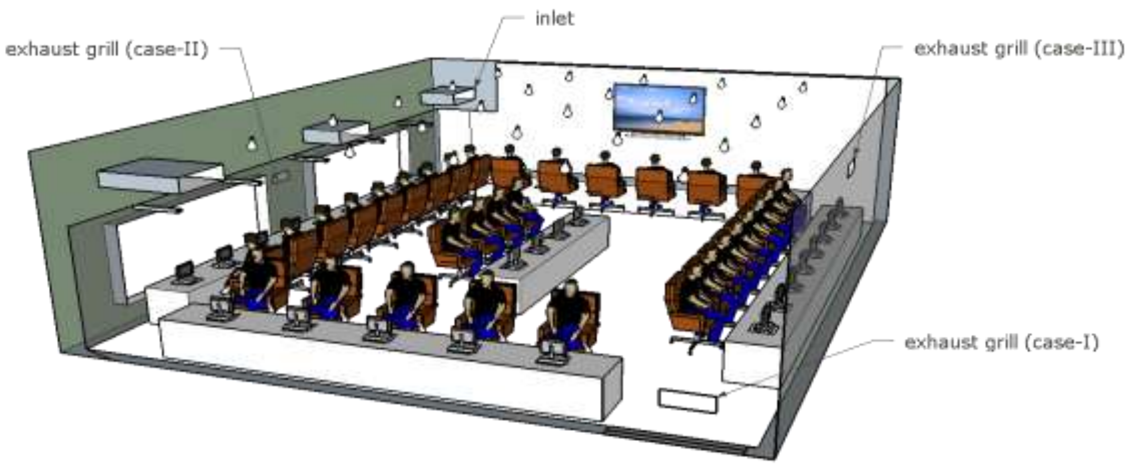
Equations	Ø	$\Gamma_{\varphi,eff}$	$S_{\emptyset}$
Continuity	1	0	0
Momentum	U_i	μ_{eff}	$-\frac{\partial p}{\partial x_i}+g_i(\rho-\rho_o)$
Turbulence kinetic energy	k	μ_{eff}/σ_k	$P_k+G_b-\rho\varepsilon+G_k$
Turbulence kinetic energy dissipation rate	ε	$\mu_{eff}/\sigma_\varepsilon$	$\frac{\varepsilon(C_1 p_k-C_{2\varepsilon})}{k}+\frac{C_3 G_K \varepsilon}{k}-R_\varepsilon+S_\varepsilon$
Thermal energy	T	μ_{eff}/σ_T	S_T
Concentration	C	μ_{eff}/σ_C	S_C

$P_k=\mu_t(U_{i,j}+U_{j,i})U_{i,j}$, $\mu_{eff}=\mu_t+\mu=\mu\left[1+\sqrt{\frac{C_\mu}{\mu}\frac{k}{\sqrt{\varepsilon}}}\right]^2$, $(\sigma_k,\sigma_\varepsilon,\sigma_T,\sigma_C,C_\mu,C_1,C_2,C_3)=(1.0,1.314,0.9,1.0,0.0845,1.42,1.68,1.233)$.

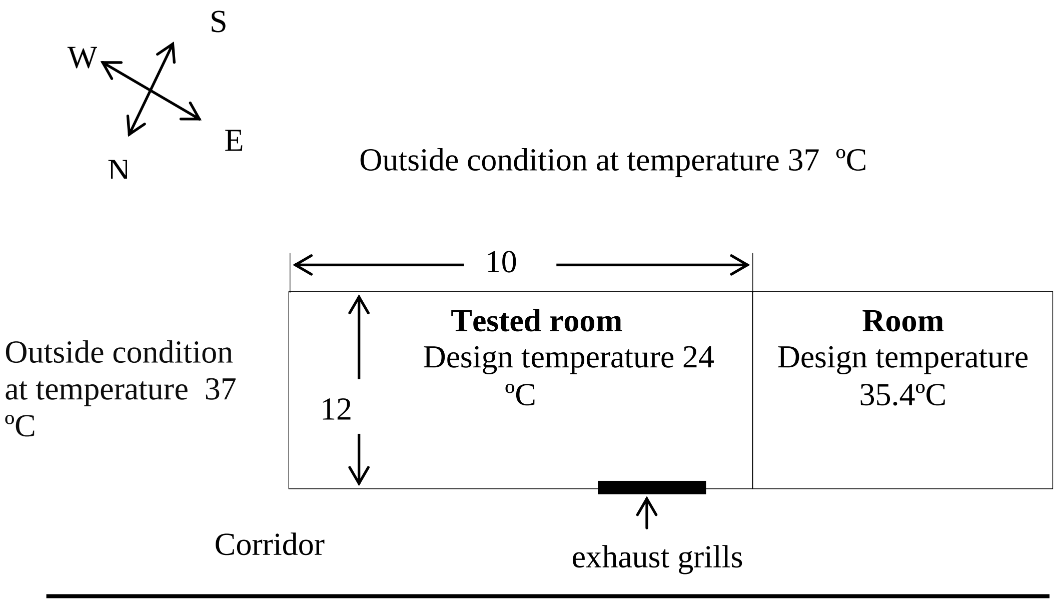
Table 5 : Major parameters used in the simulations.

Parameters	Case-I	Case-II	Case-III
Heat removal effectiveness (ε_t)	0.57	0.8036	0.864
Contaminate removal effectiveness(ε_c)	0.428	0.506	0.731
APDI	62.3	66.8	75.49
ΔT (K)	1.926	1.371	0.903

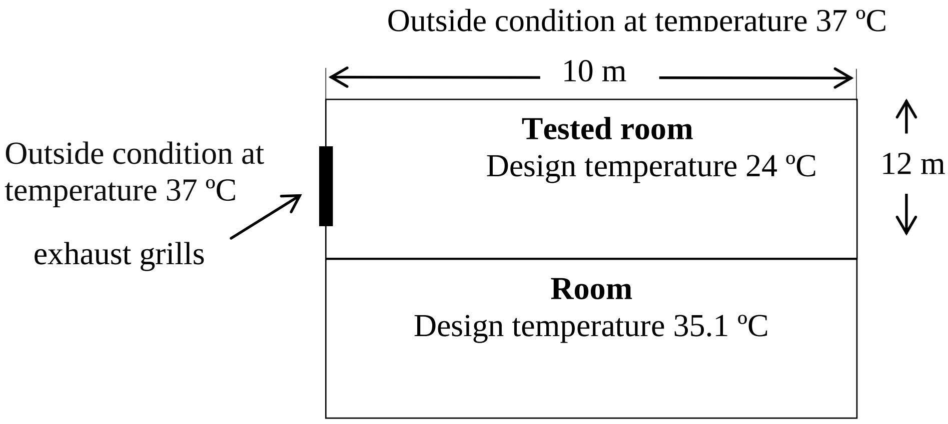
a



b



c



d

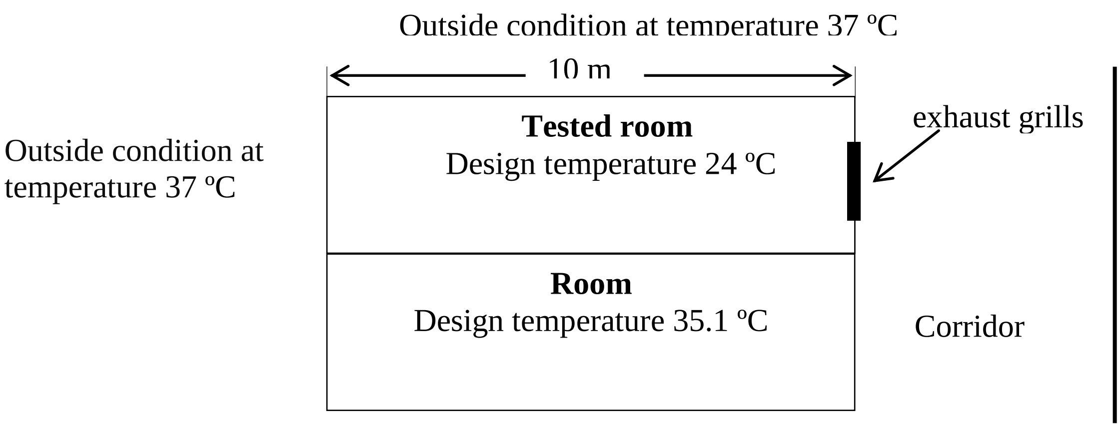


Fig. 1 (a) Photograph of computer laboratory adopting MV system.
(b) Schematic diagram for tested room
(c) Schematic diagram for tested room, (Case-I).
(d) Schematic diagram for tested room (Case-II).
(e) Schematic diagram for tested room, (Case-III).

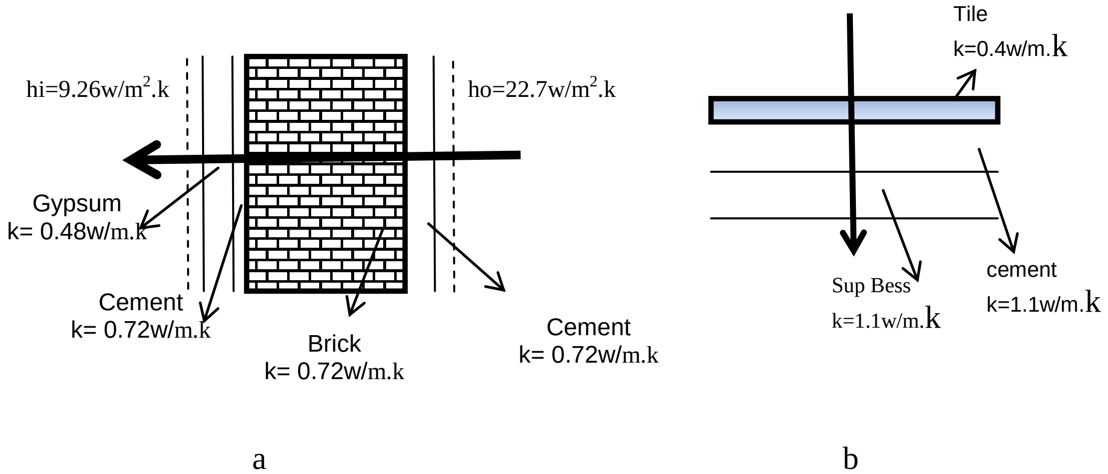


Fig.2 (a) Heat transfer through the wall, (b) Heat transfer through the floor.

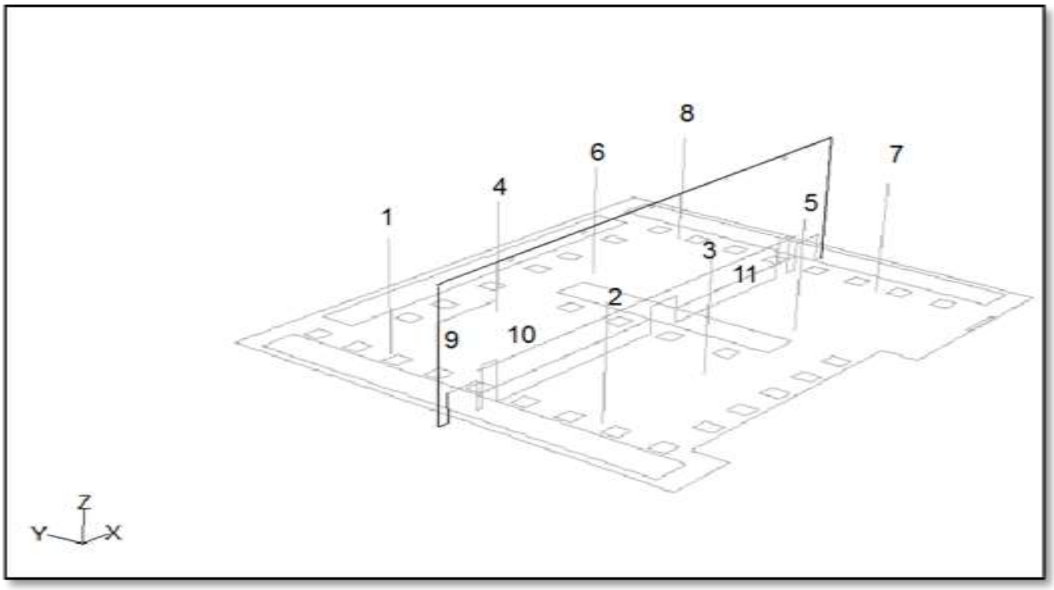
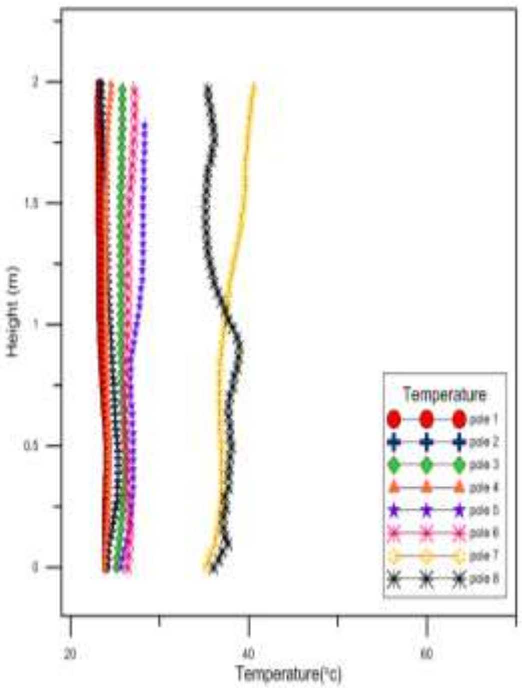
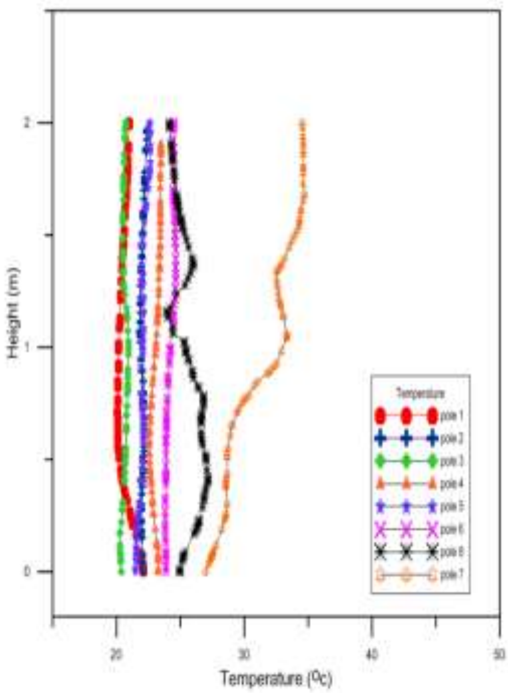


Fig. 3 Different locations of poles and planes.

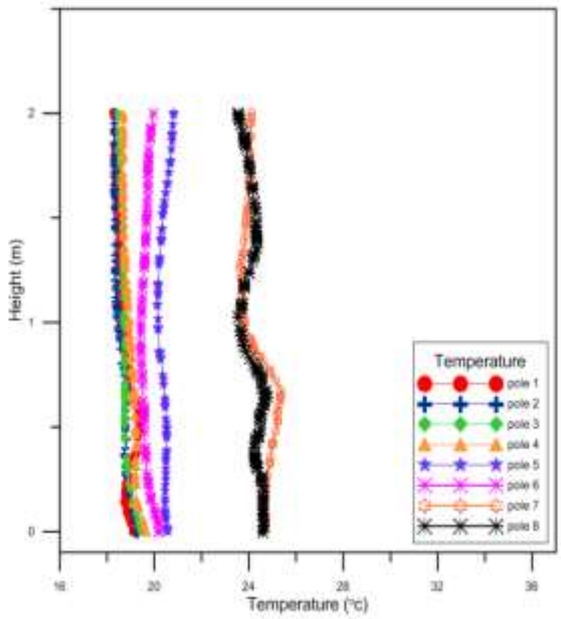
1-Pole at (x=2 and y=3), 5-Pole at (x= 8 and y=8), 9- Plane at y=5m
2-Pole at (x= 8 and y= 3), 6- Pole at (x= 2 and y=8) 10- Line at (z= 0.4 & x=5)
3-Pole at (x=8 and y=5), 7- Pole at (x= 8 and y=11) 11-Line at (z= 1 & y=5) .
4-Pole at (x= 2 and y=5), 8- Pole at (x=2 and y=11)



a

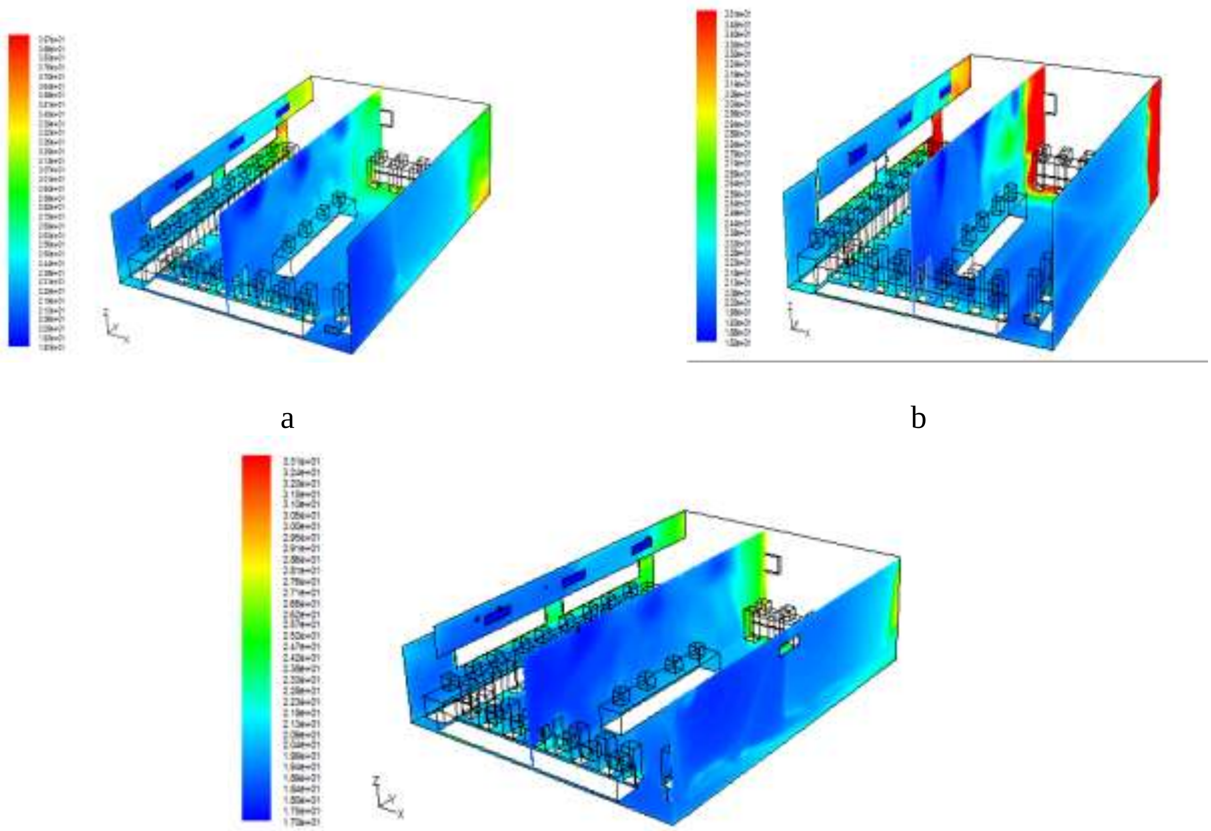


b



c

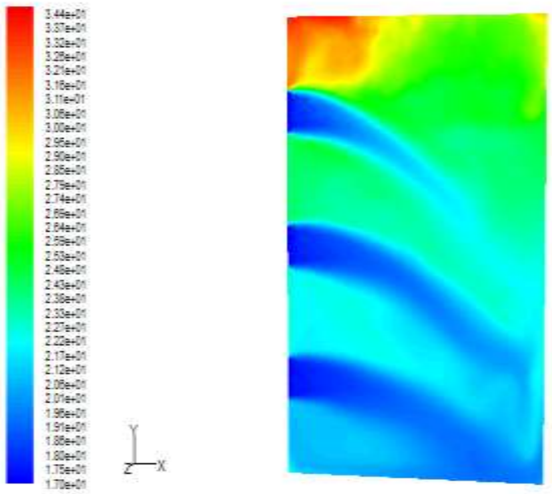
Fig. 4 :- Air temperature distribution with height at seven poles for laboratory: (a) case-I , (b) case-II, (c) case-III



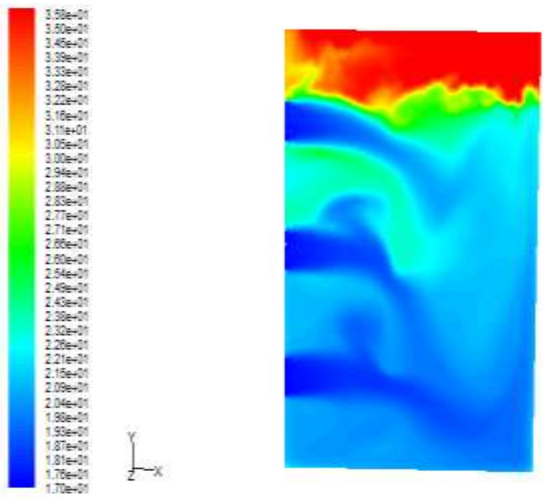
a b

c

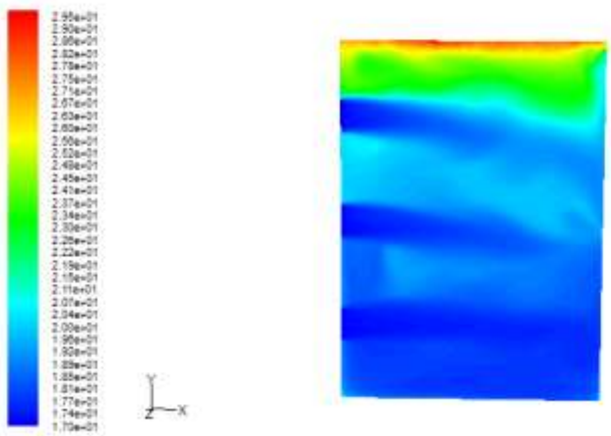
Fig.5 Distribution of air temperature contours for Plane at x= 5m, (a) case-I, (b) case-II, (c) case-III



a



b



c

Fig. 6 Distribution of air temperature contours for Plane at y= 1.8m, (a) case-I, (b) case-II, (c) case-III.

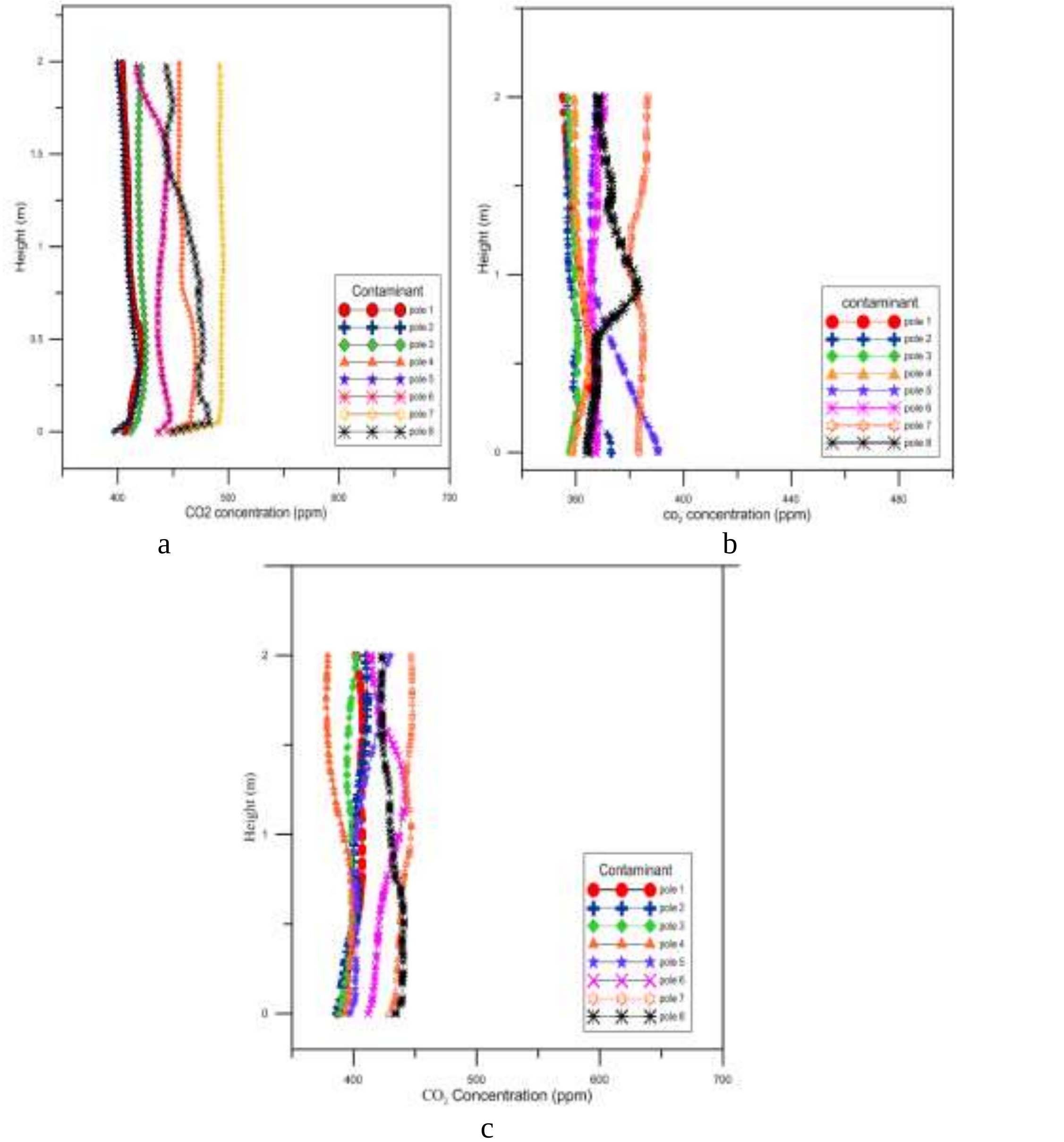


Fig. 7 CO_2 concentration with height at seven poles for laboratory: (a) case-I , (b) case-II, (c) case-III

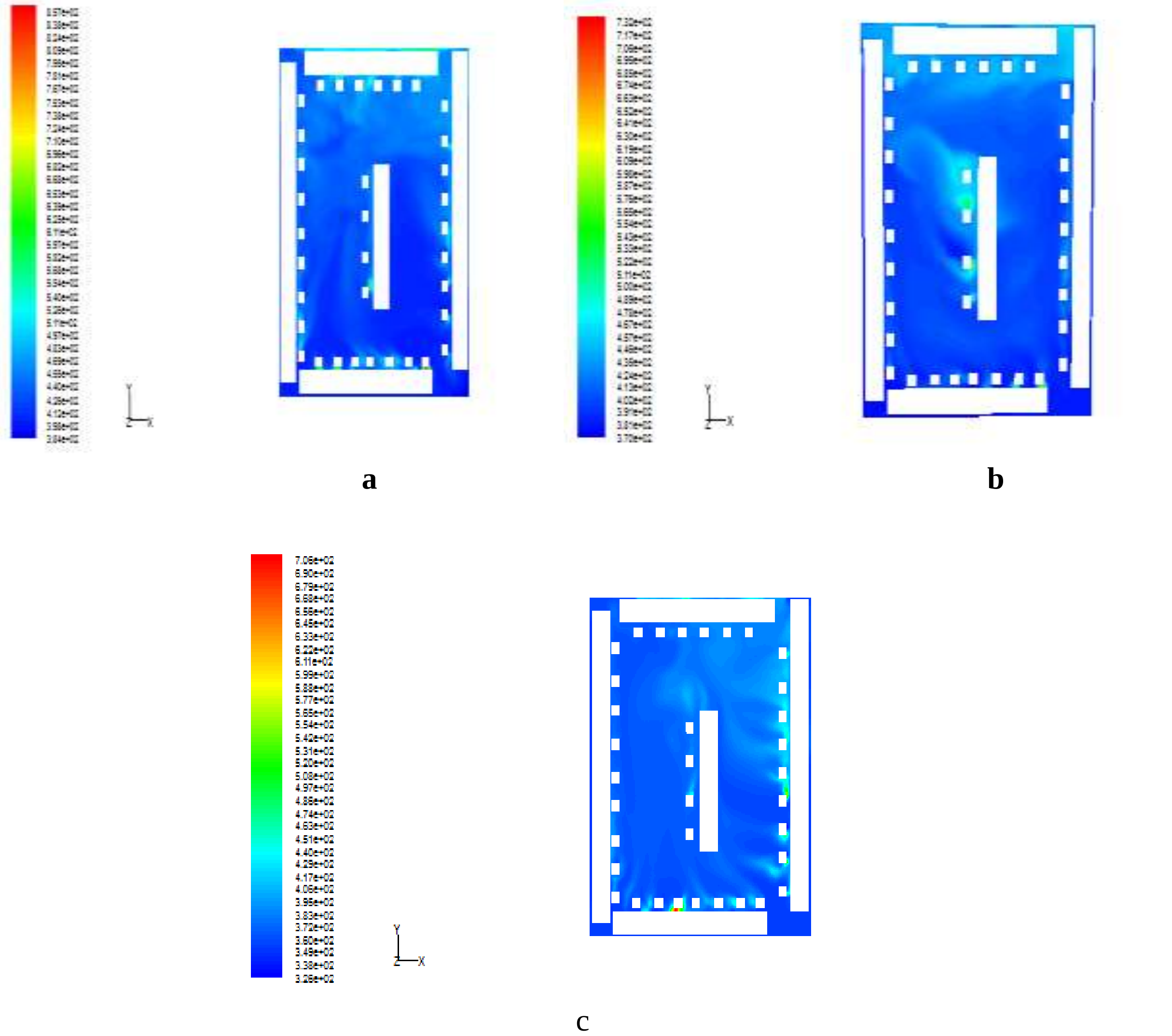
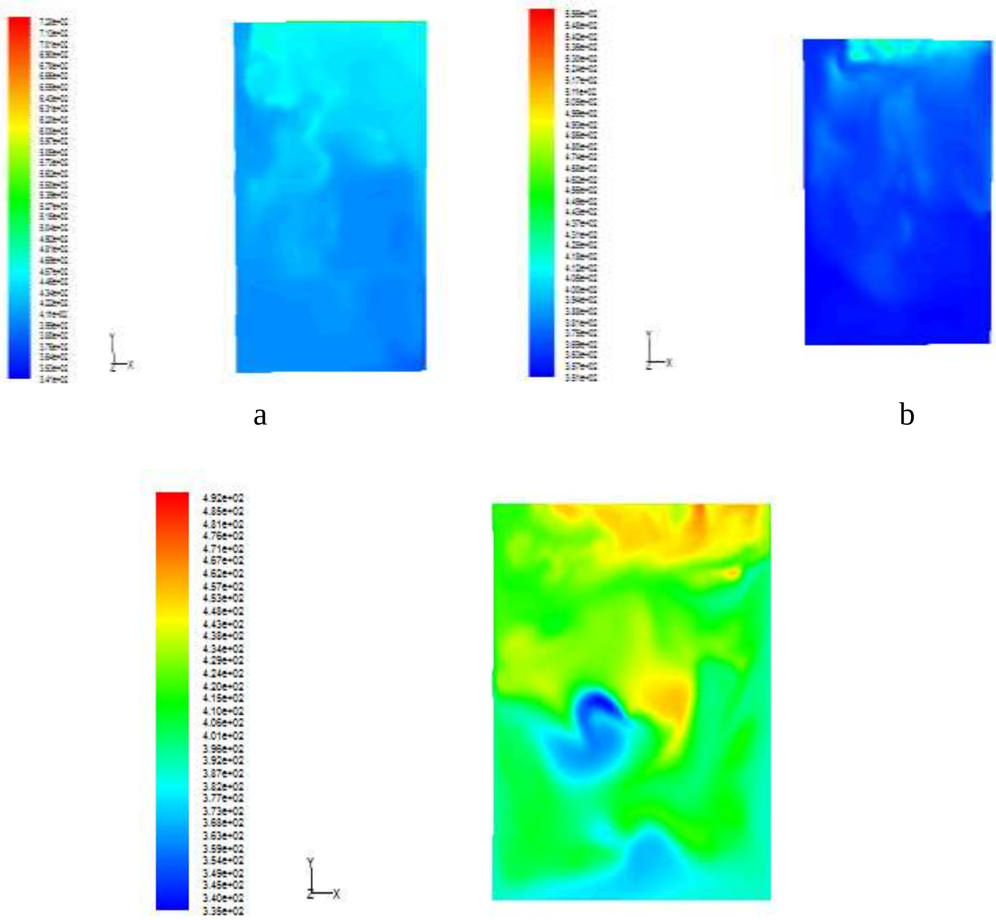


Fig. 8 Contour of contaminate concentration at seven poles with y= 0.4m for laboratory: (a) case-I , (b) case-II, (c) case-III



a b c

Fig. 9 Contour of contaminate concentration at seven poles with y= 1.8m for laboratory: (a) case-I , (b) case-II, (c) case-III

REFERENCES :-

ANSI/ASHRAE Standard 62-2001 “Ventilation for Acceptable Indoor Air Quality American Society of Heating”, Refrigerating and Air-Conditioning Engineers.

Chen, Q. & Glicksman, L., “System performance evaluation and design guidelines for displacement ventilation”, Section J, Atlanta, GA: American Society of Heating, Refrigerating, and Air-conditioning Engineers, Inc, (2003).

Fluent Inc. (2006). Fluent 6.3.26 user’s guide.

Gery Einberg “Air diffusion and solid contaminate behavior in room ventilation – a CFD based integrated approach”, Ph. D thesis, kungliga tekniska högskolan university, (2005).

Glenn Reynders and Dirk Saelens, "numerical and experimental evaluation of ventilation in laboratories: a case study", building physics section, department of civil engineering, K.U.Leuven, Belgium, (2011).

H. B. Awbi "Ventilation of Buildings", Spon Press, second edition, 2003.

Novoselac, A., Srebric, J. A critical review on the performance and design of combined cooled ceiling and displacement ventilation systems, Energy and Buildings, Vol. 34, Issue 5, pp. 497-509, (2002).

Lim, Soon Chong Johnson and Husain, Adnan and Tee, Boon Tuan "Simulation of airflow in lecture rooms", Proceedings of The AEESAP International Conference, Kuala Lumpur, 2005.

Iraqi Code for cooling, University of Technology, The Ministry of Construction and Housing - Technical Department, (2012).

Jiaqing Zhou and Chang Nyung Kim "Numerical investigation of indoor CO2 concentration distribution in an apartment", Department of Mechanical Engineering, Graduate School, Kyung Hee University, Korea, PP.446-701, (2010).

T. Karimipanah1, H.B. Awbi, M. Sandberg and C. Blomqvist, "Investigation of air quality, comfort parameters and effectiveness for two floor level air supply systems in classrooms", Journal of Building & Environment, (2207).

Zhijian Liu, Angui Li, Li Tao, and Xin Jiao, "Correlation between indoor air distribution and pollutants in natural ventilation", Bioinformatics and Biomedical Engineering (ICBBE), (2010).