ASSESSMENT OF VARIOUS TURBULANCE MODELS IN DISPLACEMENT VENTILATION SYSTEM FOR PREDICTION OF AIRFLOW AND TEMPERATURE DISTRIBUTION WITH EXPERIMENTAL DATA

تقييم عدة نماذج اضطراب باستخدام نظام التهوية الازاحية لتنبؤ حركة الهواء وتوزيع درجة الحرارة

Ali Aedan Shbeeb
University of Babylon
Department of Mechanical Engineering
aliaedan@yahoo.com

Asst. Prof.Dr. Ala'a Abbas Mahdi
University of Babylon
Department of Mechanical Engineering
alaa.abbas59@yahoo.com

ABSTRACT

This work presents an experimental and numerical study for predicting the indoor airflow and temperature distribution by adopting a displacement ventilation system within isothermal office room. Computational fluid dynamics (CFD) are used to simulate the indoor airflow and temperature distribution by using (FLUINT6.3.26) and (GAMBIT2.4.6) software for solving the Navier Stocks, energy and turbulence equations using finite volume techniques. Assessment was applied for different turbulent models (Standard K-Ɛ, RNG K-Ɛ, Realizable K-Ɛ, and SST K-ω) for a case study and its detailed geometry. The numerical results for the four turbulent models was compared with experimental readings. It was concluded that the (RNG K-Ɛ) turbulent model gives more accurate and acceptable results than the other three models.

KEYWORD

Displacement ventilation, CFD, Indoor air quality, Turbulence, Numerical model, Isothermal

الخلاصة:

يتضمن البحث دراسة عملية ونظرية لحركة الهواء وتوزيع درجة الحرارة داخل حيز باستخدام نظام التهوية الازاحية لنموذج غرفة مكتب معزولة. استخدم برامج الحاسوب (FLUINT6.3.26) و(GAMBIT2.4.6) لحل معادلات نافير ستوك باستخدام طريقة الفروقات المحددة لمحاكاة وتحليل النموذج المدروس. تم اختبار وتقييم اربعة نماذج اضطراب(Standard K-Ɛ, RNG K-Ɛ, Realizable K-Ɛ and SST K-ω) ، ثم قورنت النتائج مع القراءات العملية حيث تم التوصل الى ان موديل (RNG K-Ɛ) هو الموديل الاكثر مقبولية بين الموديلات الثلاثة الاخرى في الحالة المدروسة.

Nomenclature

Symbols	Description	Unit
A_f	Surface area for floor	m^2
A_s	Face area for diffuser	
C_p	Specific heat of the air at constant pressure.	kJ/kg.K
$C_{1\varepsilon}$, $C_{2\varepsilon}$	model constant	
dx dy dz	Control volume	m
$k_{i,j,k}$	Turbulent kinetic energy at cell (i,j,k)	m^2/s^2
P	Pressure	N/m^2
P_z	zone population	person
Q_{DV}	Air required to satisfy the sensible cooling load in a DV system	l/s
q_{ex}	The heat conduction through the room envelope and transmitted solar radiation,	W

q_l	Cooling load for the overhead lighting	W
q_{oe}	Cooling load for occupants, desk lamps and equipment.	W
Q_{oz}	Fresh air flow rate	l/s
R_A	Outdoor air flow rate required per unit area	l/s
R_p	Outdoor air flow rate required per person	l/s
S	Source term for the rate of thermal energy production	J/kg
T_{av}	average room temperature	^oC
$T_{i,j,k}$	Temperature at cell (i,j,k)	^oC
ΔT_{hf}	Temperature difference from head to foot level.	^oC
T_{sp}	Setup (design)temperature.	^oC
U_s	Supply air velocity	m/s

Abbreviations

ASHRAE	American society of heating refrigeration and air conditioning engineering
ACH	Air change per hour
CFD	Computational Fluid Dynamics
DV	Displacement Ventilation
IAQ	Indoor air quality

Sup-Scripts

av	Average	oe	Occupants and equipment
f	floor	sp	Setup
l	Overhead light	t	total

Greek letters

ρ	Air density	kg/m^3
ε	Turbulent energy dissipation rate.	
σ	Prandtl or Schmidt number	
ω	specific dissipation rate	1/s
Γ	Diffusion coefficient (diffusivity)	m^2/s

INTRODUCTION

Displacement Ventilation (DV) can be defined as “Room ventilation created by room air displacement, by deliver air at low level and velocity in a space at a low air temperature” and its suitable for large spaces,[1] and in simply may be define as any airflow pattern where “old” air is displaced by “new” air, [2].

Many experimental and simulation works have been done to investigate airflow pattern, temperature and concentration distribution in DV systems. For example, Hyder Mohammed(2013) ,[3], presented a theoretical study about three-dimensional ventilation problems. The investigation has studied the flow for four different diffusers (displacement, grille, slot, and square diffusers) were tested by using two turbulence models (Rk-ε and k-ω). This study showed there is a good agreement with (k-ω) model gives more accurate fitting and closer to reality. Chen (1995),[4], compared eight modified k-ε models and concluded that the Renormalization Group (RNG) k-ε model performs best among all the eddy-viscosity models tested for mixed convection flow. Bunn et al. (1991),[5] reported a CFD and laboratory investigation on a combined system of displacement and chilled beams. It was found that the buoyancy produced by the displacement principle was enhanced by chilled beams. Dickson D.(1994),[6] reported experimental results regarding displacement and mixing ventilation, it was shown that the room air temperature distribution was uniform for both systems, although temperature profiles were strongly dependent on the supply air temperature when using DV. Yuan et al. (1999),[7] provided experimental data of DV for a small office, a large office with partitions, and a classroom with regard to airflow pattern, temperature and concentration distribution. Lau and Chen (2007),[8] studied a workshop with floor-supply DV. It was found that indoor air quality can be improved due to the contaminant concentration in the breathing zone is lower than that of mixing system.

Most of the researchers have discussed (the location and number of diffusers, Comparing displacement ventilation with mixing ventilation system, position of the exhaust grills and effect of air change per hour on thermal comfort). However, the shortages of these studies in ability of using of displacement ventilation in the arid and hot climate for testing the occupied zones under Iraqi climate by adopting experimental and numerical investigation, after that assessment various turbulent models. Therefore, the present study will focus on:

- Assessment of four turbulent models (standard K-Ɛ, RNG K-Ɛ, realizable K-Ɛ, and SST K-ω) in order to select the acceptable model, then compared with experimental work under Iraqi climate by use displacement ventilation system.
- Calculate of the actual values of input flowrate and air supply temperature needed by the tested rooms depending on space heating load.
- Choosing the shape of diffuser depending on the value of air flowrate.
- Examining the possibility of using the displacement ventilation type under Iraqi climate experimentally and theoretically.

This work, conducted a set of full-scale room experiments to study the airflow and temperature distribution using DV system by calculating the heat from occupants and equipment and estimating the actually magnitude of the inlet air flow rate and temperature needed for best ventilation. The influence on air flow pattern and temperature distribution around occupants due to multiple heat sources are investigate. The main objective of present study is to show the more suitable turbulence model among the four models (standard K-Ɛ, RNG K-Ɛ, realizable K-Ɛ, and SST K-W) to predict airflow motion and temperature distribution by using CFD technique and obtain higher level of indoor air quality (IAQ) and human comfort.

TESTED ROOM SETUP

Tested room equipment

A set of full-scale office room experiment is conducted to study the airflow, temperature transport with the DV system. The tested room delivered and setup by all the necessary equipment for DV system as shown in Fig.(1).

Modelling of the office room

The dimensions of tested room is (3m *1.75m and 3m height) as shown in Fig.(2).The inlet air supply device was located at the south wall and the outlet grille was placed in the center of the upper side for the north wall (0.5m from the ceil) . One occupant, one computer and one lump were placed as heat sources. The cooled air delivered by using an air-condition of (24000 Btu/hr) cooling load. A full description of the configuration and location of the different details are given in table (1). Human body properties represent by used a seated manikin, Fig.(3), since the height of manikin is 1.1m and surface area is1.65m^2.Two lumps are dispensed inside of manikin to generate sensible heat of 75W,(Iraqi cooling code).

Air handling and supply system

There are two steps to find ventilation rate and supply air temperature for displacement ventilation applications.The procedures are based on the findings of ASHRAE research, Project-949.

Step one: Supply flow rate

a- Calculation of the cooling load ventilation air flow rate, Q_{DV}:

The amount of air flowrate air needed for ventilated room is calculated as,[9]

$$Q_{DV} =(0.295q_{oe}+0.132q_{l}+0.185q_{ex})/ (\rho\ c_p\ \Delta T_{hf}) \qquad (1)$$

b- Calculation of fresh air flow rate, Qoz:
From ASHRAE Standards 62.1-2004,[10] Eqn.(2) is used to determine the flow rate of fresh air.

$$Q_{oz} = \frac{R_p P_z + R_A A_f}{E_z} \qquad (2)$$

The values of (R_p, R_A, P_z and E_z) determined from (ASHRAE standards 62.1-2004) [10]. Then, choose the large value of flowrate calculated from (Eqns 1&2) as the design flow rate of the supply air:

Q_s= Max [Q_{AV}, Q_{oz}]

Step two: Supply air temperature:

The supply air temperature (T_s) for displacement ventilation applications is estimated as, [11].

$$T_s = T_{sp} - \Delta T_{hf} - [(A_f .\, q_t) / (0.584 Q_{DV}^2 + 1.208 A_f . Q_{DV})] \qquad (3)$$

Where

$$q_t = q_{oe} + q_l \qquad (4)$$

Now to find air flowrate and supply temperature used the following values,[12]:

q_{person}	$q_{computer}$	T_{sp}	ΔT_{hf}
75 W	45 W	25 ^oC	3 ^oC

Then values of air flowrate (Q_s) and air supply temperature (T_s) can be determined and listed in **table(2)**. The air change per hour (ACH) can be calculated by using Eqn.(5),[13].

$$ACH = (Q_s/V_{Room})*3600 \qquad (5)$$

SELECTION OF DIFFUSER

The supply air unit is originally designed as a general displacement unit. In displacement ventilation there are some goals must be satisfied such as calm operation ,thermal comfort ,low velocity and low noise diffuser. There are more than one type of diffusers as rectangular diffusers and circular diffusers, in this work rectangular diffusers (DF1C series type) was used. Eqn.(6) used to calculate area of diffuser, by assumed the air supply velocity 0.25m/s, then the dimensions of supply air unit are (0.4m *0.15 m) as shown in Fig.(4).

$$Q_s = u_x * A_s \qquad (6)$$

MEASUREMENTS PROCEDURES

To measured air temperature distributions inside the tested room, two vertical poles were placed. Its carried 10 thermocouple (K-type) to measured temperature distribution, as shown in Fig.(5). Five measurement devices were fixed on each pole. Probes were placed at height of 0.4m to 1.8m from the room floor which is within the range of occupied zone. The air flowrate ant temperature takes for each pole at different levels from ground (0.4m, 0.8m, 1m, 1.4m and 1.8m) as shown in Fig.(6). The location of poles inside the tested room shown in Fig.(7).

EXPERIMENTAL READINGS

The experimental readings recorded in the day of 12^{th} /Aug (2015) at steady state time (14:00 O'clock) and then listed in **table(3).**

CFD MODELLING:

Flow assumed to be steady, three-dimensional flow, incompressible and turbulent .The working fluid is air which is assumed.

The governing equations of motion based on Navier-Stockes conservation equations which are continuity, momentum and energy equations as follows,[14].

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\rho v)+\frac{\partial}{\partial z}(\rho w)=0 \qquad (7)$$

$$\frac{\partial}{\partial x}(\rho UU)+\frac{\partial}{\partial y}(\rho UV)+\frac{\partial}{\partial z}(\rho UW)=-\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}(\mu\frac{\partial U}{\partial x})+\frac{\partial}{\partial y}(\mu\frac{\partial U}{\partial y})+\frac{\partial}{\partial z}(\mu\frac{\partial U}{\partial z}) \qquad (8)$$

$$+\frac{1}{3}\frac{\partial}{\partial x}\left[\mu(\frac{\partial U}{\partial x}+\frac{\partial V}{\partial y}+\frac{\partial W}{\partial z}\right]+\rho g_x$$

$$\frac{\partial}{\partial x}(\rho UV)+\frac{\partial}{\partial y}(\rho VV)+\frac{\partial}{\partial z}(\rho VW)=-\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}(\mu\frac{\partial V}{\partial x})+\frac{\partial}{\partial y}(\mu\frac{\partial V}{\partial y})+\frac{\partial}{\partial z}(\mu\frac{\partial V}{\partial z}) \qquad (9)$$

$$+\frac{1}{3}\frac{\partial}{\partial y}\left[\mu(\frac{\partial U}{\partial x}+\frac{\partial V}{\partial y}+\frac{\partial W}{\partial z}\right]+\rho g_y$$

$$\frac{\partial}{\partial x}(\rho UW)+\frac{\partial}{\partial y}(\rho VW)+\frac{\partial}{\partial z}(\rho WW)=-\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}(\mu\frac{\partial W}{\partial x})+\frac{\partial}{\partial y}(\mu\frac{\partial W}{\partial y})+\frac{\partial}{\partial z}(\mu\frac{\partial W}{\partial z})+ \qquad (10)$$

$$\frac{1}{3}\frac{\partial}{\partial z}\left[\mu(\frac{\partial U}{\partial x}+\frac{\partial V}{\partial y}+\frac{\partial W}{\partial z}\right]+\rho g_z$$

$$\frac{\partial}{\partial x}(\rho UT)+\frac{\partial}{\partial y}(\rho VT)+\frac{\partial}{\partial z}(\rho WT)=\frac{\partial}{\partial x}\left(\Gamma\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\Gamma\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(\Gamma\frac{\partial T}{\partial z}\right) \qquad (11)$$

Turbulent models:

This study discussed four turbulent models (standard K-Ɛ, RNG K-Ɛ, realizable K-Ɛ, and SST K-ω), since most researchers used those four turbulent models. Table(4) presents some strengths and weaknesses of the four turbulent models used in the present study,[14]. The equations for the four turbulent models are,[14] :

a- Standard K-Ɛ :

$$\frac{\partial(\rho k)}{\partial t}+div(\rho k\mathbf{U})=div\left[\frac{\mu_t}{\sigma_k}grad\ k\right]+2\mu_t E_{ij}.E_{ij}-\rho\varepsilon \qquad (12)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+div(\partial\varepsilon U)=div[\frac{\mu_t}{\sigma_\varepsilon}grad\ \varepsilon]+C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}.E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \qquad (13)$$

the model constants $C_{1\varepsilon}$ and $C_{2\varepsilon}$ of 1.44 and 1.92 are used.

b- RNG K-Ɛ model:

$$\rho U_i\frac{\partial k}{\partial x_i}=\mu_t S^2+\frac{\partial}{\partial x_i}[\alpha_k\mu_{eff}\frac{\partial k}{\partial x_i}]-\rho\varepsilon \qquad (14)$$

$$\rho U_i\frac{\partial\varepsilon}{\partial x_i}=C_{1\varepsilon}(\frac{\varepsilon}{k})\mu_t S^2+\frac{\partial}{\partial x_i}[\alpha_k\mu_{eff}\frac{\partial k}{\partial x_i}]-C_{2\varepsilon}\rho(\frac{\varepsilon^2}{k})-R \qquad (15)$$

The model constants are the following values: $C_{1\varepsilon}$=1.42 and $C_{2\varepsilon}$=1.68

c-Realizable K-Ɛ:

$$\frac{\partial}{\partial t}(\rho K)+\frac{\partial}{\partial t}(\rho K u_j)=\frac{\partial}{\partial x_j}[(\mu+\frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}]+P_k+P_b-\rho\varepsilon-Y_M+S_k \quad (16)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_j}(\rho\varepsilon u_j)=\frac{\partial}{\partial x_j}[(\mu+\frac{\mu_t}{\sigma_\varepsilon})\frac{\partial\varepsilon}{\partial x_j}]+\rho C_1 S_\varepsilon-\rho C_2\frac{\varepsilon^2}{K+\sqrt{\upsilon\varepsilon}}+C_{1\varepsilon}\frac{\varepsilon}{k}C_{3\varepsilon}P_b+S_\varepsilon \quad (17)$$

The model constants are: $C1\varepsilon=1.44$, $C_2=1.9$, $\sigma_k=1.0$ and $\sigma\varepsilon=1.2$

d- SST K-ω model:

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=P_k-\beta^* k\omega+\frac{\partial}{\partial x_j}\left[(\nu+\sigma_k\nu_T)\frac{\partial k}{\partial x_j}\right] \quad (18)$$

$$\frac{\partial\omega}{\partial t}+U_j\frac{\partial\omega}{\partial x_j}=\alpha S^2-\beta\omega^2+\frac{\partial}{\partial x_j}\left[(\nu+\sigma_\omega\nu_T)\frac{\partial\omega}{\partial x_j}\right]+2(1-F_1)\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial\omega}{\partial x_i} \quad (19)$$

The model constants are: $\beta=0.0828$, $\rho_{\omega1}=0.5$, $\rho_{\omega2}=0.856$

Numerical solution and boundary conditions:

The GAMBIT 2.4.6 software was used to generate the model and mesh generate of case study is depending on many testing meshes as shown in Fig.(8). Analyses by using FLUENT6.3.26 code software until the residual error for solved equations arrived to (10^{-3}) and (10^{-6}) for energy equation as shown in Fig.(9). Second-order upwind scheme using for the convection terms. PRESTO (PREssure Staggering Option) scheme is used for the pressure. For the pressure-velocity coupling the SIMPLEC scheme is used. When working with unstructured meshes, a high-order scheme is preferred for the discretization of convection terms to minimize the discretization errors. The boundary conditions and assumptions are given in table(5).

Error calculation for validated cases

The simulated results were compared with the experimental results by calculate the absolute overall error by using the following equation:,[15].

$$E=\sum_{i=1}^{n}\frac{\left|X^i_{CFD}-X^i_{EXP}\right|}{X^i_{EXP}}*100 \quad (20)$$

Where X can be either air velocity or air temperature. $\left|X^i_{CFD}-X^i_{EXP}\right|$ is the absolute difference between simulated values and experimental measurement values for variable X and (n) number of measurements.

Finally it was necessary to validate Flunet software with another experimental study,[16]. The validation was done by comparing the RNG K-Ɛ model results with experimental data obtained in an office room with a wall-supply displacement ventilation system. The comparison depends on the vertical air temperature which measured in five points at vertical pole (pole1 as shown in Ref[16]) as shown in Fig.(18).

The comparison gives a good agreement between the experimental data, Ref.[16], and air temperature simulated by used the RNG K-Ɛ turbulent model. The average error between the experimental and numerical values calculated by used Eqn.(20) is 4.6 %

RESULTS AND DISCUSSION

To validate the CFD model, the simulated results were compared with experimental results.

Fig.(10) shows results for air temperature distribution obtained numerically from the four turbulent models (standard K-Ɛ, RNG, realizable K-Ɛ, and SST K-ω) and from the experimental measuring devices fixed at the two poles. There is some deviation between the predicted air temperatures and experimental results, which clearly the difference in the air temperature curves for the two poles . Air temperature under 1m height was increased more than top locations of the pole since most of the heat source (person & computer) was released under (1m). Each turbulent model have special method to predicted of air temperature with height since air temperature was generally increased along height due to the buoyancy effect of heat sources.

Figs.(11) shows results for air velocity distribution with height obtained from the four turbulent models from the measuring devices fixed at the two poles. Air velocity in occupied zone was for the most part under 0.05m/s, with small velocity variations (<0.05m) along the height, that shows the four turbulent models follow the similar method of treatment low air velocity. The large variations at Pole-1 and maximum velocity happened at height 0.16m on Pole-1 ,that's due to short distance from supply cold air which supplied fresh air at 0.25m/s. The velocity increased near the floor (0-0.16m/s) due to effects of boundary layer and after that its decrease to reach in most space lower than 0.05m/s, that's agree with thermal comfort zone.

Figs.(12) shows air temperature distribution contours for vertical plane at Z=0.9m and horizontal plane at y=0.2 for the four turbulent models. Air temperature distribution prediction near of heat source (person and the walls), differ for each turbulent model since the Sk-ε model Rk-ε model gives poor prediction near of hot walls, that explains the variable analyses method for each model near of wall. Another phenomenon about cold air throw from the exhaust grille, therefore exhaust grille location is very important to reach a suitable air ventilation.

Figs.(13) shows air velocity vectors for vertical plane at Z=0.9m and horizontal plane at y=0.2 for the four turbulent model. The air moves horizontally over the floor due to momentum from the supply outlet air and suction from thermal plumes. It then passes vertically to a high level in the room where it is exhausted from exhaust grille. Vertical air movement between layers is caused by stronger convection forces associated with heat sources (person and computer). Anticlockwise air swirl in zone limited between diffuser and person since the air impact with body surface wall and back flow, that causes increase air temperature at this zone as result to heat transfer by convection between air and person, this phenomena is more clearly in RNG model and SST model due to a good heat transfer analyses near wall for these two models and explain the variable analyses for each model to predict the air temperature at swirl zones.

Fig.(14) shows predicted air temperature distribution with length of (3m) for the four turbulent models (standard K-Ɛ, RNG K-Ɛ, realizable K-Ɛ, and SST K-ω) compared with design setup temperature for office room (T_{sp} =25 ^{o}C) at breathing zone line shown in **Fig.(15).** The RNG K-Ɛ turbulent model curve is nearest from design setup temperature than the other turbulent models (standard K-Ɛ, realizable K-Ɛ, and SST K-ω). The curve air temperature prediction for RNG K-Ɛ and SST K-ω turbulent models over hot zone (over person and computer between 1-1.5m) were more logical from other models since the hot air was localized over heat source and that two turbulent models are improved predictions for wall heat and mass transfer.

Table(6) shows the average error between simulated results for the four turbulent models and experimental data (10 reading recorder from the two poles) for air temperature. The minimum error in air temperature occurs at (RNG model 4.58%) and maximum error occurs in (RKE model 8.75%).

Fig.(16) shows results for air temperature distribution with height for experimental data and predicted by using RNG K-Ɛ turbulent model for two poles. Over 0.4m height (over diffuser level) the air temperature in pole-2 is less than the air temperature in pole-1 due to nearest to hot south wall.

Fig.(17) shows results for air velocity distribution with height for experimental data and predicted by using RNG K-Ɛ turbulent model. The maximum air velocity occur at 0.2m height in pole-1 since the pole-1 nearest to air supply unit, while notes low air velocity in pole-2 under 1m height due to location of pole-2 behind the thermal manikin (person) and the air moved out the supply air unit impact by thermal manikin before reach to pole-2, after that (over 0.4m height) the air distribution have the same predicted at low velocity (less than 0.05m/s).

CONCLUSIONS

The following conclusions have been listed from the results of present study:

1- Assessment of four turbulent models in displacement ventilation system obtained that the RNG.k-Ɛ turbulent model gives a minimum average error (4.58% at air temperature distribution), The RNG.k-Ɛ turbulent model is more accurate than others turbulent models (standard K-Ɛ, realizable K-Ɛ, and SST K-ω) for isothermal tested room under displacement ventilation system.

2- Each model of four turbulent models has specialty method to predicted air temperature and velocity distribution.

3- No differences in airflow patterns for each model in displacement ventilation system. The four turbulent models gives similarity in prediction air velocity distribution for different spaces. Maximum air velocity and minimum air temperature occurred near the floor of occupied zone in displacement ventilation system due to flow of the cold air from supply air devices on the floor.

4- RNG.k-Ɛ turbulent model and SST K-ω have more accurate of other models (standard K-Ɛ, realizable K-Ɛ) to predicted air temperature distribution near to heat sources and walls locations.

Table (1) Room configuration

Item	**Location m**			**Size m**			**Heat W**
	x	**y**	**z**	Δx	Δ**y**	Δ**z**	
Room	0	0	0	3	3	1.75	-
Diffuser	0	0.1	0.675	0.1	0.15	0.4	-
Exit air	3	2.3	0.75	0	0.2	0.2	-
Person	1.3	0	0.8	0.4	1.1	0.35	75
Compute	1.35	0.75	0.3	0.37	0.3	0.35	45
Lump	1.5	2.5	0	0.1	0.1	0.25	100
Table	0.75	0.65	0.1	1.25	0.1	0.5	0

Table(2): Values of air flowrate (Q_s), supply air temperature (T_s), and measured walls surface temperature

Q_s l/s	T_s ^oC	ACH	Measured wall surface temperature					
			south ^oC	north ^oC	east ^oC	west ^oC	ceil ^oC	floor ^oC
20.15	18	4.6	27.2	27.4	27.8	26.8	29.1	22.6

Table(3) Experimental readings data

height m	pole one	pole two
	temperature ^oC	temperature ^oC
0.4	27.9	27
0.8	30.6	27.5
1	31.5	27.9
1.4	33.6	28.2
1.8	32.1	28.4

Table(4) Strengths and weaknesses for the four turbulent models

Model	Strengths	Weaknesses
Sk-Ɛ	Robust, economical, reasonably accurate; long accumulated performance data.	Mediocre results for complex flows with severe pressure gradients, strong streamline curvature, swirl and rotation.
RNG k-Ɛ	Good for moderately complex behavior like jet impingement, separating flows, swirling flows, and secondary flows.	Subjected to limitations due to isotropic eddy viscosity assumption. Same problem with round jets as standard k-ε.
Realizable k-Ɛ	Offers largely the same benefits as Sk-Ɛ but also resolves the round-jet anomaly.	Subjected to limitations due to isotropic eddy viscosity assumption.
SST Model	Physically most complete model (transport, and an isotropy of turbulent stresses are all accounted for).	Requires more(CPU) ; tightly coupled momentum and turbulence equations.

Table(5) Boundary Conditions

Assumptions	Velocities	Temperature
Inlet	$U = U_s$, $V=0$, $W = 0$	$T=T_s$
Outlet	$U=U_s \frac{A_{in}}{A_{out}}$	$\frac{\partial T}{\partial X} = 0$
No slip on walls	$U=0$, $V=0$, $W=0$	$T=T_{wall}$

Table(6) Error value of air temperature between the simulated and the experimental results for 10 readings

	Erorr %			
Turbulant model	SK-Ɛ model	RNG model	RK-Ɛ model	SST model
Air temperature ^oC	6.55	4.58	8.75	8.61

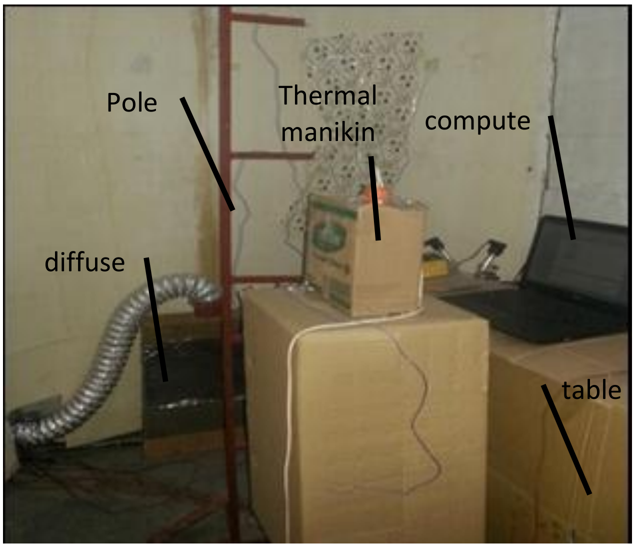


Fig.(1) Photograph of the experimental isothermal tested room

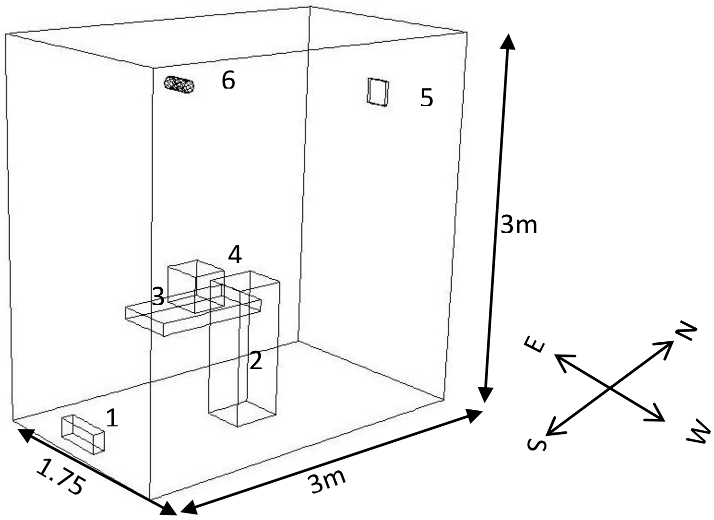
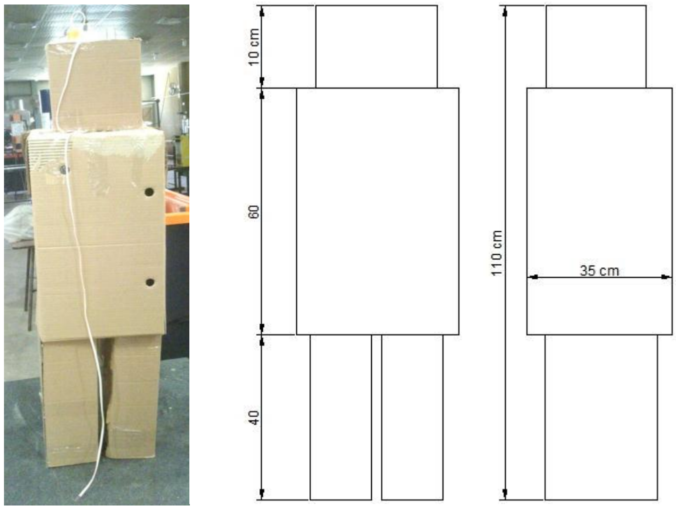


Fig.(2) Schematic diagram for tested room adopting DV system
1.air supply unit 2.person 3.tabel 4.computer 5.exhaust grille. 6-lump



a- Photograph b- Front view c- Side view

Fig.(3) Thermal manikin used in experimental work

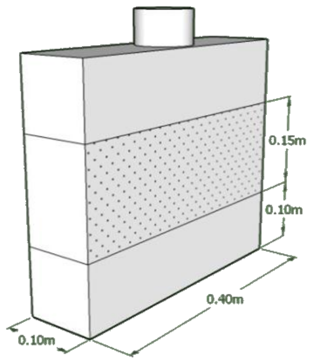
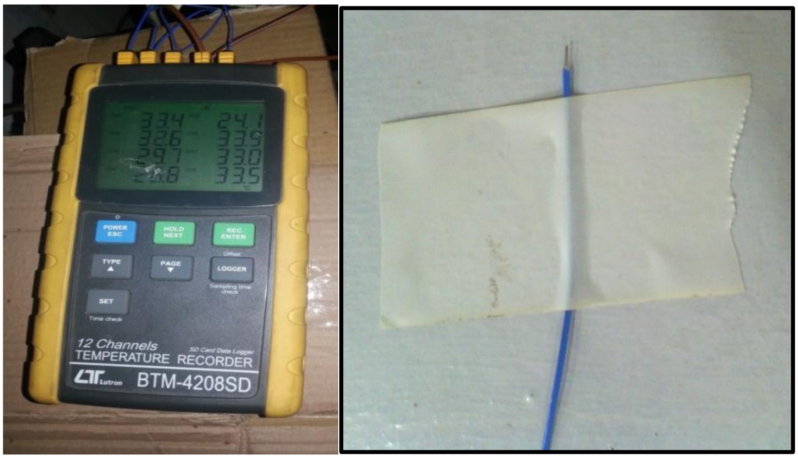
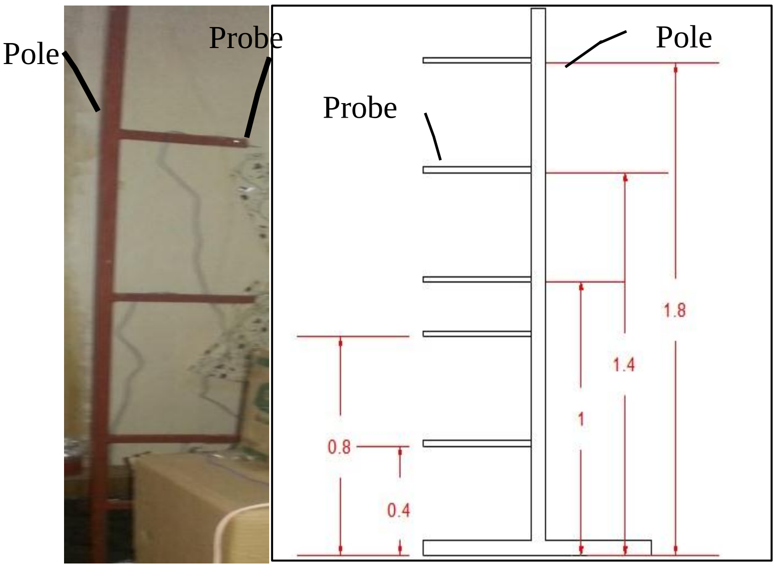


Fig.(4) The rectangular diffuser (DF1C series type) used in tested room



a- Temperature recorder b- Thermocouple attached to wall surfaces

Fig.(5) Surface temperature measurement system



a- Photograph of pole b- Schematic diagram of pole

Fig.(6) Arrangement of measured devices on pole

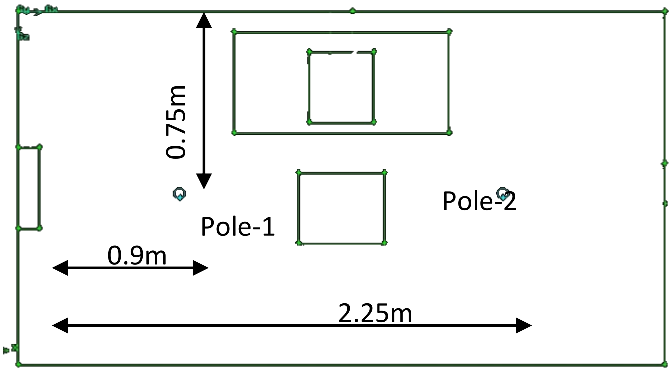


Fig.(7) Locations of the measurement devices

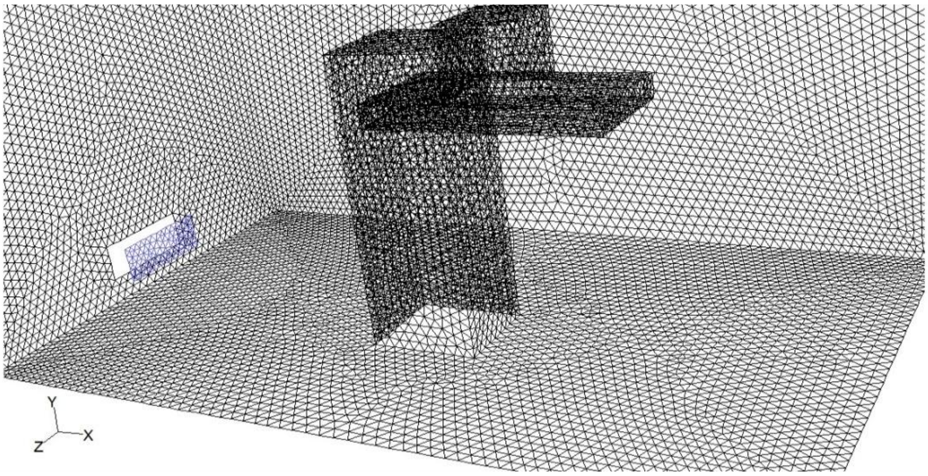


Fig.(8) Part from meshed model

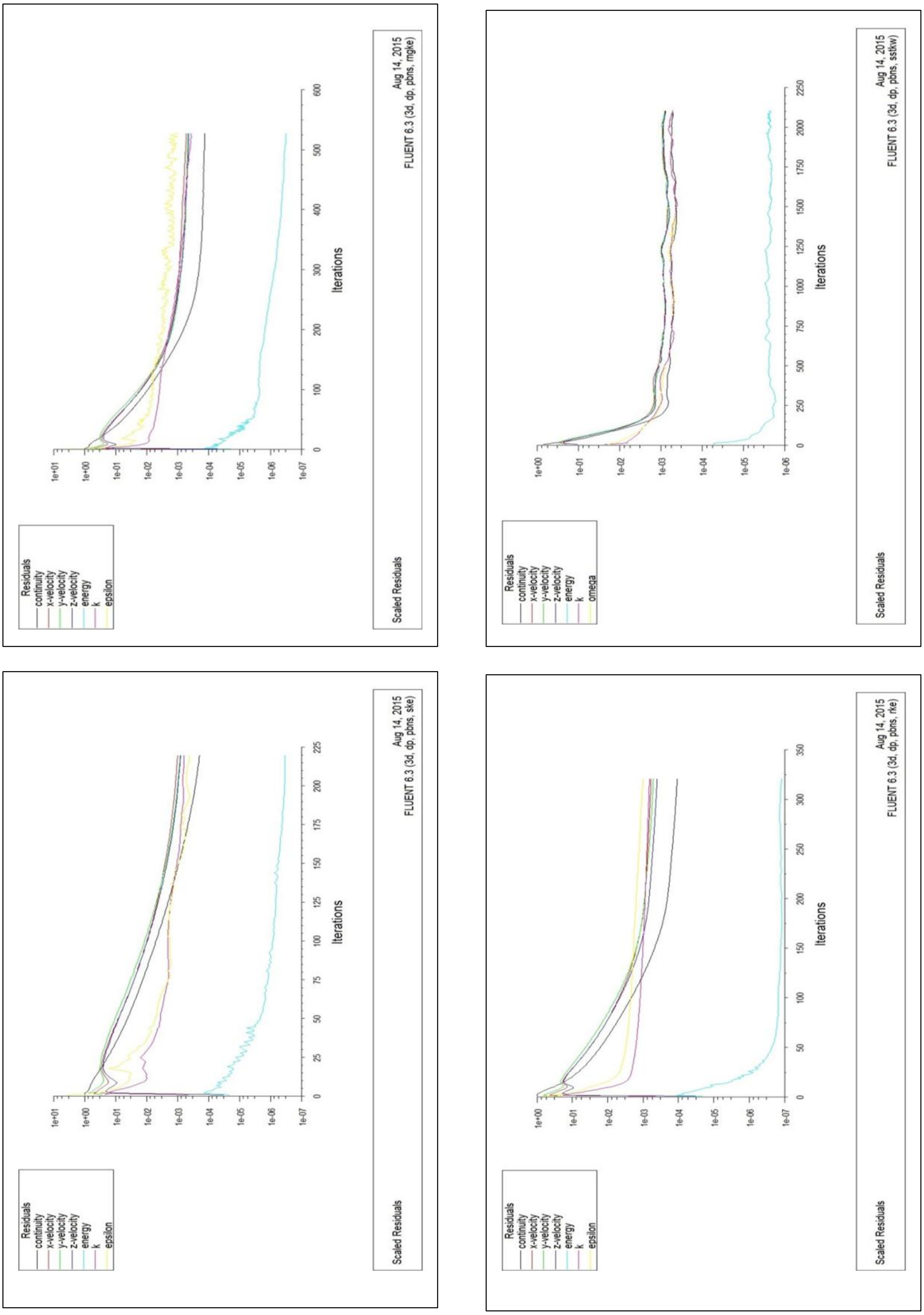
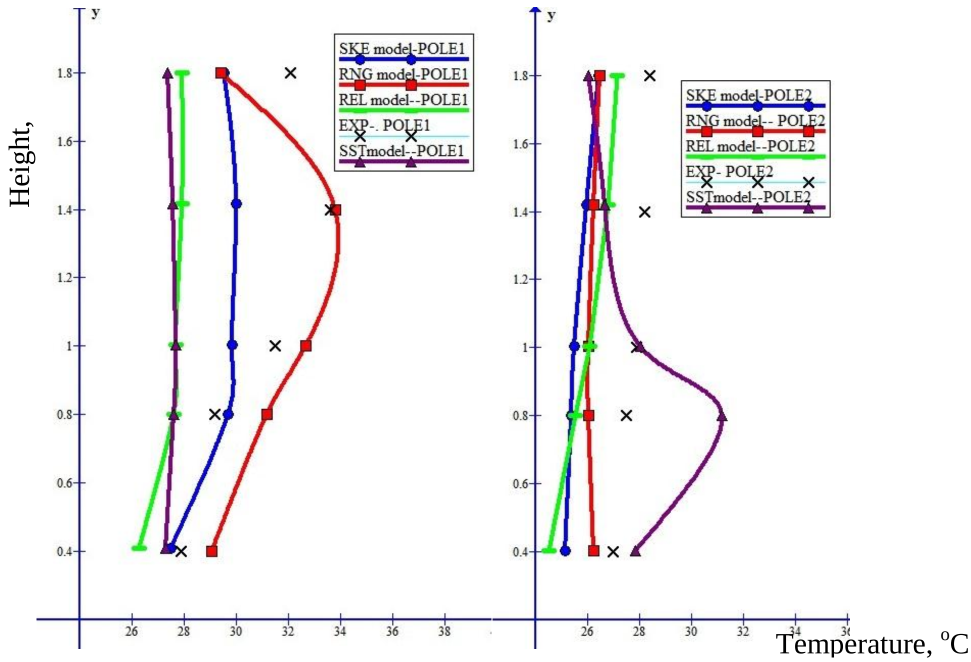
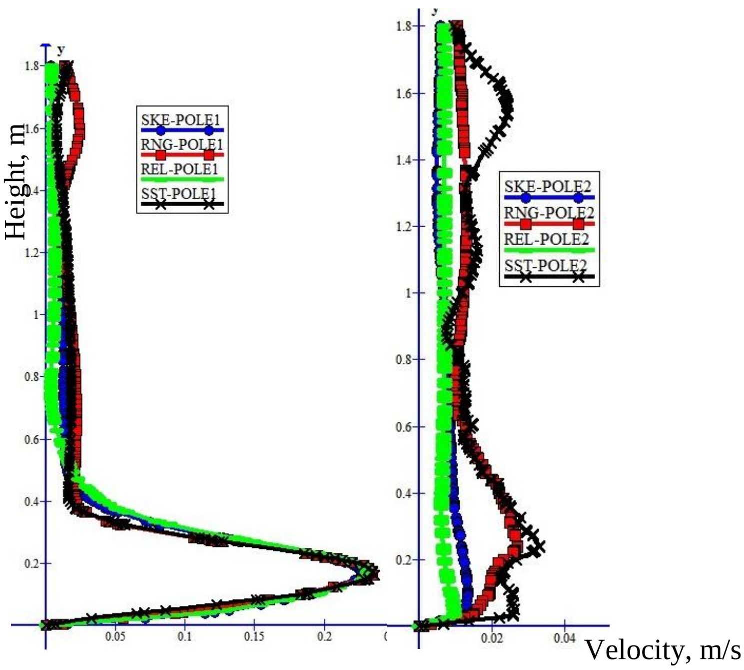


Fig.(9) Residual error monitor for the four turbulent models.



a-pole-1 b-pole-2

Fig.(10) Air temperature distribution with height for isothermal tested room



a-pole-1 b- pole-2

Fig.(11) Air velocity distribution with height

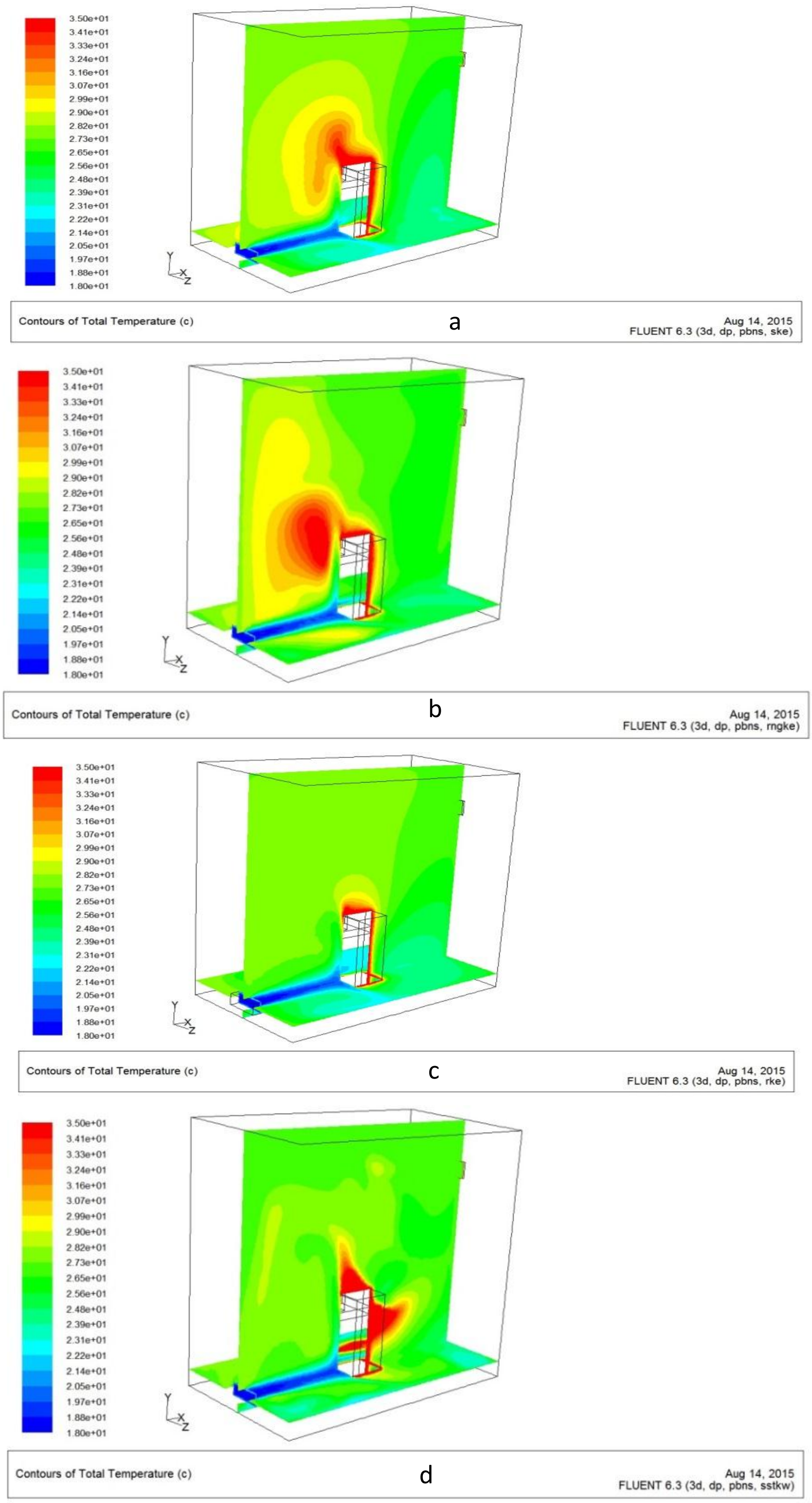


Fig.(12) Temperature distribution contours for vertical plane at Z=0.9m and horizontal plane at y=0.2
a- SK-Ɛ models b- RNG K-Ɛ models c- RK-Ɛ models d- SST K-ω models

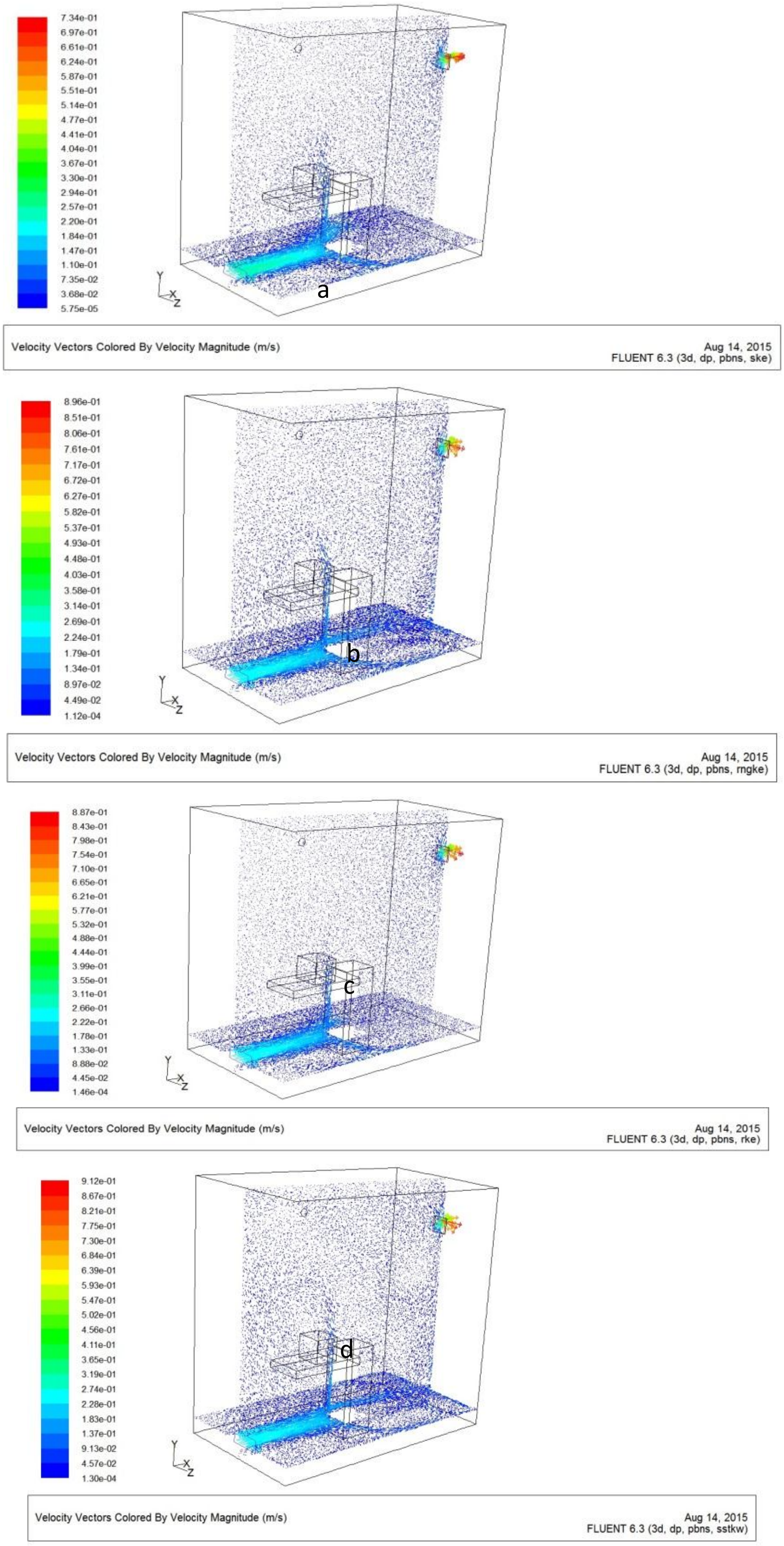


Fig.(13) Air velocity vectors for vertical plane at Z=0.9m and horizontal plane at y=0.2
a- SK-Ɛ models b- RNG K-Ɛ models c- RK-Ɛ models d- SST K-ω models

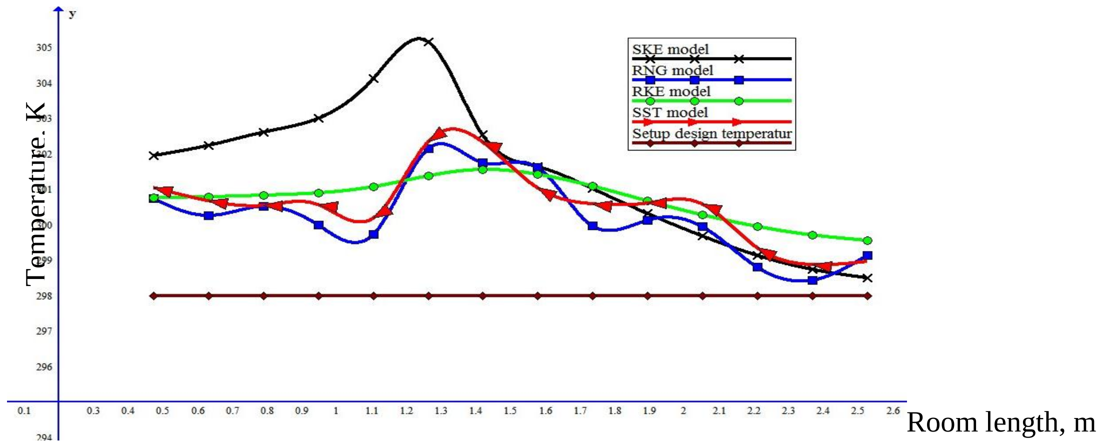


Fig.(14) Air temperature distribution with length(3m) of tested room at breathing zone line

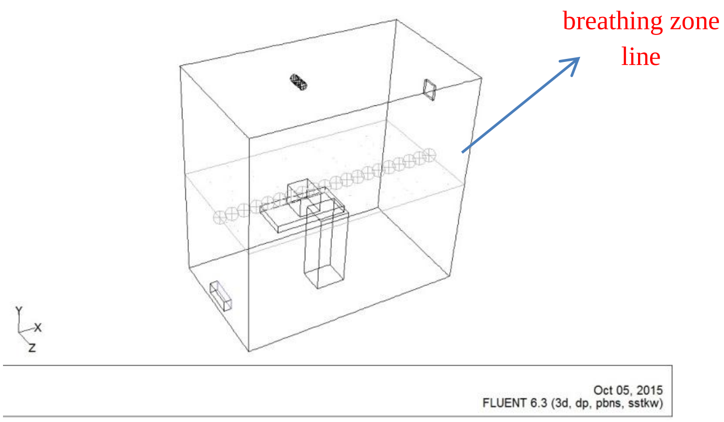


Fig.(15) Schematic diagram for tested room adopting DV system, shows the breathing zone line between points (0,1.5,1.75) and (4,1.5,1.75)

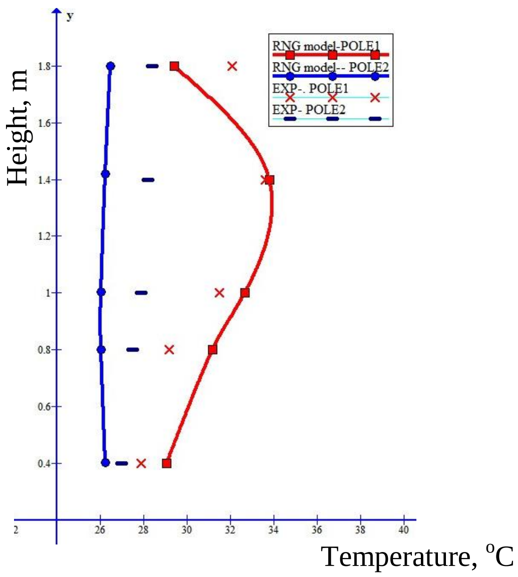


Fig.(16) Air temperature distribution with height for experimental data and predicted by RNG K-Ɛ turbulent model for two poles

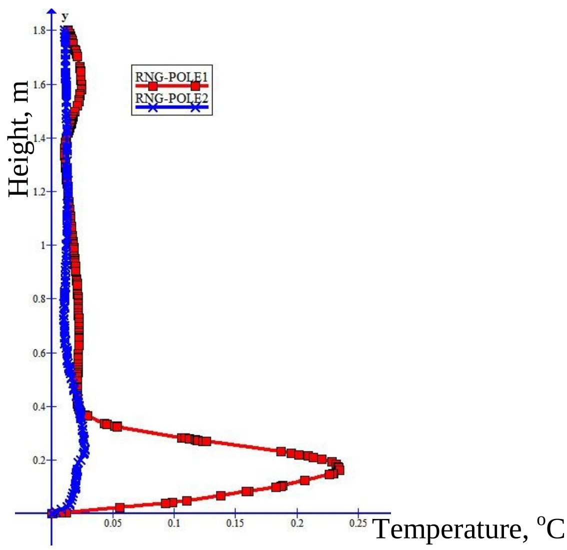


Fig.(17) predicted air velocity distribution with height adopting RNG-KƐ turbulent model for two poles

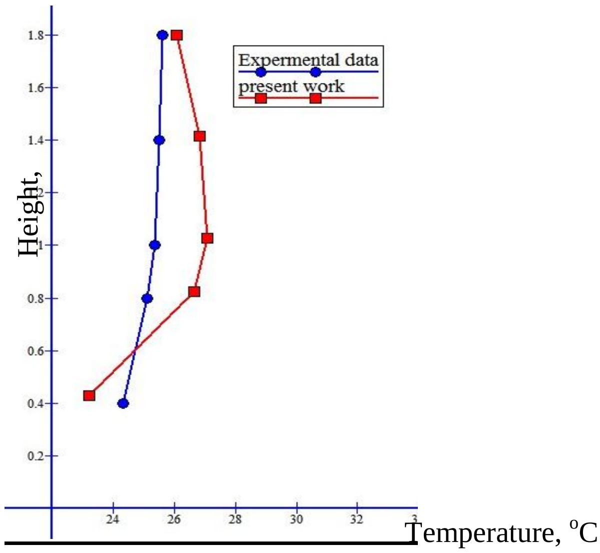


Fig.(18) Validate the RNG K-Ɛ turbulent model with another experimental study.

References:

1- Loomans, M. “The Measurement and Simulation of Indoor Air Flow”. PhD. Thesis. The Netherlands: University of Eindhoven, 1998
2- Hakon Skistad ,”displacement ventilation and air curtain zoning”, M.Sc.2007.
3- Hyder Mohammed, “A theoretical study of a cold air distribution system with different supply patterns”,Msc thesis, 2013.
4- Chen Q. 1995. “Comparison of different k-ε models for indoor air flow computations,” Numerical Heat Transfer. 28 (B): 353-369.
5- Bunn, R.. “Finding the right mix. building Services Journal”, 1996
6- Dickson, D.” A testing time for chilled ceiling”. Building Services Journal 16,1994.
7- Y. Ren, D. Li, Y. Zhang, “Numerical Simulation of Thermal Comfort Degree in Radiant Floor Cooling Room” Building Simulation,2007.
8- Lau, J. and Chen, Q.. "Floor-supply displacement ventilation for workshops," Building and environment, 2007
9- Chen, Q., Glicksman, L.R., Yuan, X., Hu, S., & Yang, X., “Final report for ASHRAE RP-949: Performance evaluation and development of design guidelines for displacement ventilation”. Cambridge, MA: Department of Architecture, Massachusetts Institute of Technology, 1999.
10- ASHRAE. Standard 55-2004,”Thermal environmental conditions for human occupancy”. Atlanta, GA: American Society of Heating, Refrigerating and Air Conditioning Engineers,2004.
11- Chen, Q. & Glicksman, L. “System performance evaluation and design guidelines for displacement ventilation”. Atlanta, GA: American Society of Heating, Refrigerating, and Air-conditioning Engineers, Inc,2003.
12- Iraqi cooling code,2012.
13- H. B. Awbi, “Ventilation of Buildings”, Spon Press, second edition, 2003.
14- HK Versteeg and W Malalasekera,”An introduction to computational fluid dynamics the finite volume method”,1996.
15- Johnson Lim Soon Chong, Adnan Husain & Tee Boon Tuan, "Simulation of airflow in lecture rooms", proceedings of the AEESAP international conference, 2005.
16- Yue Xu, “Modeling and evaluation of personal displacement ventilation system for improving indoor air quality”,Msc thesis, University of Miami,2007.