Eng. & Tech. Journal ,Vol . 30, No. 3 , 2012

Experimental and Numeriacal Study of Moisture Movement Inside an Air Conditioned Space

Dr. Wahid S. Mohammad
Machines & Equipments Engineering Department, University of Technology/ Baghdad
Email: wahid-1983@yahoo.com
Dr. Omar M. Mohamme
Institute of Technology/Baghdad

Received on: 12/6/2011 & Accepted on: 1/12/2011

ABSTRACT

This research is concerned with a computational study to simulate a turbulent three –dimensional buoyancy recirculation flow in an air-conditioned space. The study includes the simulation of heat and moisture generated from an internal source inside the space. An experimental facility to study temperature and moisture distribution in an air-conditioned space has also been designed, constructed and tested in a laboratory scale. A numerical procedure was carried out to solve the elliptic partial differential equations that govern the flow, heat and mass transfer in a finite –volume form. The finite -volume approach was applied to solve these equations using the upwind –differencing scheme. The SIMPLE iterative procedure [1] for solving the algebraic equations is employed in the present study. The proposed method is the line by line technique uses the Tri-Diagonal Matrix Algorithm (TDMA) as its basic unit. A modified version of a three-dimensional elliptic computer code was used to simulate heat and moisture transfer generated from the internal source inside the space. A study of the flow, heat and mass transfer in air-conditioned space are used as test cases to justify the performance of the computational procedure. The temperature and moisture distribution were compared with predictions of previous researchers. The data from the experimental model was also used to verify the computational procedure predictions.

Keywords: CFD, Heat and Mass transfer, Thermal comfort.

دراسةعملية ونظرية عن حركة بخار الماء داخل حيز مكيف

الخلاصة

يهتم هذا البحث بالدراسة الحاسوبية لمحاكاة الجريانات الدورانية المضطربة والطفوية الثلاثية الابعاد في حيز مكيف. تتضمن الدراسة محاكاة الحرارة والرطوبة المتولدة من مصدر داخلي داخل الحيز. لقد تم دراسة توزيع الحرارة والرطوبة في حيز مكيف بشكل تجريبي من خلال تصميم وانشاء واختبار لنموذج داخل مختبر. لقد ابتدأ العمل العددي بحل المعادلات التفاضلية الجزئية الاهليليجية المتمثلة بحفظ الكتلة، الزخم، الطاقة، الطاقة المضطربة ومعدل ضياعها باستخام الحجوم المحددة (Finite –volume). تم تطبيق الحجوم المحددة لحل هذه المعادلات باستخدام (Upwind differencing scheme). لقد تم استخدام نسخة مطورة من برنامج حاسوبي اهليلجي ثلاثي الابعاد (Three –dimensional elliptic computer code) لحساب الحرارة والرطوبة المنتقلة المتولدة من مصدر داخلي داخل حيز .لقد تم

https://doi.org/10.30684/etj.30.3.6
2412-0758/University of Technology-Iraq, Baghdad, Iraq

دراسة الجريان وانتقال الحرارة والرطوبة في حيز مكيف كحـــالات اختباريـــة لتقيـــيم اداء البرنـــامج الحاسوبي الحالي. ان توزيع الحرارة والرطوبة قد قورنت مع النتائج النظريـــة لبـــاحثين ســـابقين. لقـــد استخدمت النتائج المستحصلة من العمل التجريبي لغرض تقييم اداء البرنامج.

INTRODUCTION

The main aim of conditioning the interior environments of buildings is to provide a comfortable and healthy indoor environment. Increasingly, concern is being expressed over the quality of the indoor environment, where by failure to provide satisfactory thermal conditions has resulted in many reports of discomfort and ill health. As such, there has been a constant need to study the thermal conditions of these indoor environments. Numerous studies have been conducted to explore the conditions of residential and commercial buildings, in which occupants spend large amount of their time everyday [2]. The concepts of both thermal comfort and computational fluid dynamics have been widely researched upon. Some studies have been published on the fundamentals of thermal comfort and case studies have been conducted for buildings in the temperate and tropical climate. Other studies have been conducted to develop models, which could enhance the study of thermal comfort in the indoor environments. Research works on computational fluid dynamics result in innumerable articles on the application of CFD, as well as its usage in designs. CFD was used to model air movement and thermal comfort in mechanically and naturally ventilated offices [3,4]. Cheong et.al.[5] presented a CFD thermal comfort study of an air-conditioned lecture theatre. The CFD tool was used to model and simulate the indoors thermal conditions provided by the air-conditioning system of the lecture theatre. Rohles and Nevins [6] studied the nature of thermal comfort for sedentary men. Negrao et.al.[7] described a simplified computational model to predict indoor air temperature distribution. Pitram et. al. [8] introduced a new advanced active tool in environmental education, directed to indoor – environment quality, that permits the predication and visualization of air movement, air temperature and contaminant (such as tobacco smoke) distribution in rooms. **Abid** [9] investigated the differences in thermal comfort conditions between the northern Iraqi and middle Iraqi groups.

EXPERIMENTAL WORK

An experimental heat and mass transfer facility of small-scale room has been designed and constructed to investigate the characteristics of temperature and moisture distribution in rooms. The experimental model which is a single room with interior dimensions of ($80\times 60 \times 60$) cm. The model can best be described as an isolated box with well-insulated walls. The model was connected to an air-conditioning device that supplies the air to the model at a given flow rate, temperature and humidity (Figure 1). The walls of the experimental model are constructed from insulated perspex sheets of 4mm thickness. The internal moisture source has exterior dimensions of ($20 \times 10\times 10$) cm. It is constructed from insulated perspex sheets of 6mm thickness and contains an electrical immersion heater (steel structure, 100 Watt, 220volt) to produce water vapor. A sensor (type K, (Nickel–Corm/ Nickel-Aluminum)) was used in the source to regulate water temperature, and then the vapor generation rate. Heater and sensor are connected to (0-400°C) digital thermal control and temperature selector switch. Temperature and humidity were measured by temperature –humidity sensor which is connected to humidity / temperature – meter (type HT – 3005 HA) with specifications: Humidity range:10% to 95% RH; Temperature range: 0 to 50 °C / 32 to 122°F; Humidity accuracy: 1% RH; Temperature accuracy: ± 0.8°C. Seven sensors are used in the experiment to measure temperature and humidity at various points. All of them were connected to the multiplexer of seven inputs and one output to the humidity /Temperature meter. Multiplexer with 220 volt / DC current and relay system manufactured with a timer (1-10) second between every one sensor and the meter is fixed to read each sensor at 10 seconds, interval, respectively. The sensors are placed inside the experimental model as seen in Figures (2 and 3). The airflow rate was measured using a digital flow meter.

MATHEMATICAL FORMULATION AND NUMERICAL SOLUTION

The essence of CFD's is to numerically model physical processes occurring within a fluid by the solution of a set of non-linear partial differential equations to express the fundamental physical laws that govern the conservation of mass, momentum, energy and moisture content. For room air motion the driving forces are pressure differences, which are caused by wind, thermal buoyancy, mechanical ventilation system or combinations of these. The characteristics of indoor airflow are low velocity and high turbulence intensity [10]. When applying CFD to the indoor air motion field, the Navier-Stokes equations are derived by applying the principles of conservation of mass and momentum to a control volume of fluid [11].

Conservation of Mass:

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

Conservation of Momentum:

$$\frac{\partial (ruu)}{\partial x}+\frac{\partial (rvu)}{\partial y}+\frac{\partial (rwu)}{\partial z}= -\frac{\partial p}{\partial x}+m[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}] \qquad \dots (2)$$

$$\frac{\partial(ruv)}{\partial x}+\frac{\partial(rvv)}{\partial y}+\frac{\partial(rwv)}{\partial z}= -\frac{\partial p}{\partial y}+m[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}] \quad \ldots (3)$$

$$\frac{\partial(ruw)}{\partial x}+\frac{\partial(rvw)}{\partial y}+\frac{\partial(rww)}{\partial z}= -\frac{\partial p}{\partial z}+m[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}] \quad \ldots (4)$$

It is more compact to write equation (2) to (4) in Cartesian tensor notation as [12].

$$\frac{\partial(ru_j u_i)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_i}[(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i})]+S_{bj} \quad \ldots (5)$$

Where the suffixes i and j represent the three-co-ordinate direction .S_{bj}, are buoyancy source or sink terms.

The conservation equations governing the dispersion of certain scalar properties of the flow, such as temperature, humidity, turbulence kinetic energy K or turbulence dissipation rate e, have a similarity of form. They can therefore be represented by a single differential equation for some general scalar property (Φ) as follows:

$$\frac{\partial}{\partial x_j}(ru_j\Phi)=\frac{\partial J_{\Phi.j}}{\partial x_j}+S_\Phi \quad \ldots (6)$$

The scalar property flux $J_{\Phi,j}$ in equation (6) can be related to gradient of velocity components and scalar properties respectively. In constant-property flows this constitutive relation is as follows [12]:

$$J_{\Phi,j}=\frac{m}{s_\Phi}(\frac{\partial\Phi}{\partial x_j}) \quad \ldots (7)$$

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

$$\frac{\partial}{\partial x}(rfu)+\frac{\partial}{\partial y}(rfv)+r\frac{\partial}{\partial z}(rfw)= \frac{\partial}{\partial x}(\Gamma_f\frac{\partial f}{\partial x})+\frac{\partial}{\partial y}(\Gamma_f\frac{\partial f}{\partial y})+\frac{\partial}{\partial z}(\Gamma_f\frac{\partial f}{\partial z})+S_f \quad \ldots (8)$$

The expressions for the diffusion coefficient Γ_f, and source term, S_f, corresponding to each variable are given in table (1).

The effective viscosity, generation rate and buoyancy source in the K and e transport equation is given by [13].

$$G_k = m_t[2((\frac{\partial U}{\partial x})^2 + (\frac{\partial V}{\partial y})^2 + (\frac{\partial W}{\partial z})^2) + (\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x})^2 + (\frac{\partial U}{\partial z} + \frac{\partial W}{\partial x})^2 + (\frac{\partial V}{\partial z} + \frac{\partial W}{\partial y})^2] \quad \dots (9)$$

$$G_B = -bg\frac{m_t}{s_t}\frac{\partial q}{\partial Y} \quad \dots (10)$$

$$m_{eff} = \frac{C_D r.K^2}{e} + m_l \quad \dots (11)$$

The constants (C_1, C_2, C_D, s_K, s_e) which are assigned the values given in table (2)[14]. The above set of constants has been applied successfully to three-dimensional flows in buildings [14].

MOISTURE GENERATION

To simulate the cases that water evaporates from the bath, cooking, washing, personal activity etc., not only moisture transfer, but also heat transfer should be taken into account. Some methods have been developed for different research fields [15]. As shown in Figure (4), the principle can be considered mainly as a time-dependent process, which includes two parts:-

1- Saturated vapor across the surface to air will be changed into liquid state and latent heat will be transferred to air.

2-Vapor will be in a continuous gas phase and the simple heat-mass transfer analogy can be considered as valid.

Because it is very hard to study the transient value, a simple model is introduced in this research to express the average heat flux during a certain period, [16].

$$\left\{ \begin{array}{ll} q_1 = \alpha \dot{m} \gamma & (12) \\ q_2 = \frac{(1-\alpha)\times \dot{m}}{(\omega_i - \omega_\infty)} \times (h_i - h_\infty) & (13) \\ q'' = q_1 + q_2 & (14) \end{array} \right.$$

RESULTS AND DISCUSSION

The physical problem that is considered as the test case is the one studied numerically and experimental by Liu et. al.[16]. This problem is concerned with the numerical computation of three - dimensional, turbulent, generated heat and moisture from internal source in a rectangular space, into which buoyancy effects are introduced by heating the internal source in the centre of the base space. The main driving force is provided by an adjoining channel flow as shown in Figure (5).The fluid enters to the space from lower part of the left side through a channel with height (h). Recirculation flow occurs inside the space due to driving force at the entrance (force convection). The fluid leaves at the upper part of the right side through another channel, which has the same height (h), and the internal source in the center of base, the measuring plane is fixed in (Z1 = 300 and Z2 = 900) mm. Figure (5) , shows the dimensions of the space as (H = 1500,L=1220, W_d=1210, H_{in}= 210, H_{out}=1160, h=80, ls=236, Hs=180 and Ws=234) in millimeters. This case was used to verify the present computer code. A grid size of (39 x47 x 39 x) nodes was employed. The values for the velocity, temperature, moisture content and turbulent quantities at the inlet and all the solid boundaries are given in table (3).The computational results of the buoyant flow have been compared with predictions of Liu et.al. [16].The results are presented in Figures (6 and 7). These figures show a comparison between the predictions of the present code with that of Liu et.al.[16] for the steady-state temperature and moisture distribution. Both figures represent stations at (Z1=300 mm) and (Z2=900 mm), respectively. The comparison shows a relatively good agreement with the present predictions. The condensation was avoided in this case, because of the higher temperature of the water. The air temperature was increased to 15.5ºC as in Figure (6.b). Due to the buoyancy effect and the exhaust open near the ceiling, supply air moved upward and results in a higher a humidity a ratio to (10 $g_{w.v.}/kg_{d.a.}$) near the ceiling, compared to that at the floor of (7.5 $g_{w.v.}/kg_{d.a.}$). The discrepancies are less than (1.0 ºC) for Figure (6) and (1.5 $g_{w.v.}/kg_{d.a.}$) in Figure (7) which are relatively small.

Figures (8-11) show the velocity vector, temperature and moisture contour for this case predicted by the present code. Figure (8, a) shows the flow movement parallel to the floor and leaves toward outlet opening. In Figure (8.b) the flow drops before the heat – moisture source, due to the buoyancy effect and moves parallel to the floor then rises up along the heat-moisture source and mixes with recirculated flow after the heat –moisture source before leaving towards outlet opening.

It is clear that the flow reciculates above the heat- moisture source and boundary surface forming a recirculating core in the room centre above the source which can be seen in Figure (9,a) at a plane just above the source . There is a small recirculation zone near the floor due to the pressure difference in this zone. Figure (9.b) shows that the inlet flow forming a large recicrulating core above the heat – moisture source and another small vortex to the right of that zone.

The temperature and humidity contour are plotted in figures (10 and 11) respectively. It can be seen from these figures that the humidity diffused more than temperature .This is due to the buoyancy effect that forced the supply air to move up ward to ward the ceiling carrying the water vapor from the source pan with it. This phenomenon has also been noticed by Liu et al [16].Unfortunately there was no velocity vector, temperature and moisture contour for Liu et. al.[16] to compare with.

COMPARISON WITH EXPERIMENTAL WORK

Another verification case to the present computer code was the three dimensional flows of the present experimental model. The geometrical arrangements that have been examined are the same of experimental work. The air-conditioned room was chosen to be a single room with length (L=80cm), height (H=60cm) and width (W_d=60cm) as shown in Figure (5). It contain a heat moisture source with dimensions (Ls=20cm, Hs=10cm, Ws=10cm) located in the floor at distance (0.5L) from left hand wall. The room is ventilated with a side wall slot of (length=2.4cm, height 3cm) located at a distance (Hin=2cm) from the floor. The outlet was chosen to be in the opposite wall at a distance (Hout =55cm) from the floor level. The upwind differencing scheme was used in the calculation with a (19×19×19) grid of non uniform distribution. With expansion factor = (1.01) for x, y and z direction. The comparisons between the results obtained with the present elliptic code and experimental work are shown in Figures (12-16).

Figure (12) shows a comparison between present predictions and experimental data obtained for 2 ACH. It can be seen that the agreement between the present prediction and the experimental data for temperature values is quite reasonable for the two positions (1, 2) see Figure (2). In position (1) the temperature above the source is larger than others around the source due to buoyancy effect. The temperature of position (2) above the source is lower than position (1) because it is higher than position (1). Temperature around source for position (1) and (2) are in little difference because of the small buoyancy effect in this zone.

Figure (13) shows a comparison between present predictions and experimental results for moisture content. A reasonable agreement between present prediction and experimental data can be seen for the two positions (1, 2). In position (1), the moisture content above the source is larger than others around it and the middle one is more than others due to moisture diffusion. The moisture content in position (2) above the source is lower than position (1) because it is higher than position (1).Moisture content around the source for positions (1, 2) are differs because of the buoyancy effect and diffusion of moisture content.

Figure (14) shows the velocity vectors representing the air flow distribution inside the space. It shows the flow moves parallel to the floor deflected upward by the source wall. Because there is no difference in temperature between inlet and source walls except the top of the source which has a higher temperature and the flow moves upwards due to the effect of buoyancy. A recirculation zones above the source at left and right appears and then mixing with the flow above the source then moves upward to the outlet.

Figure (15) shows the distribution of the temperature above the source due to the buoyancy effect at the middle height of the model. The diffusion of temperature is higher because of the heat transfer from water vapor to air.

The diffusion of moisture content in Figure (16) is clear from the top surface of the source because of the other surface are the same moisture content of inlet. It diffuses upwards because of buoyancy effect. It is evidence that moisture content diffuses more than temperature because the source works as a humidifier. The diffusion of moisture content in the 3-dimension is shown in Figure (17).

$$K_{in} = 0.003\ (W_{in})^2 \qquad \ldots(15)$$

$$\varepsilon_{in} = \frac{C_m\, K_{in}^{3/2}}{0.001325 \times h} \qquad \ldots.(16)$$

Where h: is the grille height.

REFERENCES

[1] Patanker S.V., and Spalding D.B.(1972)," A calculation Procedure for Heat , Mass and Momentum Transfer in Three-Dimensional Parabolic Flows", Int. Journal of Heat and Mass Transfer , Vol. 15, pp. 1787-1806.

[2]Fanger, P.O., "*Thermal comfort*, McGraw-Hill", New York, 1970.

[3]Markatos, N. C., "The Computer Analysis of Building Ventilation and Heating Problems". *Passive and Low Energy Architecture*, 1983, 667-675.

[4]Chow, W. K. and **Fung**, W. Y., "Numerical studies on the indoor air flow in the occupied zone of ventilated and air-conditioned Space". *Building and Environment,* 1996, 31(4), 319-344.

[5]Cheong, K. W. D., **Djunaedy** E., **Chua** Y.L., **Tham** K.W., **Shekar** S.C., **Wong** N.H., **Ullah** M.B "Thermal comfort study of a lecture theatre environment-a case study approach",National Univercity of Singapore 2000.

[6]Rohles, F.H., and Nevins, R.G., "the nature of thermal comfort for sedentary man", ASHRAE transactions, 1971 vol. 77, part1, PP. 239-346.

[7]Negrao C.O.R., **Franco** A.T. and **Macedo** L.M. "Simplified model to predict indoor air temperature distribution" Rio de Janeiro,Brazil August 13-15,2001.

[8]Pitarma, R.A., Ramos, J.E., Ferreira, M.E. and Carvalho, M.G. "Computational fluid dynamics. Advanced active tool in environmental management and education". 2004.

[9]Abid Zedan KHLAF "evaluation of thermal comfort", M.sc. thesis, university of mosul,Iraq, 1982.

[10]Chen Q. (1988),"Indoor Air Flow, air Quality and Energy Consumption of Buildings", PhD. Thesis, Delft. University of Technology, the Netherlands

[11]Hui, S.C.M.(1998),"Simulation Based Design Tools For Energy Efficient Buildings", Hong Kong Papers in Design and Development, Volume 1.

[12]Launder B.E., and Spalding D.B. (1974),"The Numerical Computation of Turbulent Flows", Computer Methods in Applied Mechanics and Engineering, Vol. 3, pp. 269-289.

[13]Matthew P.S. (2000),"Computation and Measurement of Wind Induced ventilation", PhD Thesis, University of Nottingham.

[14]Pun, W. M. and **Spalding**, **D. B., "**A general computer program for two-dimensional elliptic flows", 1977, Imperial College (London), Mech. Eng. Dept. Report HTS/76/2.

[15]Mohammad W. S. “Space air-conditioning of mechanically ventilated room: computation of flow and heat transfer”, Ph.D. thesis, Granfield Institute of Technology, 1986, London.

[16] Liu,J., **Aizawa**, Y. **Yoshino**,H., 2000. “Experimental and numerical study on simultaneous temperature and humidity distributions”, Technical research institute, Tokyo gas Co., Ltd. 16-25, Shibaura

SYMBOLS

SIMPLE semi-Implicit Method for Pressure Liked Equation RH Relative Humidity

ACH Air Change per Hour q_1 Latent heat flux from source (W. m^{-2})

CFD Computation Fluid Dynamics q_2 sensible heat flux from source (W. m^{-2})

C_1, C_2, C_3 Coefficient in turbulence model.

G_K Generation rate of turbulence energy. G_B Buoyancy production.

H Room height, building height (m). W_d Room width (m)

H_S Source height (m). L Room length, building length (m).

J Total (convection +diffusion) flux. Φ Time-averaged value of Φ.

L_S Length of internal source (m). *bs* Width of internal source (m)

ε Rate of dissipation of (K). ($m^2.s^{-3}$) k Turbulent kinetic energy.

= moisture flux, (kg/s . m)

ω_i, ω_∞ = humidity at the saturated air layer and main free-stream stream, kg.w.v/ kg.d.a

h_i, h_∞ =Enthalpy at the saturated air layer and main free –stream, kJ / kg

γ = latent heat, kJ / kg

q'' = heat flux per unit area, W/ m^2

α = coefficient that expresses the ratio of heat flux which is generated by latent heat to the total heat flux. It can be considered as a function of temperature difference between water surface and ambient air.

Table (1) Exchange Coefficient and Source Terms of f

Conserved property	f	Γ_f	S_f
Mass (continuity)	1	0	*0*
Direction-I momentum	u_i	m_{eff}	$-\frac{\partial p}{\partial x_i}+\frac{\partial p}{\partial x_j}[m_{eff}+\frac{\partial u_j}{\partial x_i}]-rg_i bq$
Thermal energy (Temperature)	T	$\Gamma eff ,T$	*0*
Moisture content	ω	Γ_{eff} ,w	*0*
Turbulence kinetic energy	K	$\frac{m_{eff}}{s_k}$	G_k+G_B-re
Turbulence energy dissipation	e	$\frac{m_{eff}}{s_e}$	$\frac{e}{k}[C_1G_k+G_B)-C_2re]$

Table (2) Constant of Turbulence Model.

C_1	C_2	C_D	s_K	s_e
1.44	1.92	0.09	1.0	1.3

Table (3) Boundary conditions for the verification case

Variables	Boundary condition							
	At inlet	At source	At walls					
			Floor	Ceiling	Left wall	Right wall	Up wall	Down wall
U (m / sec)	0	0	0	0	0	0	0	0
V (m / sec)	0	0	0	0	0	0	0	0
W (m /sec) mean axis velocity	0.5	0	0	0	0	0	0	0
T (^oC)	14	35	14	14	14	14	14	14
ω (kg / $kg_{d.a}$)	0.0053	0.033	0.0075	gradient=0				
K (J)	K_{in}	The gradient normal to the walls set equal to zero						
ε	ε_{in}	0	0	0	0	0	0	0

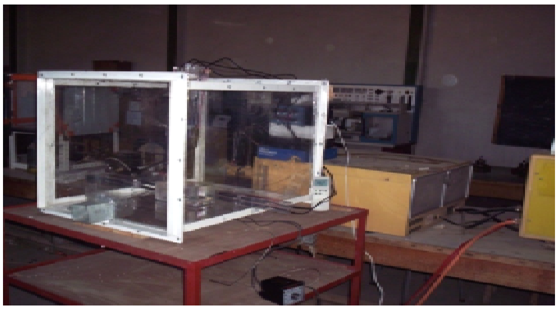


Figure (1): Photograph of the experimental rig

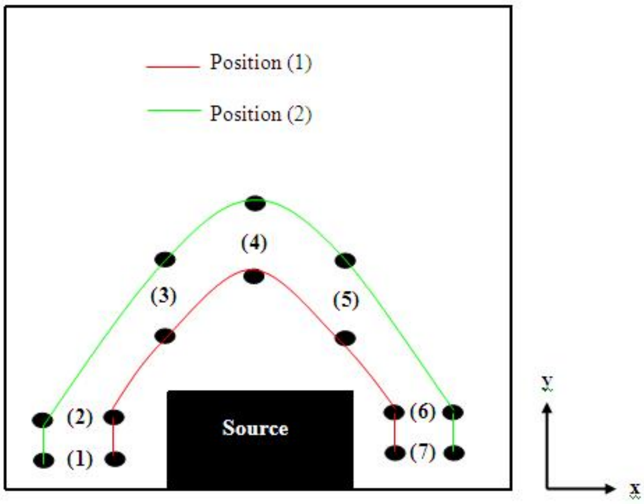


Figure (2): Sensors layout in the experimental models

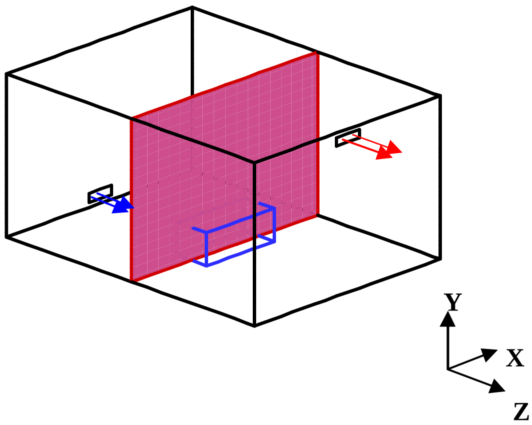


Figure (3) Schematic diagram of measuring plane in the experimental models

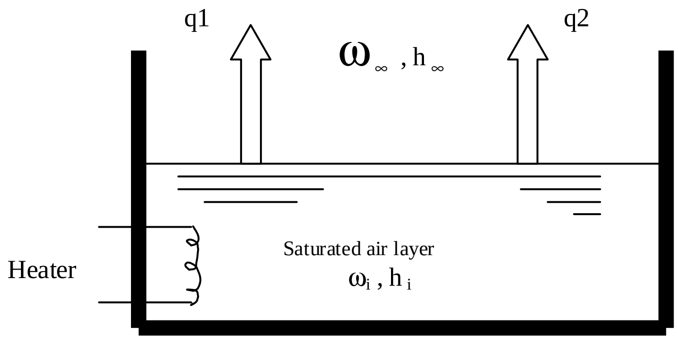


Figure (4) Schematic of simultaneous heat and moisture transfer [14]

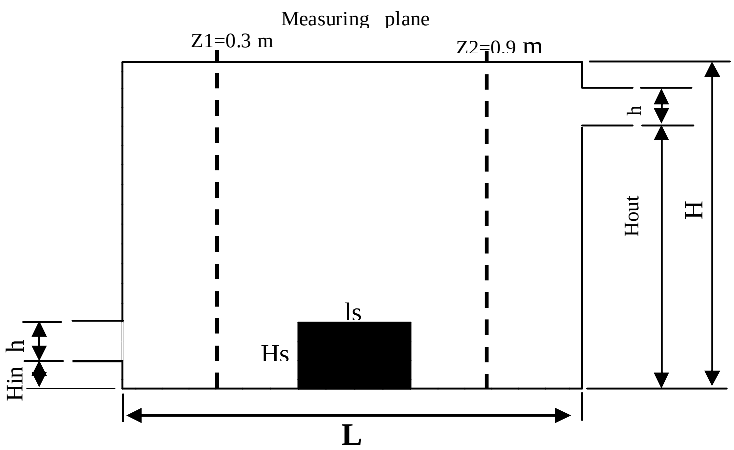
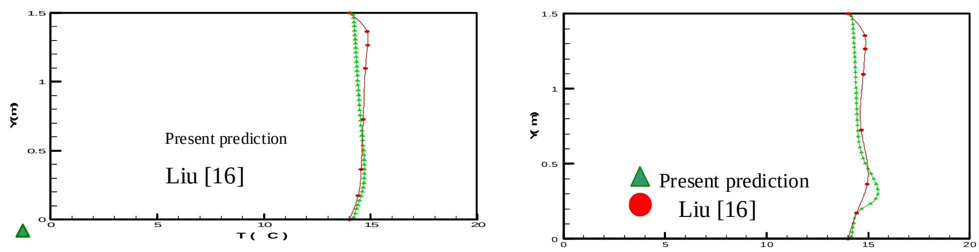
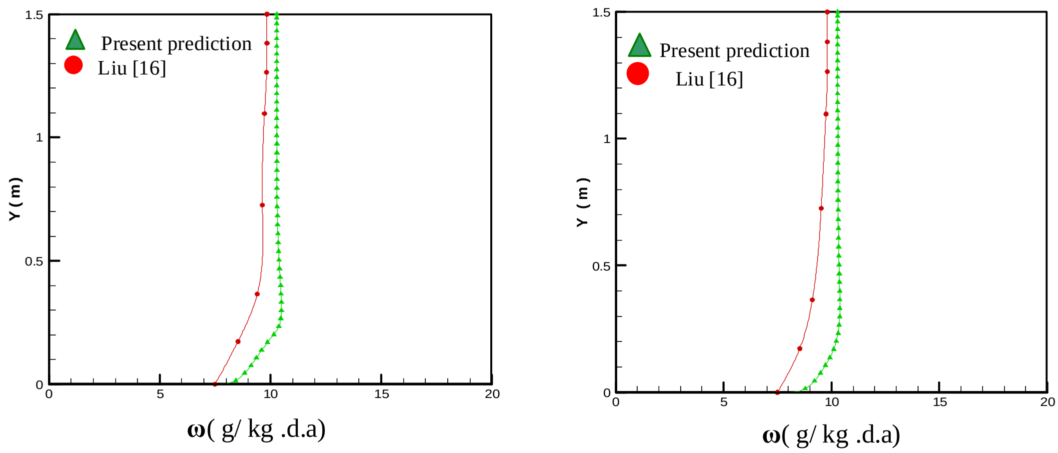


Figure (5) Schematic Diagram of space flow



(a) **(b)**

Figure(6) A comparison of the temperature between the present prediction and that of Liu [16] at:(a) Z1=0.3 m , (b) Z2=0.9 m



(b) **(a)**

Figure (7) a comparison of the moisture content between the present Prediction and that of Liu et al [16] at :
(a) Z1=0.3 m , (b) Z2=0.9 m

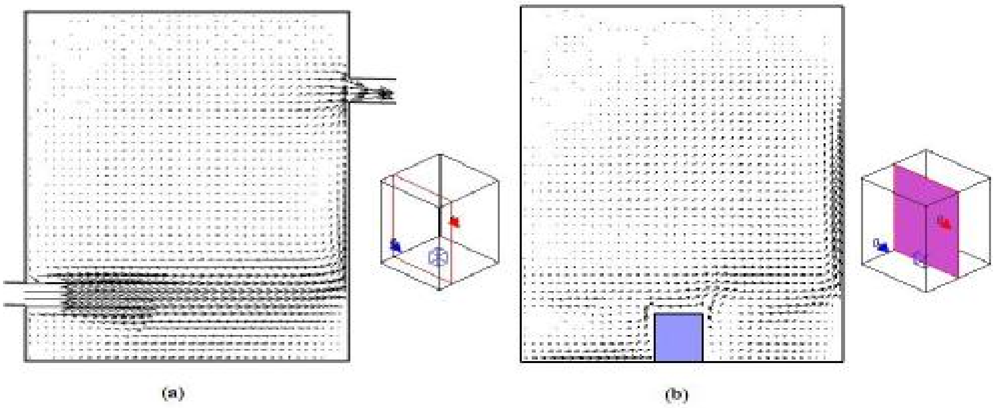


Figure (8) Velocity vector representing the air-flow distribution inside the space for:
(a) Inlet and outlet (b) middle of the space

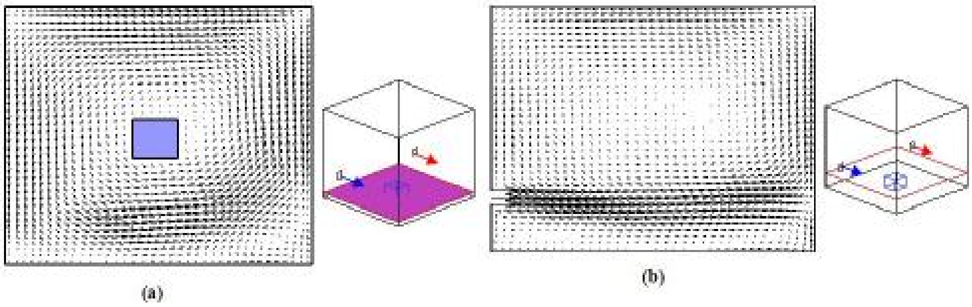
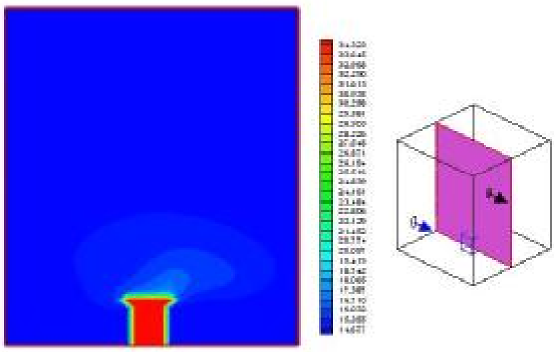


Figure (9) Velocity vector co representing the air-flow distribution inside the space for top view:
(a) Middle of the source (b) Inlet



Figure(10) Temperature contour at the middle of the space

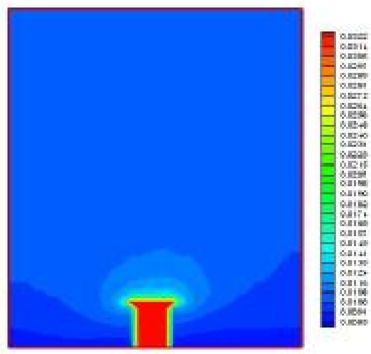
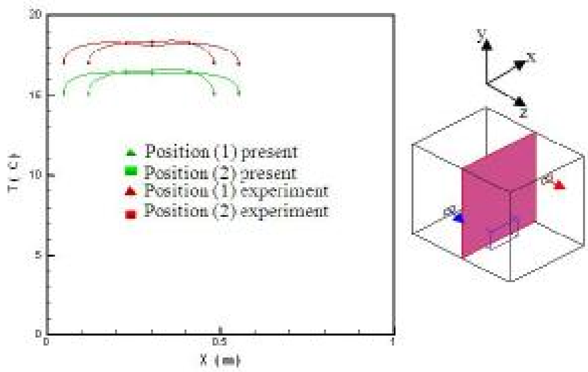


Figure (11) Moisture content contour at the middle of the space



Figure(12) a comparison of the temperature between the present prediction and experiment results for positions (1, 2).

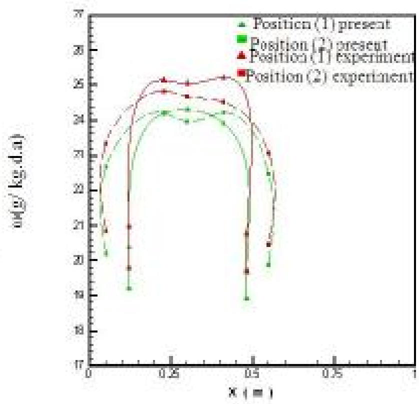
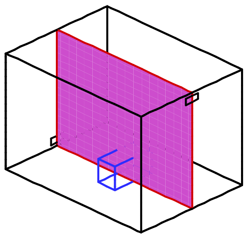


Figure (13) a comparison of the moisture content between the present prediction and experiment results for positions (1, 2)



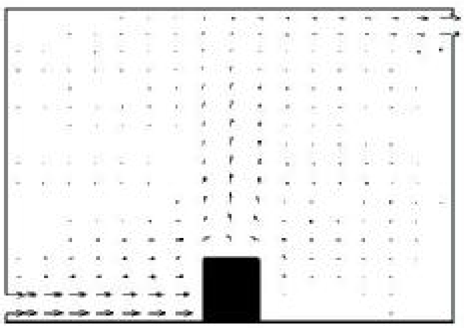


Figure (14) Velocity vector representing the air-flow distribution inside the space

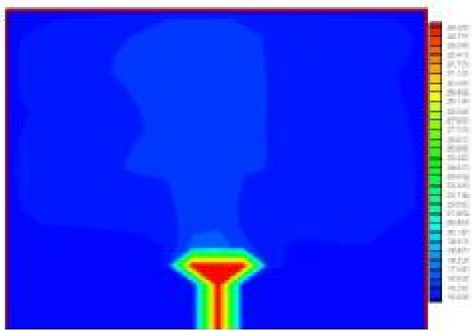


Figure (15) Temperature contour at the middle of the space

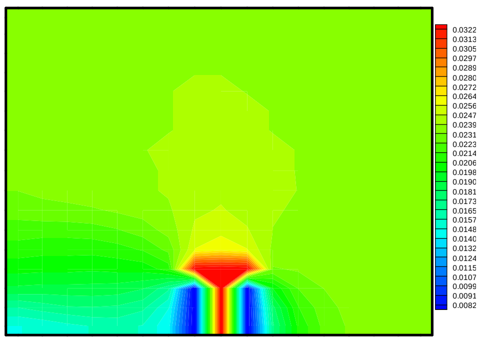


Figure (16) Moisture content contour at the middle of the space

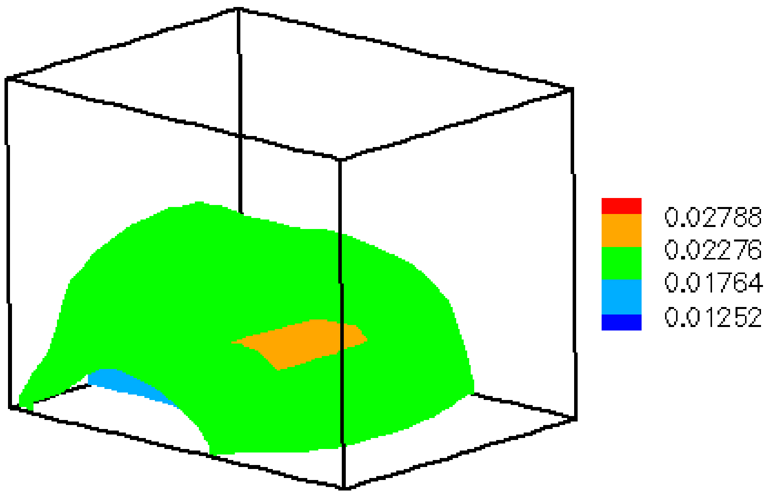


Figure (17) Moisture content contour at the 3-dimension of the space