The Effect of Workers and Computers on the Indoor Air Flow in Office Room

Dr. Mohammed Ali Mahmood **Dr. Wajeeh Kamal Hasan**

drmohammed_ali2003@yahoo.com dr.wajeeh21@gmail.com

Al-Rafidain University Collage

Abstract: *This paper deals with the Three-dimensional CFD simulations are carried out to investigate the effect of workers and computers on the indoor air flow in an office room. Standard k-ω turbulence model was used to modulate the air flow velocities and indoor temperatures with its gradients. Physical processes in room with different planes are modeled using specialized computational fluid dynamics (CFD) software ANSYS /FLUENT 13. The enclosed space is fundamental impressed by the velocity of airflows, and the vertical temperature gradient. It is shown that it is possible to save good flow distributions, at the same time maintaining the conditions of thermal comfort in the room which is the main goal. Creating suitable thermal conditions for satisfying human desires, the thermal comfort has been recognized to be an essential requirement of the indoor environment.*

Key Wards: Three-dimensional, cfd , flow simulations, room

1. Introduction

The need of precise determination of air flow pattern and temperature distribution in a room was realized by HVAC engineers to provide comfortable temperature conditions, and air velocities in the occupied space.Since the distribution of air velocity and temperature in a ventilated space is not uniform, whole flow field airflow patterns and air velocity distributions are essential information to understand performance of air conditioning systems, occupant comfort and well-being. Prototype studies are expensive and time consuming, largely due to limitations of the current available measurement technologies and instrumentation. Interests in simulating the airflow through large openings in buildings, such as Person's, computers, windows or doors, allowing for bidirectional flow, are increasing. However, perfect modeling of air flow and temperature distribution is very tricky due to suitability of different turbulent models by Nielsen [1] used for CFD simulation of room air flow and hence validation is essential. An extensive literature review on the application of CFD to building ventilation and IAQ (indoor air quality) problems was performed by Emmerich [2], which includes room airflow case for various ventilation system, and it was determined that Large eddy simulation has unique capabilities compared to other CFD model. Emmerich and McGrattan [3] described a three-dimensional, large eddy simulation (LES) model developed for studying the transport of smoke and hot gases during a fire in an enclosure. Williams, et. al. [4] described three-part paper documents research completed on the topical aspects of computational fluid dynamics (CFD) associated with the room air motion problem. Eftekhari ,etal. [5] Investigated airflow distribution inside a test room, which is naturally ventilated

Abanto ,et. al. [6] described the study of the numerical simulation of airflow and the prediction of comfort properties in a visualization room of a research centre. Fan [7] has used various models to solve the air and contaminant distribution in rooms. Chen and Xu [8] have proposed a new zero-equation model to simulate three dimensional distributions of air velocity, temperature, and contaminant concentrations in rooms. Zhao ,etal. [9] proposed new numerical method and claimed that it can correctly simulate air velocity and temperature distributions in the room except in a few positions by less computing time than using conventional CFD methods. Jouvray and Tucher ,et.al. [10] used nonlinear RANS model for complex geometry, which showed impressive performance for validation. Zhao ,et.al. [11] developed a PIV (Particle Image Velocimetry) measurement system for indoor airflow field studies. Quantative air velocity distribution can be obtained to validate numeric models and analyze ventilation strategies. The results showed that the PIV technique can be effective method to quantitatively measure the room air velocity, especially for those regions with very low velocity .Posner ,etal. [12] compared result from relatively simple three dimensional numerical simulations (CFD) with laser Doppler anemometry (LDA) and particle image velocity meter (PIV) experimental measurement of indoor air flows in a one tenth scale model room.Staman and Katsiris [13] used the SST based k-ω model to calculate air-flow velocities and temperatures in a model office room. Calculations were compared with experiments and with the results of the standard k-ε, RNG k-ω model and the laminar model and concluded that all the three tested turbulent models predict satisfactorily the main qualitative features of the flow and the layered type of temperature fields and computations with the SST k-ω based model showed the best agreement with measurements. The main aim of the present study is to investigate the effect of workers and computers on the indoor air flow in in a model room numerically using three dimensional CFD simulation.

2. CFD Simulation

The physical model of the office room for which the numerical studies were conducted is shown in Figure 1. A rectangular enclosure X=7m, y=8m, and z=4m.

The initial and boundary conditions for the model are:-

- ❖ Enable Gravity and specify Z-component of Gravitational Acceleration as -9.81 m/s2.

- Enter Operating Density as 1.225 kg/m3.
- Operating Pressure (Pascal) to 101325.
- Set a constant wall temperature of 310 K.
- Cold flow Temperature of 294K.
- Cold flow rate 0.225 kg/s.

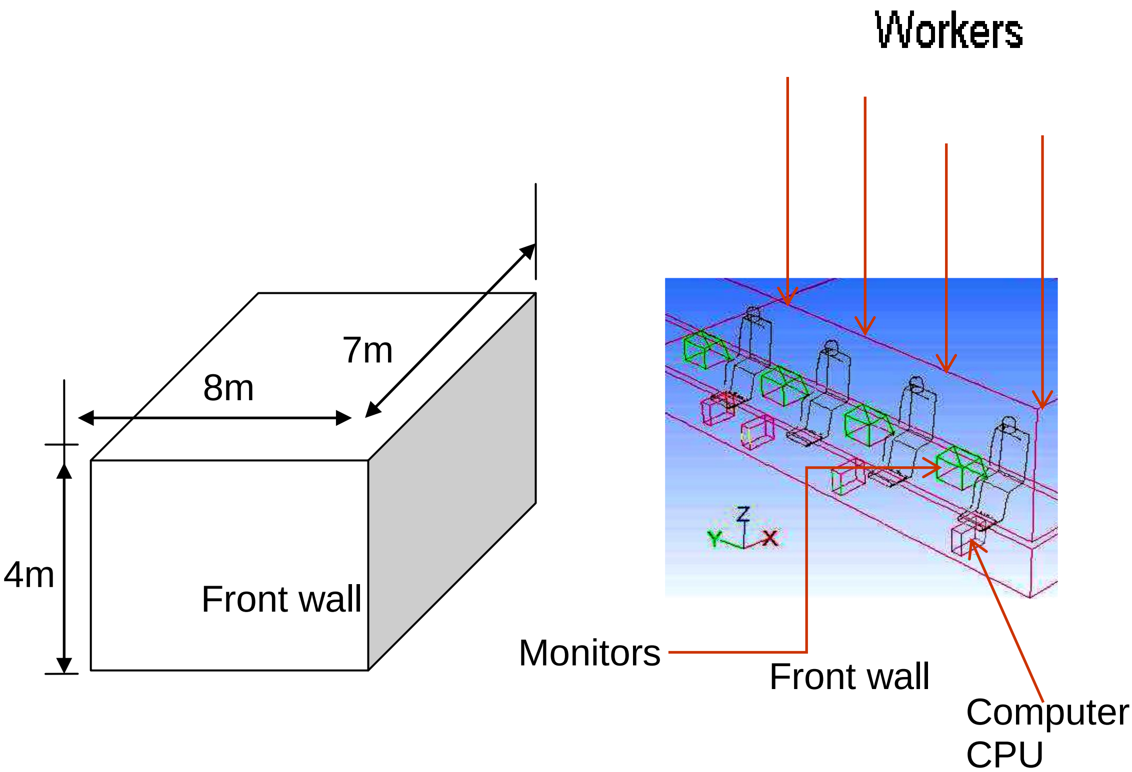

Figure 1: The physical model of the office room

The following continuity, momentum and energy equations are applicable for constant properties and describing the three-dimensional fluid flow and heat transfer in the air-conditioned or ventilated space:

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

$$\rho\left(\bar{u}\frac{\partial\bar{u}}{\partial x}+\bar{v}\frac{\partial\bar{u}}{\partial y}+\bar{w}\frac{\partial\bar{u}}{\partial z}\right)=-\frac{\partial\bar{p}}{\partial x}+\mu_{eff}\left(\frac{\partial^2\bar{u}}{\partial x^2}+\frac{\partial^2\bar{u}}{\partial y^2}+\frac{\partial^2\bar{u}}{\partial z^2}\right) \tag{2}$$

$$\rho\left(\bar{u}\frac{\partial\bar{v}}{\partial x}+\bar{v}\frac{\partial\bar{v}}{\partial y}+\bar{w}\frac{\partial\bar{v}}{\partial z}\right)=-\frac{\partial\bar{p}}{\partial y}+\mu_{eff}\left(\frac{\partial^2\bar{v}}{\partial x^2}+\frac{\partial^2\bar{v}}{\partial y^2}+\frac{\partial^2\bar{v}}{\partial z^2}\right) \tag{3}$$

$$\rho\left(\bar{u}\frac{\partial\bar{w}}{\partial x}+\bar{v}\frac{\partial\bar{w}}{\partial y}+\bar{w}\frac{\partial\bar{w}}{\partial z}\right)=-\frac{\partial\bar{p}}{\partial z}+\mu_{eff}\left(\frac{\partial^2\bar{w}}{\partial x^2}+\frac{\partial^2\bar{w}}{\partial y^2}+\frac{\partial^2\bar{w}}{\partial z^2}\right) \tag{4}$$

$$\rho c_p\left(\frac{\partial\bar{T}}{\partial t}+\bar{u}\frac{\partial\bar{T}}{\partial x}+\bar{v}\frac{\partial\bar{T}}{\partial y}+\bar{w}\frac{\partial\bar{T}}{\partial z}\right)=k_{eff}\left(\frac{\partial^2\bar{T}}{\partial x^2}+\frac{\partial^2\bar{T}}{\partial y^2}+\frac{\partial^2\bar{T}}{\partial z^2}\right) \tag{5}$$

Where, effective viscosity and thermal conductivity are given by,

$$\mu_{eff}=\mu+\mu_t\,,\quad k_{eff}=k+k_t \tag{6}$$

The shear-stress transport (SST) k-omega turbulence model is a type of hybrid model, combining two models in order to better calculate flow in the near-wall region. It was designed in response to the problem of the k-epsilon model's unsatisfactory near-wall performance for boundary layers with adverse pressure gradients. It utilizes a standard k-epsilon model to calculate flow properties in the free-stream (turbulent) flow region far from the wall, while using a modified k-ε model near the wall using the turbulence frequency ω as a second variable instead of turbulent kinetic energy dissipation term ε.

The SST k-omega model is similar to the k-epsilon turbulence model, but instead of ε as the second variable, it uses a turbulence frequency variable omega, which is expressed as $\omega = \varepsilon/k$ [s^{-1}]. The SST *k-omega* model computes Reynolds stresses in the same way as in the *k-epsilon* model. The transport equation for turbulent kinetic energy k for the k-ω model is:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho U_i k\right)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)grad(k)\right]+P_k-\beta^*\rho k\omega$$

$$where\ P_k=\left(2\mu_t\frac{\partial U_i}{\partial x_j}\cdot\frac{\partial U_i}{\partial x_j}-\frac{2}{3}\rho k\frac{\partial U_i}{\partial x_j}\delta_{ij}\right) \qquad \text{…(7)}$$

σ_k and β^* are equation constants.

The transport equation for turbulent frequency ω for the k-ω model is:

$$\frac{\partial\rho\omega}{\partial t}+\frac{\partial}{\partial x_i}\left(\rho U_i\omega\right)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_{\omega,1}}\right)grad(\omega)\right]+\gamma_2\left(2\rho\frac{\partial U_i}{\partial x_j}\cdot\frac{\partial U_i}{\partial x_j}-\frac{2}{3}\rho\omega\frac{\partial U_i}{\partial x_j}\delta_{ij}\right)-\beta_2\rho\omega^2+2\frac{\rho}{\sigma_{\omega,2}\omega}\frac{\partial k}{\partial x_k}\frac{\partial\omega}{\partial x_k}$$

Where $\sigma_{\omega,1}$, γ_2, β_2, *and* $\sigma_{\omega,2}$ are constants **…(8)**

The constants for the SST *k-omega* turbulence model are listed in table 1.

Table 1, Constant values for SST k-omega turbulence model equations.

β^*	β_2	σ_k	$\sigma_{\omega,1}$	$\sigma_{\omega,2}$	γ_2
0.09	0.083	1.0	2.0	1.17	0.44

Additional modifications have been made to the model for performance optimisation. There are blending functions added to improve the numerical stability and make a smoother transition between the two models Versteeg [14].

The results included are obtained after running for 554 iterations by using iso-surfaces mesh created at various locations of the domain to examine the results at any location within the domain, not just at the boundaries.

3. Results and Discussions

Numerical simulations results have been discussed and performed to simulate the effect of workers and computers on the indoor air flow in the test geometry representative of the enclosed space The flow field is examined; key physical mechanisms, resulting from the interactions between workers , computers and the room air are identified and their effect on the thermal field are highlighted .

- **The computed distribution of temperature contour:**

In order to understand the behavior of the flow, several (Z-Y) planes were taken to give details about the effect of workers and computers on the temperature distribution behavior in the room as shown in Figures (2-6). Figure (2) shown the temperature contours in (Z-Y) plane at X=0.25m from the front wall. This figure showed the vertical temperature gradient due to the effect of the computers and workers heat. Figures (3,4,5&6) shown the temperature contours in (Z-Y)plane at X=1m,1.25m,1.5&2m from the front wall, where the vertical temperature gradient decreases as we move far away from the front wall also we can see that air movement is controlled by buoyancy. The cconvective flows, is the engine of vertical temperature gradient.Air movement is controlled by buoyancy due to free convection around heat sources Stratified flow in the room .

Figure (2) indicates that the maximum interactions between workers, computers and the indoor air flow at x=0.25m from the wall at the middle point between workers and computers in which the interaction area for this case covers a very wide range.

- **The computed distribution of velocity vectors :**

The velocity vectors distribution shown in Figure (7). The air movement is controlled by buoyancy .The free convection around heat sources (workers and computers) placed close to each other will merge into a space, stratified flow. Figure (7) indicates that the velocity increase at the region between workers at x=0.25 m from the front wall where a counter clockwise and a clockwise recirculation cell covering this region.

4. Conclusion

In this work the effect of workers and computers on temperature and velocity is simulated. Numerical analysis is used to solve three dimensional steady flows in enclosed space and resulting temperature distribution and air velocity field for closed room. The analysis of all the above results have led to the following conclusion:-

- The maximum interactions between workers, computers and the indoor air flow at x=0.25m from the wall at the middle point between workers and computers in which the interaction area for this case covers a very wide range.
- Displacement flow Controlled by buoyancy (temperature differences).
- The velocity vector and temperature contours reduced as we move forward from the middle point between workers and computers.

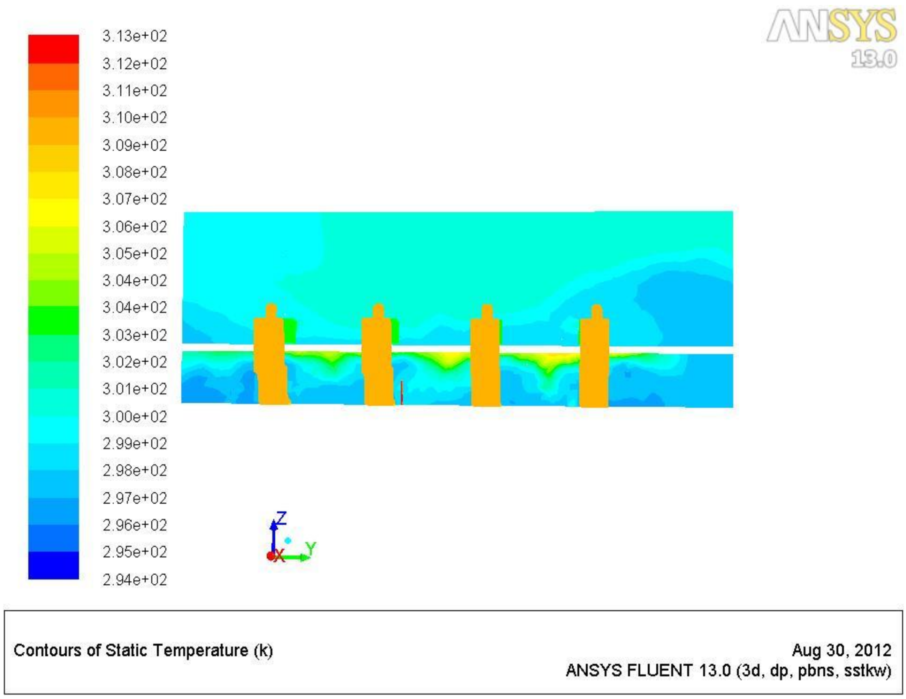

Figure 2: Temperature contour in (Z-Y)plane at X=0.25m

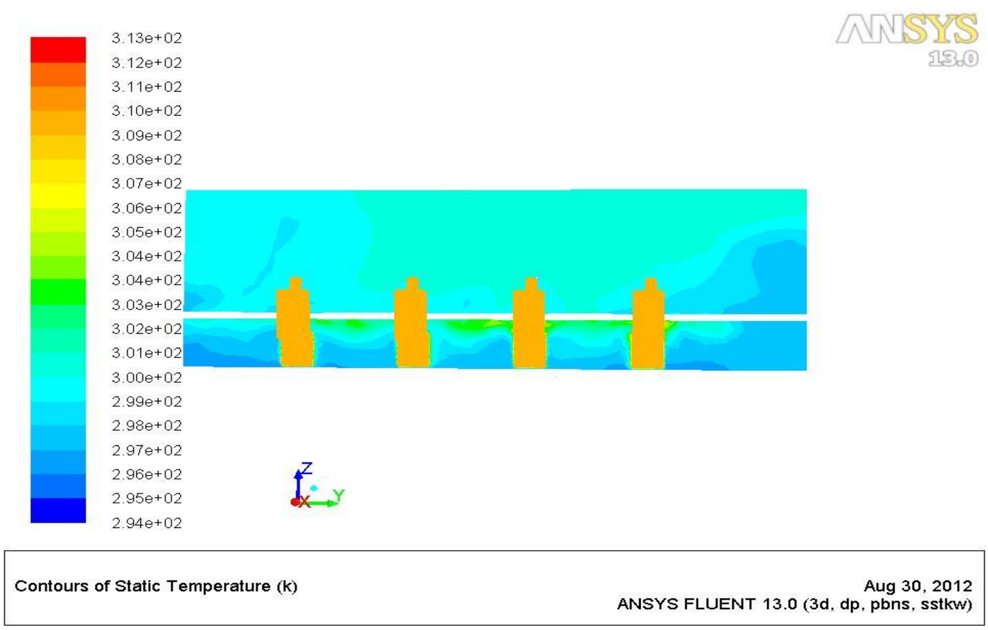

Figure 3: Temperature contour in (Z-Y)plane at X=1m

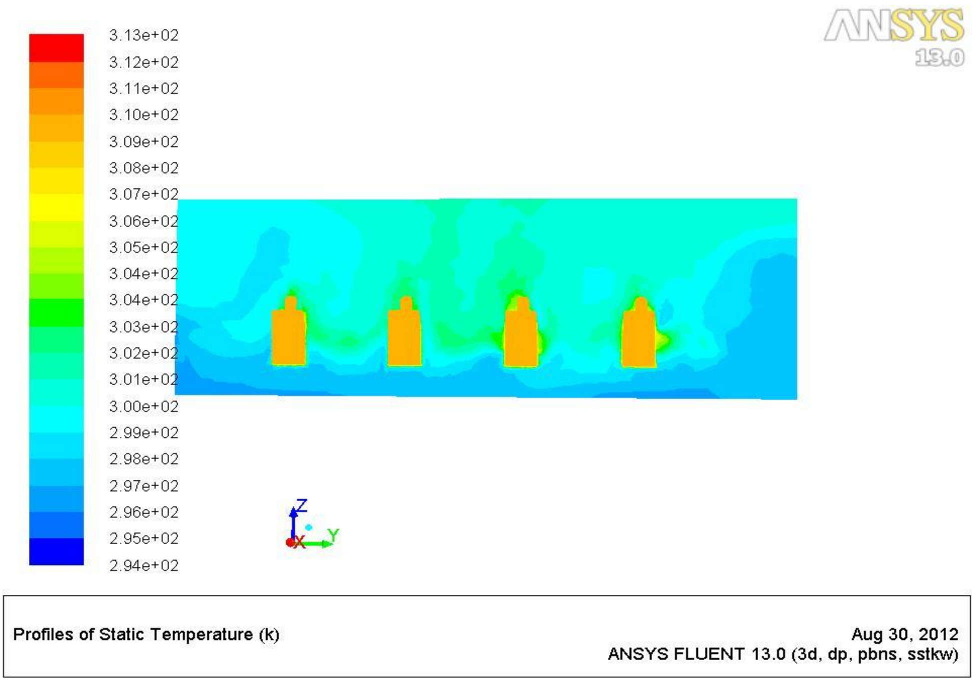

Figure 4: Temperature contour in (Z-Y)plane at X=1.25m

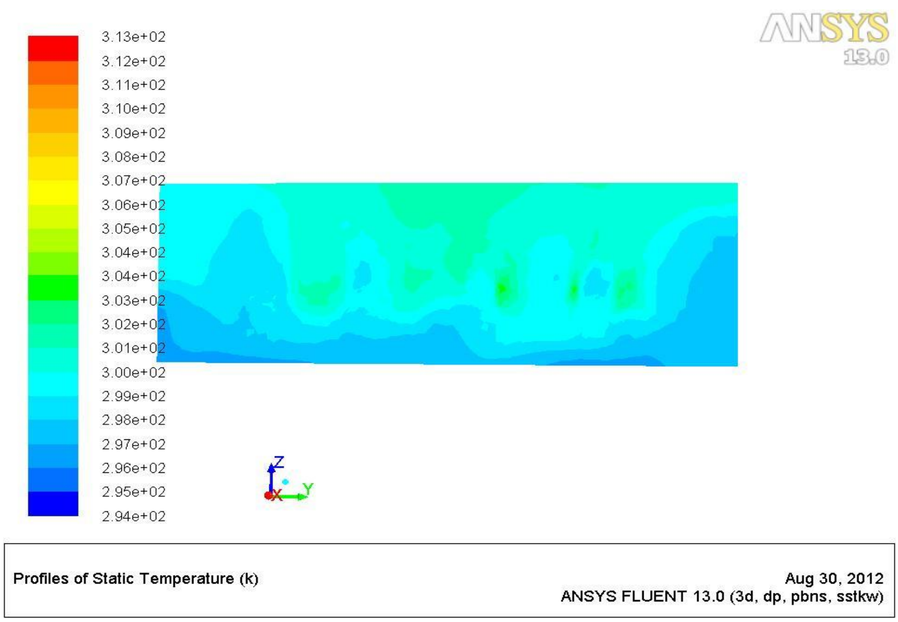

Figure 5: Temperature contour in (Z-Y)plane at X=1.5m

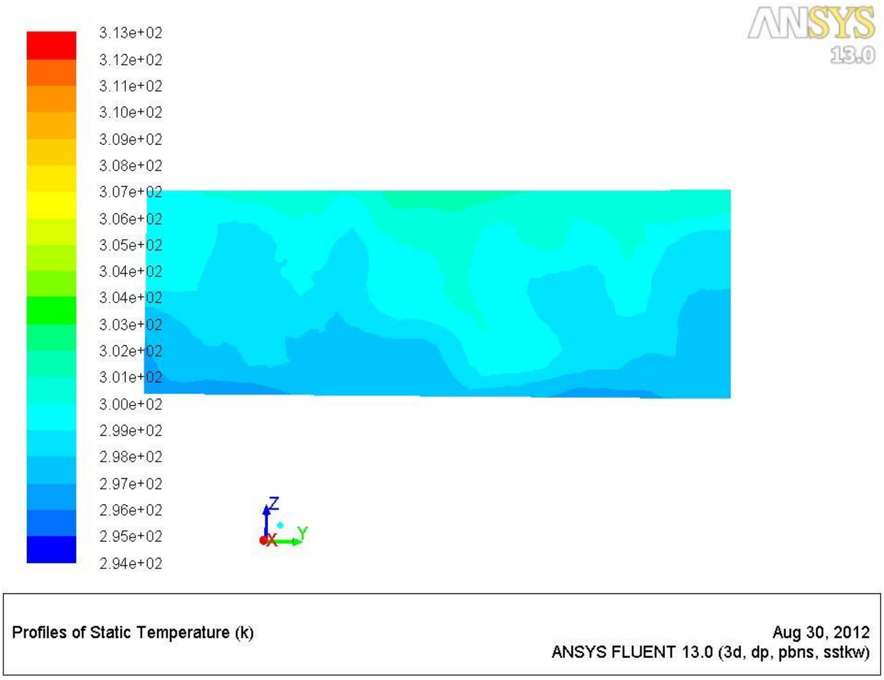

Figure 6: Temperature contour in (Z-Y)plane at X=2m

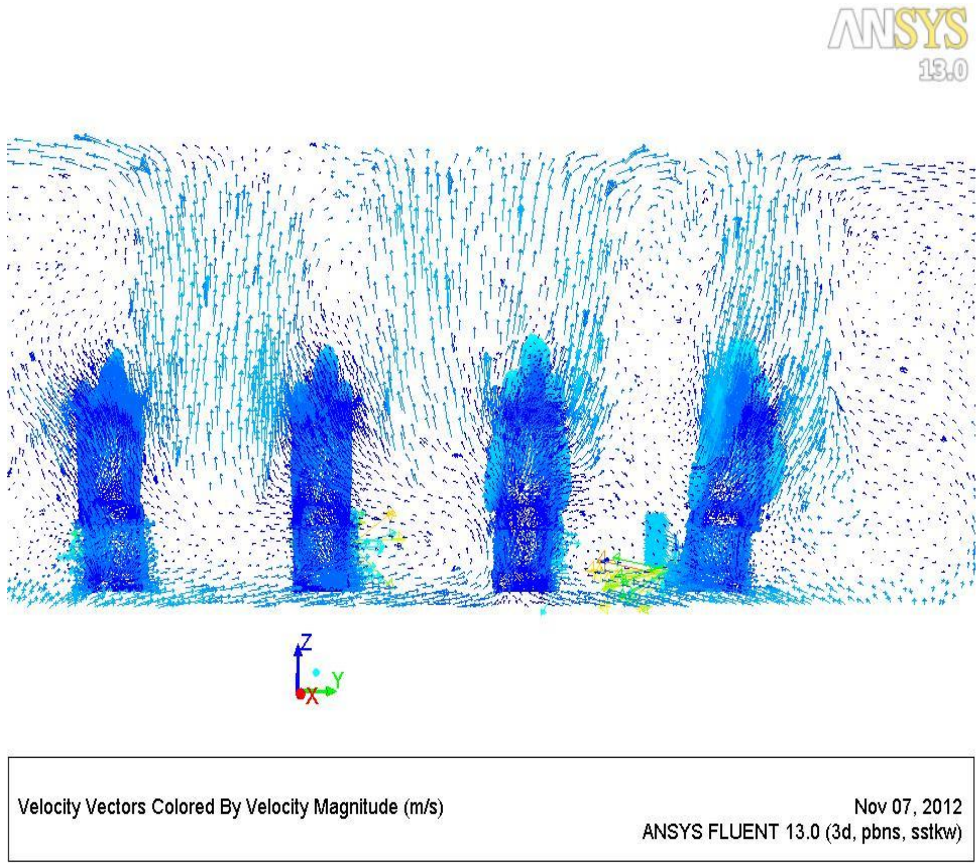

Figure 7: Velocity vectors in (Z-Y)plane at X=0.25m

References

[1] Nielsen, P.V., ***"The Selection of Turbulence Models for Prediction of Room Airflow"***, ASHRAE Transactions,vol. 104, no. 1: p. 1119-1127, 1998

[2] Emmerich, S.J., ***"Use of computational fluid dynamics to analyze indoor air quality issues"***, NISTIR 5997,National Institute of Standards and Technology, USA, 1997

[3] Emmerich, S.J., McGrattan, K.B., ***"Application of Large Eddy Simulation Model to Study Room Airflow"***, ASHRAE Transactions, vol. 104, no. 1: p. 1128-1137, 1998

[4] Williams, P.T., Baker, A.J., Kelso, R.M., ***"Numerical calculation of room air motion-Part 1: Math, Physics, andCFD modeling"***, ASHRAE Transactions, vol. 100, no. 1: p. 514-530, 1994

[5] Eftekhari, M.M., Marjanovic, L.D., Holmes, M., Hanby, V., ***"Air flow distribution for different wind pressure coefficient values in a naturally ventilated room"***, ASHRAE Transactions, vol. 108, no. 2: p. 1233-1240, 2002.

[6] Abanto, J., Barreo, D., Reggio, M., Ozell B., ***"Airflow modeling in a computer room"***, Building & Environment, vol. 39, no. 12: p. 1393-1402,2004

[7] Fan, Y., ***"CFD modeling of the air and contaminant distribution in rooms"***, Energy & Buildings, vol. 23, no. 1:p. 33-39. 1995

[8] Chen, Q., Xu, W., ***"A Zero-equation turbulence model for indoor airflow simulation"***, Energy & Buildings, vol.28, no. 2: p. 137-144. 1998,

[9] Zhao, B., Li, X., Yan, Q., ***"A simplified system for indoor airflow simulation"***, Building & Environment, vol.38, no. 4: p. 543-552. 2003

[10] Jouvray, A., Tucker, P.G., Liu, Y., ***"On nonlinear RANS models when predicting more complex geometry room air flows"***, Int. J. Heat Fluid Flow, vol. 28, no. 2: p. 275–288. 2007

[11] Zhao, L., Zhang, Y., Wang, X., Riskowski, G.L., Christianson, L.L., ***"Measurement of two-dimensional air velocity in a full-***

scale room using particle image", ASHRAE Transactions, vol. 107, no. 2: p. 434-443. 2001

[12] Posner, J.D., Buchanan, C.R., Dunn-Rankin, D., ***"Measurement and prediction of indoor air flow in a model room"***, Energy & Buildings, vol. 35, no. 5: p. 515-526. 2003

[13] Stamau, A., Katsiris, I., **"Verification of a CFD model for indoor air flow and heat transfer"**, Building & Environment, vol. 41, no. 9: p. 1171-1181. 2006

[14] Versteeg, H K; Malalasekera, W. ***"An Introduction to Computational Fluid Dynamics, The Finite Volume Method"***, Second edition, Pearson Education Limited, Essex, England (2007).

NOMENCLATURE

SYMBOL	DESCRIPTION	UNIT
K	Thermal Conductivity	W/m. °C
k_{eff}	Effective Thermal Conductivity	W/m. °C
T	Temperature	°C
u	Component of Velocity in x-Dir.	m/s
v	Component of Velocity in y-Dir.	m/s
w	Component of Velocity in Z-Dir.	m/s
μ	Dynamic Viscosity	kg /m.s
μ_{eff}	Effective Dynamic Viscosity	kg /m.s
ρ	Density	kg /m^3
$\sigma_{\omega,1}$, γ_2, β_2, *and* $\sigma_{\omega,2}$	Constants of turbulence model (*K*-*w*).	-
Subscripts		
eff	Effective value.	
_	Average value	

تأثير العاملين والحاسباتِ على جريان الهواء داخل غرفةِ مكتبَ

م.د. محمد علي محمود حسين
drmohammed_ali2003@yahoo.com

م.د. وجيه كمال حسن
dr.wajeeh21@gmail.com

كلية الرافدين الجامعة

المستخلص:

يتناول البحث محاكاة ثلاثية ابعاد باستخدام ديناميك الموائع الحسابي الاضطرابي لتَحرّي تأثيرِ العاملين والحاسباتِ على جريان الهواء داخل غرفة مكتبِ . تم استخدام نموذج (k-ω) القياسي للجريان الاضطرابي لعرض سُرَعِ التدفقِ ودرجاتِ الحرارة الداخليةِ. تمت محاكاة العمليات الفيزيائية التي تحدث داخل المكتب بالاعتماد على حقيبة برمجيات (ANSYS /FLUENT). تم تبيان التدرج الحراري العمودي وسرعة الهواء داخل الفضاء المحدد واللذان يَلْعبانِ دورَ مهمَ في التصميمِ الامثل لتكييف الحيز وتهويته. تم بيان امكانية تَوفير توزيعاتِ التدفقِ الجيدةِ والتي يَبقي شروطَ الراحةِ الحراريةِ في الغرفةِ وهو الهدفُ الرئيسيُ.

الكلمات المرشدة: ثلاثية الابعاد ، ديناميك الموائع الحسابي ، تمثيل الجريان ، غرفة.