NUMERICAL SIMULATION OF THE CEILING DUCT COLD FLOW EFFECT ON TEMPERATURE DISTRIBUTION IN CLOSED SPACE+

Mohammed Ali Mahmood Hussein *

Abstract :

This research deals with the study of the three-dimensional heat transfer and air flow for enclosed space air conditioning with cold ceiling shower. Numerical modeling has been used to analyze the effect of ceiling air conditioning on temperature and air velocity distribution . The analyses showed the influence of these factors on the air circulation and the related temperature distributions through enclosed space. The modeling and simulation of a space with using standard k-ω turbulence model for modeling air flow velocities and indoor temperatures with its gradients are analyzed. 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. This work indicated that the best penetration of cold flow is for the plane passes through the center of the ceiling duct in which the penetration area for this case covers a very wide range in room these planes are modeled using specialized computational fluid dynamics (CFD) software ANSYS /FLUENT 13.

المحاكاة العددية لتأثيرِ تدفقِ التيار البارد من قناةِ السقفَ على توزيعِ درجةِ الحرارة في فضاء مغلق

محمد علي محمود حسين

المستخلص:

تناول هذا البحثِ دراسةِ انتقال الحرارةِ وجريان الهواء الثلاثي الأبعاد لتكييف فضاءِ مُغلقٍ من خلال منفذين للتيار البارد. حيث تم استخدام العرض العددي لتَحليل تأثير التكييف السقفي على توزيع درجةِ الحرارة والسرعةِ. وبينت التحليلاتُ تأثيرَ هذه العواملِ على دورانِ الهواء وتوزيعاتِ درجةِ الحرارة ذات العلاقةِ خلال فضاءِ مغلق . وباستخدام موديل قياسي (k-ω) نموذج اضطراب لعرض سُرَعِ التدفقِ ودرجاتِ الحرارة الداخلي. حيث تم بيان امكانية تَوفير توزيعاتِ التدفقِ الجيدةِ، في نفس الوقت يَبقي شروطَ الراحةِ الحراريةِ في الغرفةِ وهو الهدفُ الرئيسيُ .هذا العملِ أشارَ بأنّ أفضل إختراقِ للتيارِ البارد في المستوي الذي يَمُرُّ من خلالِ مركزِ قناةِ السقفَ الذي حقق الاختراق مدى عريض جداً في الغرفةِ حيث تم الاعتماد على ديناميك الموائع الحسابيةَ باستخدام برنامج(ANSYS /FLUENT 13).

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.

+Received on 3/9/2012 , Accepted on 15/5/2013 .
*Lech./ Al-Rafidain University Collagem

In recent years, extensive research effort has been made in the development of general fluid flow and heat transfer programs for solving the flow of air in a room. Similar studies had been carried out, Lee [1] applied the finite element method to study the characteristics of forced and mixed convection in an air-cooled room for both the laminar and turbulent regimes, Sinha [2] deals with case of two-dimensional numerical prediction of an air flow configuration in a room (enclosure) with inlet and outlet as applicable to cooling and ventilation. Three-dimensional model of a computer room with regard to thermal comfort was presented by Richtr [3] using standard k-ε turbulence model for modeling air flow. Rutman [4] Give extensive view on using numerical modeling tools for improving indoor environment quality, component optimization and complete system optimization.

For air temperature a general recommendation for design of heating and cooling processes for working offices and enclosed areas, where no major physical activities are carried out, is that the temperature has to be held constant in between 21-23°C. During summertime, when outdoor temperatures are higher, it is advisable to keep air-conditioned offices slightly warmer than in winter periods to minimize the high temperature difference between indoors and outdoors. Recommendation is that difference should not be higher than 7°C. Thermal comfort depends also on vertical temperature distribution in the room. Allowed vertical air temperature difference between 1.1m and 0.1m above the floor (head and ankle respectively of the person in sitting position) or between 1.7m and 0.1m above floor (head and ankle respectively of the person in upright position) is less than 3°C (ISO 7730 -1994).). According to Zukowski [5], the most comfortable state is that where floor temperature is the same or slightly higher than air temperature in head level.

State of thermal comfort cannot be determined simply by knowing air temperature in the room. It is known fact that at least 50% of all thermal energy between the environment and a human body exchanges through radiation. A human body is constantly exchanging radiant heat with all objects that surrounds it due to temperature difference between them. Mean Radiant Temperature (MRT) of a space is really the measure of the combined effects of temperatures of surfaces within that space on human body. A scale of seven degrees from -3

to +3 can be used to measure thermal sensation based on the scale of ASHRAE [6]. The larger the surface area and the closer one is to it, the more effect the surface temperature has on the individual. MRT thus changes with position in the room and with it corresponding air temperature which provides thermal comfort. In general if MRT drops by 1°C, air temperature has to be raised by 1 - 1.5°C.

Air flow velocity is important factor in determining thermal comfort because increased air velocity can cause discomfort due to increased convective heat transfer from surface of a human body. ISO 7730-1994 limits local air flow velocity in offices and spaces where no major physical activities occur at 0.15 m/s during winter periods (heating) and 0.25 m/s during summer (cooling). Sense of draft is more pronounced with decreased physical activity. Also, discomfort caused by draft increases with air temperature drop. Contrary, if air velocity in enclosed space is to low it could cause feeling of stuffiness and bad out door especially in winter (heating) periods. It is, therefore, necessary to assure optimal level of air circulation in the room. Fan [7] has used various models to solve the air and contaminant distribution in rooms.Chen [8] have proposed a new zero-equation model to simulate three dimensional distributions of air velocity, temperature, and contaminant concentrations in rooms. Zhao[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[10] used nonlinear RANS model for complex geometry, which showed impressive performance for validation.

Mathematical Model:

The fundamental set of equations that govern the flow of a fluid are derived from Newton's second law (the conservation of momentum). The equations are called the Navier-Stokes equation , it is more compact to write this equation in Cartesian tensor notation as [11]:

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_j u_i)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_i}[(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i})]+S_{bj} \quad \ldots(1)$$

Where: u, v and w are the stream wise, lateral and vertical velocity components respectively.

x, y and z are the corresponding directions to the velocity components.

μ : is the fluid viscosity,

ρ : is the fluid density, and

p : is the pressure.

represent the three-co-ordinate direction upon expansion back to j and i The suffixes equations (1) to (3). (S_{bj}), are buoyancy source or sink terms, while (t) is the time.

Time-Averaged Equation For Turbulent Flow :

The turbulent flow problems solve the time-dependent Navier-Stokes equations directly using numerical methods. The usual practice is to consider the time-averaged versions of the governing equations. This follows the assumption that the main flow variables such as the velocity components, etc, may be expressed as the sum of a time-averaged mean value and a fluctuating component about this mean value is referred to Patankar [12].

In this formulation, it is assumed that the time average of the fluctuating components is zero when these expressions are substituted into the aim governing equations and time-averaging see energy equation [13,14].

.

$$\rho[U\frac{\partial T}{\partial x}+V\frac{\partial T}{\partial y}+W\frac{\partial T}{\partial z}]=\frac{\partial}{\partial x}[\frac{\mu}{\sigma_T}\frac{\partial T}{\partial x}-\overline{\rho u' t'}]+\frac{\partial}{\partial y}[\frac{\mu}{\sigma_T}\frac{\partial T}{\partial y}-\overline{\rho v' t'}]$$
$$\frac{\partial}{\partial z}[\frac{\mu}{\sigma_T}\frac{\partial T}{\partial z}-\overline{\rho w' t'}] \quad \ldots(2)$$

Where:

U: is the time-average velocity in x-direction.

V: is the time-average velocity in y-direction.

W: is the time-average velocity in z-direction.

P: is the time-average static pressure.

T: is the time-average Temperature.

u' : is the fluctuating component value of u,

v' : is the fluctuating component value of v,

w' : is the fluctuating component value of w,and

t' : is the fluctuating value of T .

SST *k-omega* Turbulence Model :

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*. Since the air in this project's the flow from a number of diffusers placed close to each other, it is expected that the boundary layer flow has a strong influence on the results, and properly modelling this near-wall flow could be important for accuracy of the calculations. Therefore the SST *k-omega* turbulence model has also been chosen for computational fluid dynamics simulations in this project.

This 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:

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

(I) (II) (III) (IV) (V)

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

The terms **(I)-(V)** in Equation3, the turbulent kinetic energy *k* transport equation for the SST *k-omega* turbulence model, can be interpreted as the following:

(I)	Transient term	Accumulation of *k* (rate of change of *k*)
(II)	Convective transport	Transport of *k* by convection
(III)	Diffusive transport	Turbulent diffusion transport of *k*
(IV)	Production term	Rate of production of *k*
(V)	Dissipation	Rate of dissipation of *k*

σ_k and β^* are equation constants.

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

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

(I) (II) (III) (IV) (V) (VI)

where $\sigma_{\omega,1}$, γ_2, β_2, *and* $\sigma_{\omega,2}$ are constants.*)* …(4)

The general description for each of the terms in Equation 4 (I) to (V) are the usual terms for accumulation, convection, diffusion, production, and dissipation of ω. The last term (VI) is called a 'cross-diffusion' term, an additional source term, and has a role in the transition of the modelling from ε to ω.

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. There have also been limiting functions made to control the eddy viscosity in wake region and adverse pressure flows [12].

Geometry Of Room :

For this study, a room of dimension X=7m, y=8m ,andz=4.25m. The sketch of the room geometry shown in Figure 1.

- Enter *Operating Density* as *1.225* kg/m3
- Set a constant wall temperature of 310 K.
- *Cold flow Temperature* of *294K:*
- Cold flow rate 0.225 kg/s

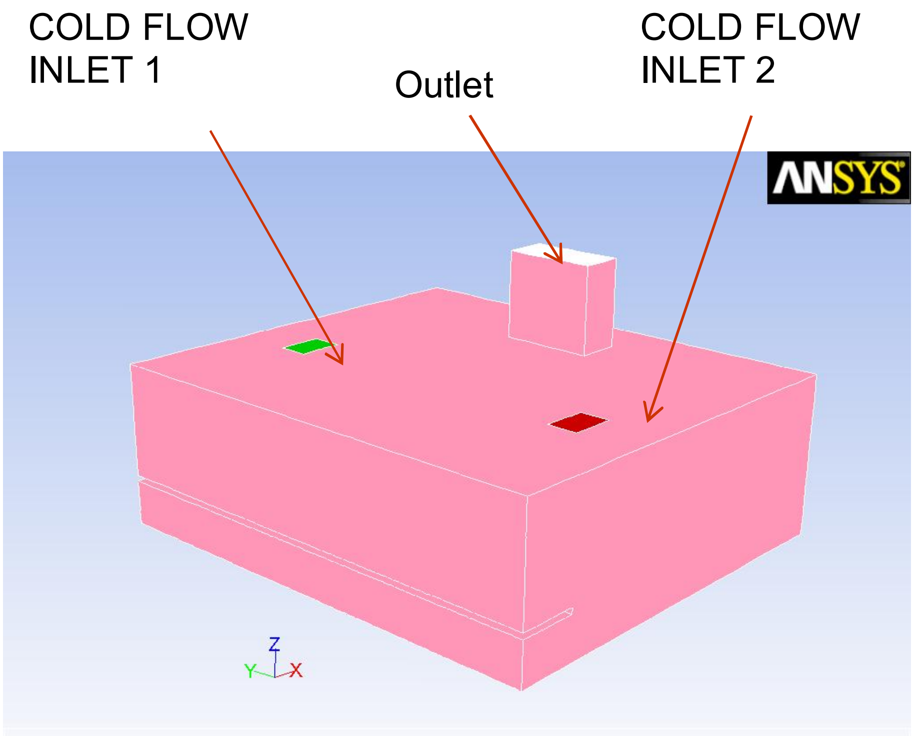


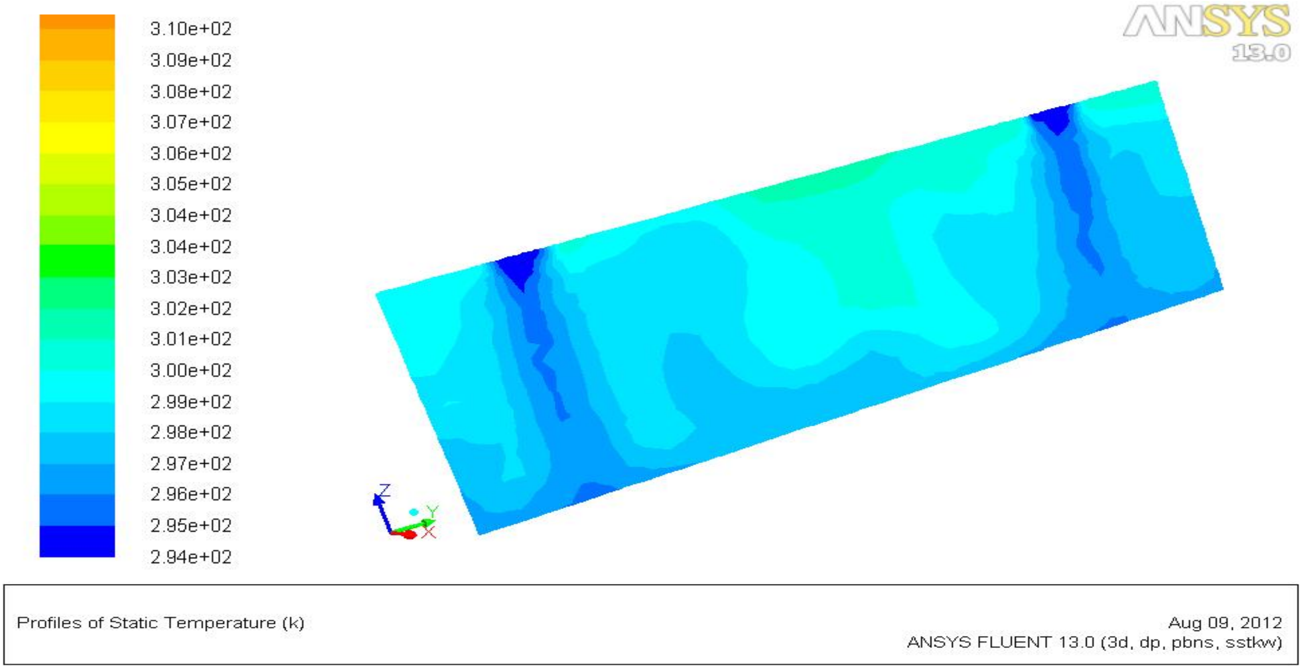
Figure 1 Three dimensional room geometry

Results And Discussions :

Numerical simulations results have been discussed and performed to simulate two a discrete inlet ceiling duct flow in the test geometry representative of the enclosed space The flow field is examined; key physical mechanisms, resulting from the interactions between the cooling flow and the room air are identified and their effect on the thermal field.

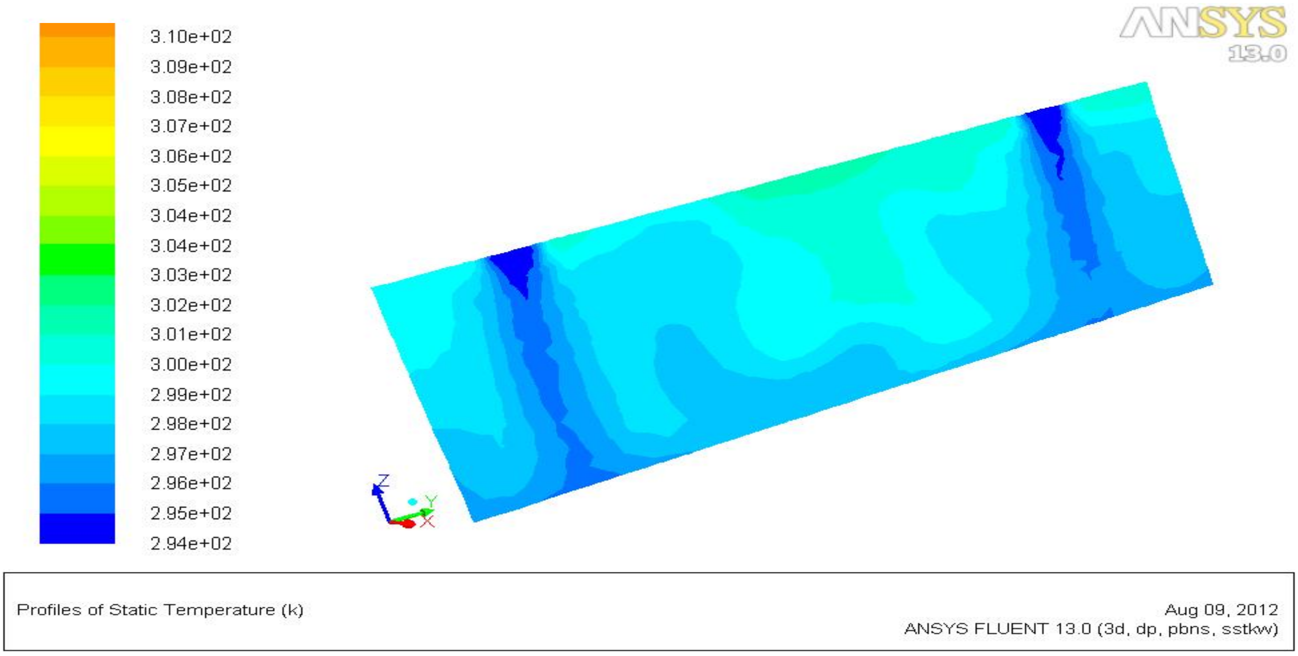
- ***The computed distribution of temperature contour:***

In order to understand the behavior of the inlet flow, several (Z-Y) planes were taken to give details about the effect of ceiling cold flow 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=2.5m from the wall, where the vertical temperature gradient (and vertical concentration gradient)shown below.



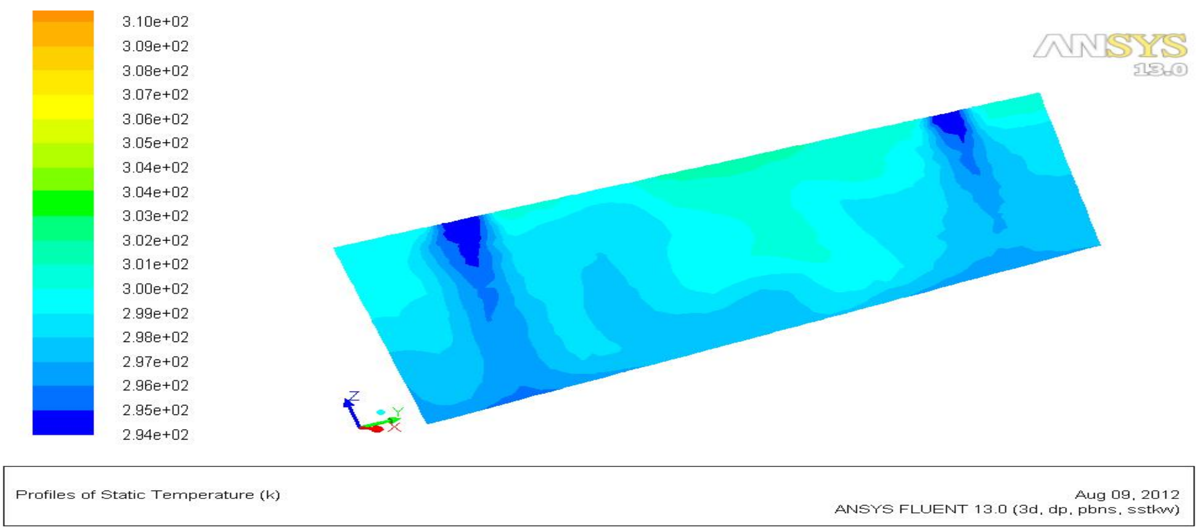
Figure(2) Temperature contours in (Z-Y)plane at X=2.5m

Figure(3) indicated that the best penetration of cold flow is for x=2.7m from the wall which represent the center of ceiling duct in which the penetration area for this case covers a very wide range .

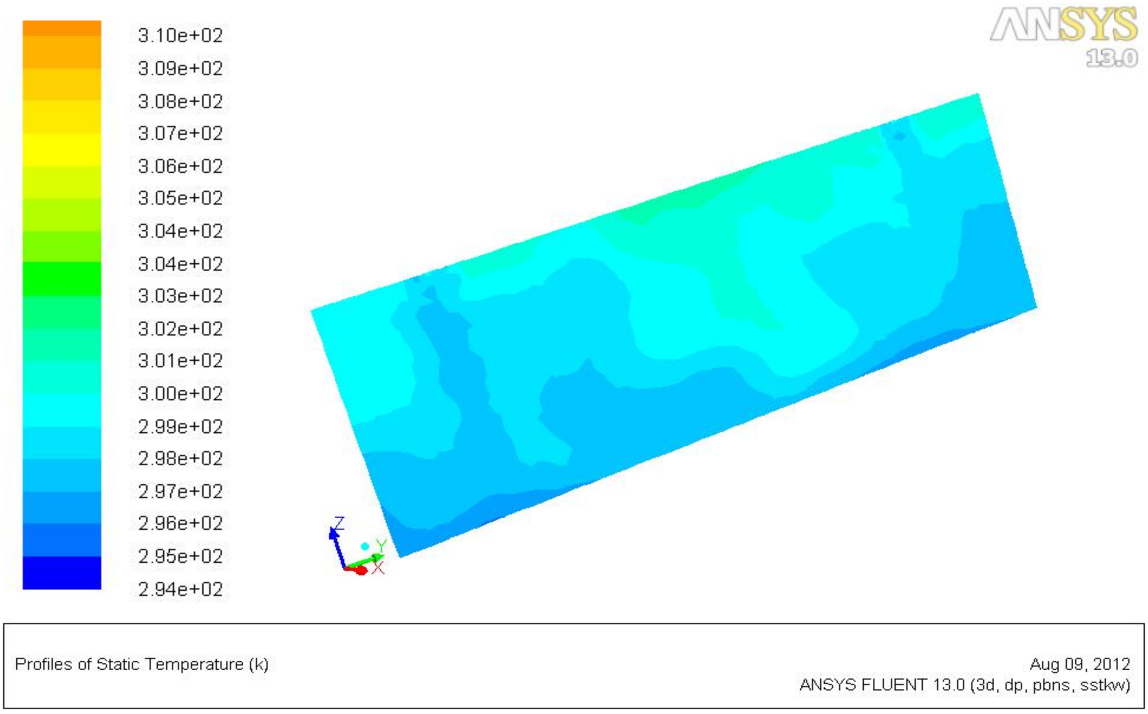


Figure(3) Temperature contours in (Z-Y)plane at X=2.7m

Figures (4&5) shows that the cold flow penetration effect decreases as we move far away from the center ceiling duct. In these figures air movement is controlled by buoyancy the flow from a number of diffusers placed close to each other will merge into a plane, stratified flow.

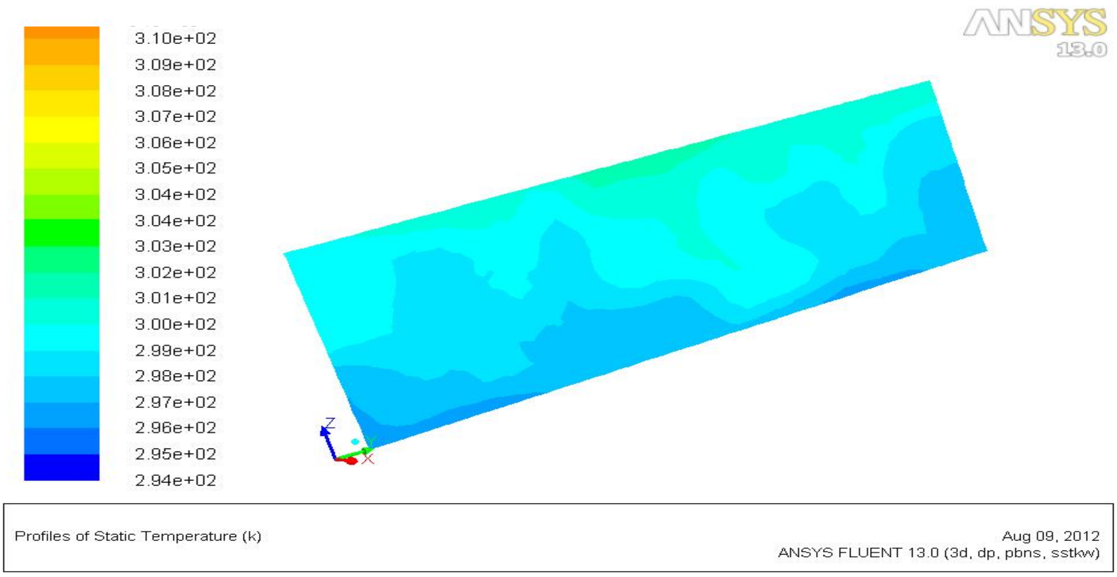


Figure(4) Temperature contours in (Z-Y)plane at X=2.9m



Figure(5) Temperature contours in (Z-Y)plane at X=3.1m

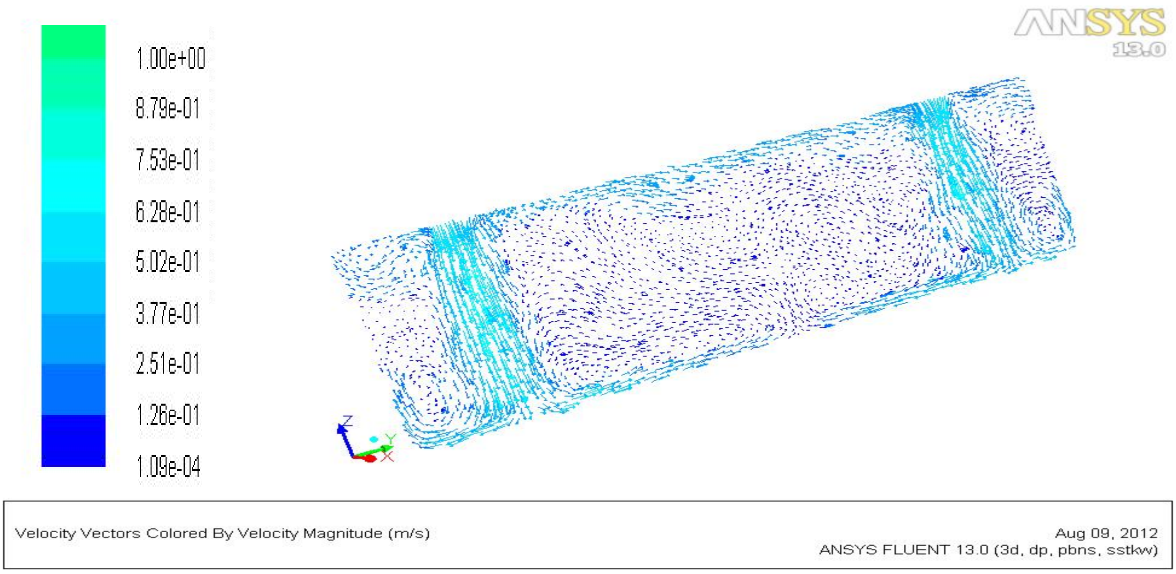
Figure (6) show that the cold flow penetration effect disappear at X=3.3m. In this figure the cconvective flows, is the engine of vertical temperature gradient (and vertical concentration gradient) the flow from a number of diffusers placed close to each other will merge into a plane, stratified flow.



Figure(6) Temperature contours in (Z-Y)plane at X=3.3m

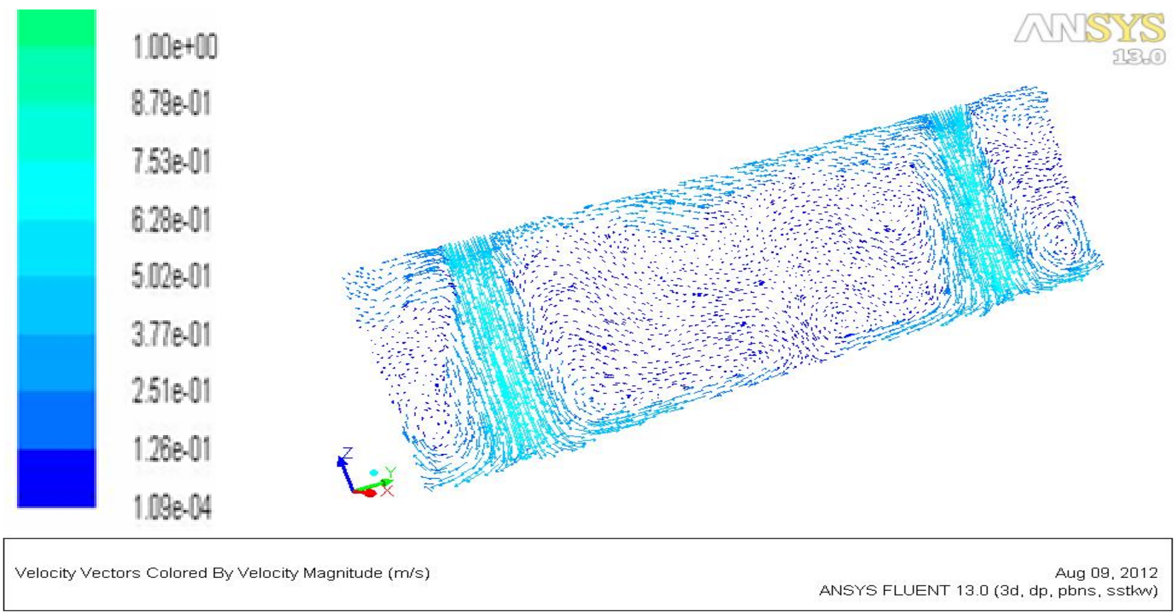
- ***The computed distribution of velocity vectors*** :

Figure (7) indicate the velocity vectors in (Z-Y)plane at X=2.5m which represent the edge of the duct .The distribution of velocity vectors are very clear in the region under the duct cold jet.



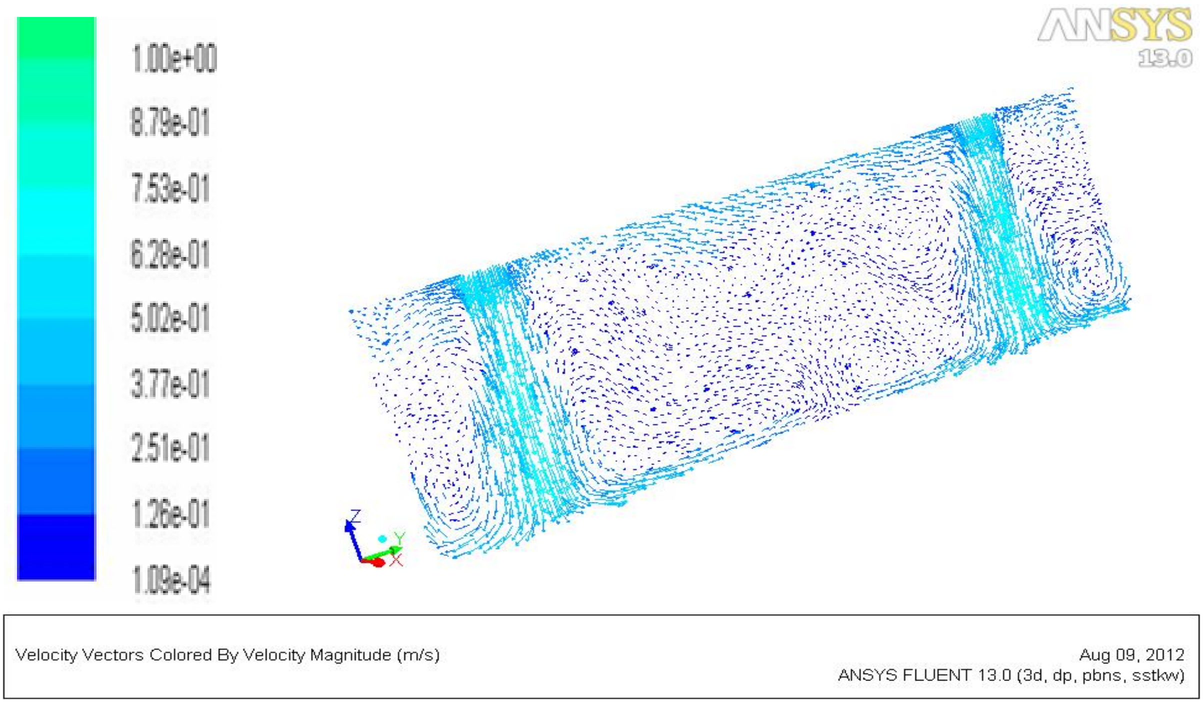
Figure(7) velocity vectors in (Z-Y)plane at X=2.5m

Figure (8) shown the velocity vectors in (Z-Y) plane at X=2.7m, which indicate the maximum penetration at the center of duct also this figure indicate the general trend of cold flow circulation.



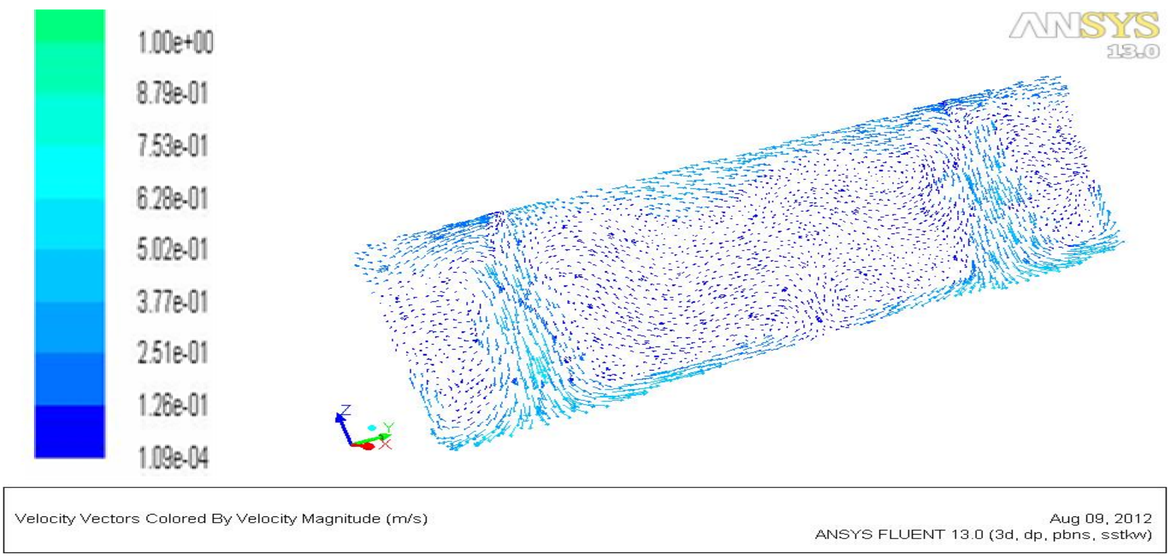
Figure(8) velocity vectors in (Z-Y)plane at X=2.7m

Figure (9) shows velocity vectors in (Z-Y) plane at X=2.9m in this region the velocity vectors under the ceiling duct still high

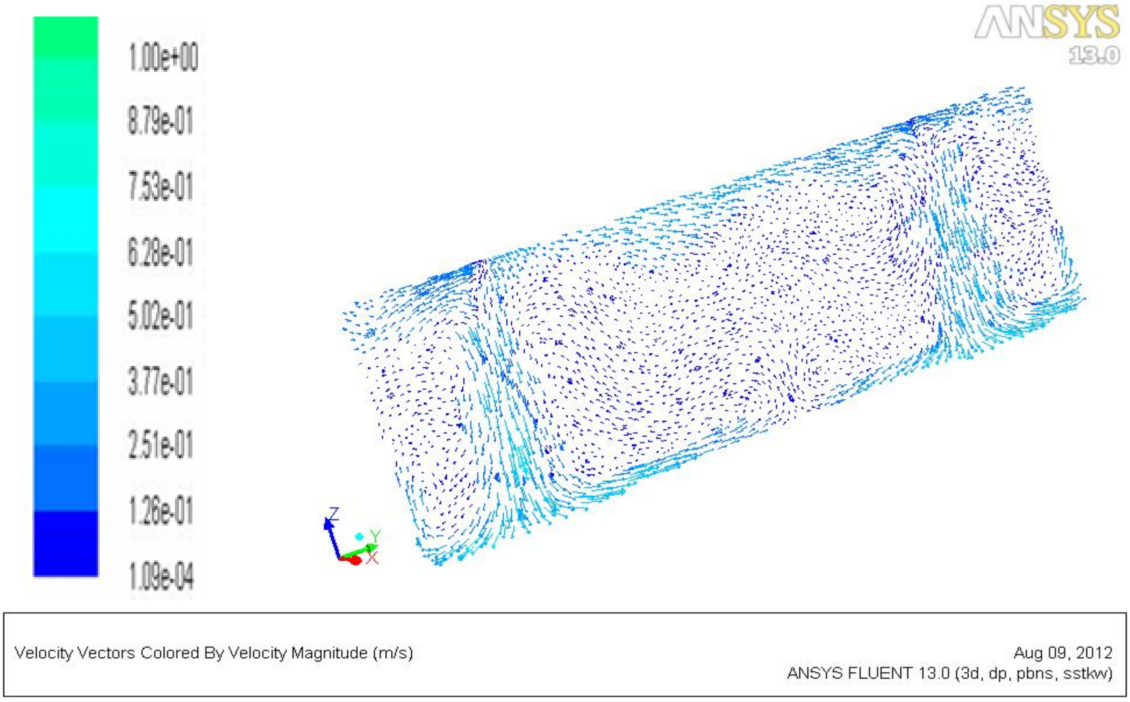


Figure(9) velocity vectors in (Z-Y)plane at X=2.9m

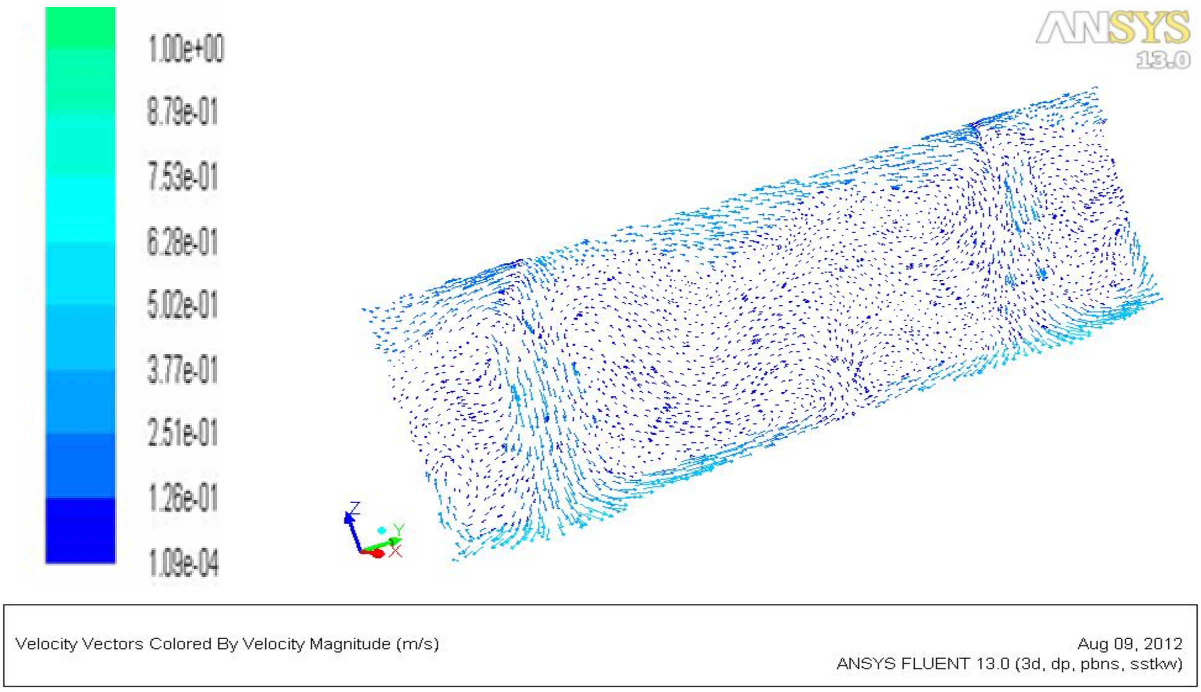
Figures(9,11&12) shows velocity vectors in (Z-Y)planes at X=3.1,3.3&3.5 .These planes indicate that the velocity vector reduced as we move forward from the jet region.



Figure(10) velocity vectors in (Z-Y)plane at X=3.1m



Figure(11) velocity vectors in (Z-Y)plane at X=3.3m



Figure(12) velocity vectors in (Z-Y)plane at X=3.5m

Conclusion :

In this work the effect of thecooling shower from the ceiling ducts is simulated. Numerical analysis is used to solve three dimensional steady flow 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 best penetration of cold flow is for x=2.7m from the wall which represent the center of the ceiling duct in which the penetration area for this case covers a very wide range.
- Displacement flow Controlled by buoyancy (temperature differences)
- Mixing flow Controlled by the momentum flow from supply openings.
- The velocity vector and temperature contours reduced as we move forward from the jet region.

References :

1- Lee, S.C., Cheng, C.Y., Chen, C.K., 1997, Finite elements solutions of laminar and turbulent flows with forced and mixed convection in an air-cooled room, Numerical Heat Transfer, Part A, 31, pp. 529–50.

2- Sinha, S. L., 2001, Behaviour of Inclined Jet on Room Cooling, Building and Environment 36, pp. 569-578.

3-Richtr, J., Katolicky, J., Jicha, M., 2001, Numerical modeling of the influence of angle adjustment of A/C diffuser vanes on thermal comfort in a computer room, Seventh International IBPSA Conference, Rio de Janeiro, Brazil, pp. 1135-1141.

4- Rutman, E., Fournier, F., Jurkowski, R., Bailly, A., Frankovic, B., Vilicic, I., 2002, Industrial Approach for CFD Modelling Applications for Air-conditioning and Heat Exchanger Systems, Energy and the Environment 2002, Vol. 2, Opatija, Croatia, pp.11-18

5- Zukowski, M., 2003, A Numerical Analisys of Heat and Mass Transfer in a Room with the Air-Underfloor Heating, 8th International IBPSA Conference, Bialystok Technical University, Poland.

6- ASHRAE ANSI/ASHRAE Standard 55-1992 Thermal environmental conditions for human occupancy, 1992.

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

12- Patankar S. V. , Basu, D. K. & Alpay, S. A., 1977, "Prediction of the three-dimensional velocity field of a deflected turbulent jet" Trans. SME I: J. Fluids Engng 99, pp. 758-762

13- Hussin H.M., 1998," Prediction of Buoyant Flow inside a Heated Enclosure", Msc. Thesis, Military College of engineering.

14-ASHRAE, 1997," Hand Book of Fundamental".