

A SIMULATION STUDY OF THE AERAULIC BEHAVIOR OF A HABITATION ROOM USING CFD APPROACH

Nedjmeddine KIMOUCHE, Zinelabiddine MAHRI

Laboratoire de Génie Climatique,
Université des Frères Mentouri Constantine, Algeria

Reçu le 12 Novembre 2015 – Accepté le 22 Janvier 2017

Résumé

Dans cette étude, nous allons nous intéresser à l'influence de la géométrie des ouvertures sur le comportement aéraulique d'un local d'habitation. Le choix des ouvertures pour aérer un local d'habitation représente un facteur essentiel pour assurer le confort thermique de ses occupants.

Dans ce travail, une simulation numérique du comportement aéraulique de l'environnement intérieur d'un local a été effectuée, en utilisant le logiciel Fluent.

L'objectif premier de cette étude demeure dans l'évaluation du confort thermique dans un local d'habitation, en comparant les résultats numériques obtenus et qui concerne un local aéré naturellement par différentes géométries des ouvertures d'entrée et de sortie de l'air.

Les résultats de cette simulation, prouvent que l'utilisation de deux ouvertures de sortie de l'air assure un meilleur confort thermique.

Mots clés : *Ventilation naturelle, méthode CFD, comportement aéraulique, confort thermique*

Abstract

In this study, the influence of openings on the aeraulic behavior in a room of habitation is investigated. The choice of openings to ventilate a room is an essential factor to ensure the comfort of the occupants.

In this work, a numerical simulation of the aeraulic behavior of the interior environment of the building is carried out using the FLUENT software.

The main objective of this work is the evaluation of thermal comfort in a room by comparing the numerical results obtained for a room ventilated naturally, by changing positions of openings used as inlet and outlet for air flow.

The results show that the use of two outlet openings ensures a better thermal comfort than the use just one opening even if the total surface is kept the same.

Key words: *natural ventilation, CFD, aeraulic behavior, thermal comfort.*

ملخص

تهتم هذه الدراسة بتأثير الشكل الهندسي لفتحات التهوية على الرفاهية الحرارية الواجب توفرها. تعد عملية اختيار فتحات التهوية عاملاً أساسياً لضمان الرفاهية الحرارية للسكان.

في عملنا هذا، تم إجراء محاكاة رقمية للسلوك الحراري داخل بيئة داخلية لمسكن معين، وذلك باستخدام برمجية Fluent. إن الهدف الأساسي لهذه الدراسة، يتمثل في تقييم الرفاهية الحرارية في مسكن ذو تهوية طبيعية، عن طريق إجراء مقارنة بين أشكال مختلفة لفتحات التهوية في هذا المسكن.

نتائج هذه المحاكاة أبرزت أن استخدام فتحتين لخروج الهواء يوفر أكبر قدر من الرفاهية الحرارية.

الكلمات المفتاحية :

التهوية الطبيعية-طريقة CFD-السلوك الحراري – الرفاهية الحرارية

الكلمات المفتاحية : *التهوية الطبيعية-طريقة CFD-السلوك الحراري – الرفاهية الحرارية*

The essential function of a residence is to ensure an interior environment well adapted to our needs and our comfort. The inhabitant often places his comfort before energy saving. Despite the importance of energy saving in building usage, it is thus worthwhile to design buildings and installations that consume less energy while ensuring a suitable comfort. Several studies were interested in natural ventilation [1-5].

Hee-Jin Park has performed a two-dimensional computation to examine the effect of vertical location of a convective heat source on thermal ventilation systems. The temperature distribution, air flow pattern and the contaminant dispersion have been simulated numerically and experimentally [6]. The study placed an emphasis on the health risk of the air borne bacteria released from the surgical team on the patient, and vice versa. An evaluation of the current thermal comfort conditions of an air-conditioned lecture theatre in a tertiary institution using objective measurement, computational fluid dynamics (CFD) modeling and subjective assessment was carried out [7].

Other studies have investigated the distribution of velocity in an enclosed room in the case of natural convection [8], where the main objective of these studies was the prediction of the distribution of temperature in a room cooled by the ceiling and heated partially by one side, to analyze and check the conditions of thermal comfort according to the position of the heat source for different values for the number of Rayleigh

In this study, we are interested in visualizing, the profile of temperature and air velocity, in turbulent mode of a naturally ventilated room using computational fluid dynamic (CFD). This approach was developed in 1970 by Y.L.Ong, since it constitutes one of the most efficient numerical tools and it has been validated by confronting its results to measurement [9]. In this simulation the FLUENT software has been used.

A uniform distribution of air temperature and air velocity can guaranty a good thermal comfort. An important fluctuation in the values of both velocity and temperature of the air inside the room is not desirable. The essential goal of this simulation is to evaluate the influence of the position and the surface of the openings on the air flow.

1. GOVERNING EQUATIONS

The movement of a fluid can be described by the following equations: continuity equation, Navier-Stockes and the energy equation (4). However, in our study these hypotheses have been taken into account:

- The fluid (air in our case) is Newtonian.
- Air movement obeys Boussinesq approximation.
- The turbulence is isotropic.
- A steady state air flow is considered.
- The viscous dissipation in the energy equation is nil.
- The magnetic field is nil.
- The Coriolis force is nil.

Continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

Navier Stockes equation:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} \quad (2)$$

Energy equation

$$\frac{\partial(C_p T)}{\partial t} + u_j \frac{\partial(C_p T)}{\partial x_j} = \frac{\nu}{2} C_p \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)^2 + \frac{\lambda}{\rho} \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{1}{\rho} \left(\frac{\partial p}{\partial t} + u_j \frac{\partial p}{\partial x_j} \right) \quad (3)$$

2. APPROXIMATION OF BOUSSINESQ

In natural convection of thermal origin, the movement of the fluid is due to the variation of density which is related to the variation of temperature. When this variation remains weak, the thermo-physical properties of the fluid (like dynamic viscosity, thermal conductivity and the specific heat C_p) are supposed to be constant, except for the density, where the variation is supposed to be linear and given by the glowing Boussinesq relation:

$$\rho(T) = \rho(T_0) [1 - \beta(T - T_0)] \quad (4)$$

With:

$$\beta = -\frac{1}{V} \left(\frac{\partial V}{\partial T} \right) \quad (5)$$

Where:

β : is the thermal coefficient of expansion at constant pressure, it describes the change of volume at constant pressure with respect to the temperature.

ρ_0 : Is the density at the temperature of reference T_0 .

ρ : The density (kg/m^3), C_p : the specific heat.

3. TURBULENCE MODEL

The standard K-epsilon engineering model is the most used to calculate industrial airflow, it calculate two variables: the turbulent kinetic energy (k) and the turbulent dissipation rate (epsilon).

4. NUMERICAL SIMULATION AND GEOMETRY CREATION

A CFD method is used to predict the distribution of the temperature and velocity of the air in the room. This approach has a good accuracy and a reasonable calculation cost.

Fluent is one of the specialized CFD software that focuses on solving airflow and problems in building environment, this CFD solver package was used to perform all the CFD computations. The solutions were obtained by iteration process; the computation is completed when the convergence of results has reached the prescribed criteria.

For the geometry and meshing we have used Gambit, Since the occupation zone in a room is less than the global zone (all volume of the room), we have not taken care of the boundary layer near the walls, and so we have used a quadratic mesh ($x=y$) that allowed us to reach convergence and good results.

4.1. Description of the room

In order to validate our mathematical model and results obtained with the software FLUENT, we have compared it with the results of Liping Wang et al [10] work. Figures (1, 2) represent velocity vectors in the section $y= 1.5$ for the two studies.

The two figures show a good agreement between our study and the study of CHEN QINGYAN, therefore the simulation that we carried out using FLUENT software, has allowed us to obtain good results.

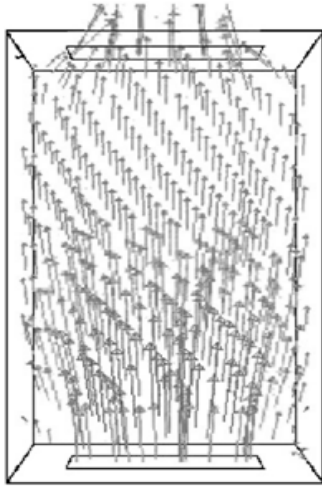


Figure 1 : velocity vectors for CHEN QINGYAN et al

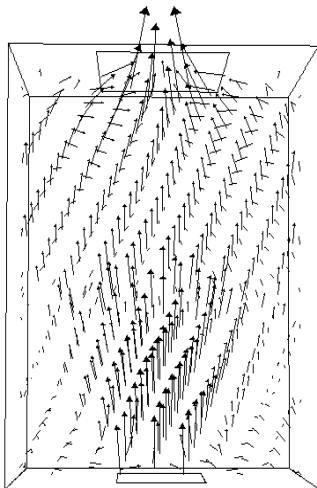


Figure 2 : velocity vectors for our study

5. FIRST STUDY SIMULATION

5.1. Description of the room

We have considered a room of interior size “5x4x3”m³, with two openings, one for the air inlet and the

second for the exit. The inlet has a fixed surface, and the outlet has a varying geometry. In the first case, the inlet and the exit of the air are located opposite to each other.

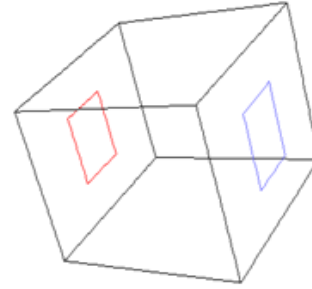


Figure 3 : geometry of case1

In the second case, we keep the same conditions as the first case with variation of dimensions of the exit.

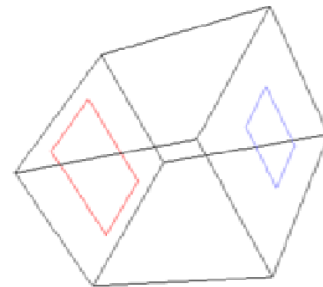


Figure 4 : geometry of case2

In the third case, we use two openings for the exit while keeping the same total surface.

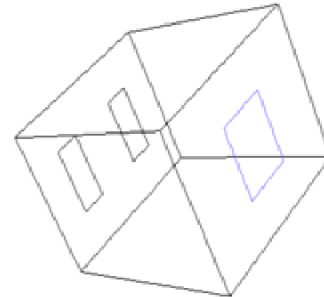


Figure 5 : geometry of Case3

5.2. Boundary conditions

The following boundary conditions are considered:

- ❖ Entry: inlet velocity 2m/s,
- ❖ Exit: outlet condition
- ❖ Walls:
 - Right T=300K,
 - Face T=305K,
 - Left Y=300K,
 - Ceiling 300K,
 - Ground 300K,
 - Back 300K
- ❖ Turbulent flow: approximation of Boussinesq
- ❖ Fluid: air
- ❖ Density: approximation of Boussinesq

These values are chosen arbitrary in order to allow us to predict the aeraulic behavior of the room.

5.3. Results

The results presented in figures 6, 7 and 8, figure 9 represent the distribution of the temperature for the three studied cases; for a good representation of this distribution, three levels have been taken in consideration, low level, middle level and high level. Also for the horizontal distribution, the comparison was made between the three cases on a same level, for this purpose, a middle level line has been created.

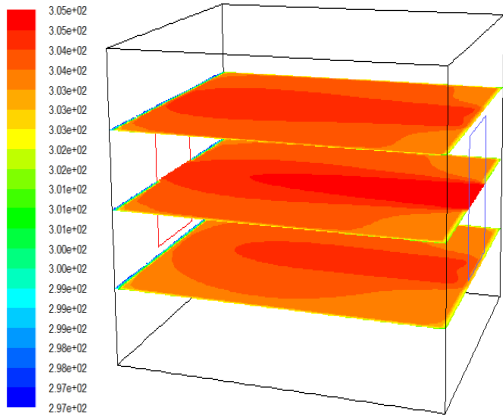


Figure 6 : Case 1: distribution of temperature in the room

For the first case where the inlet and the exit have the same surface, the dominant temperature is about 304K.

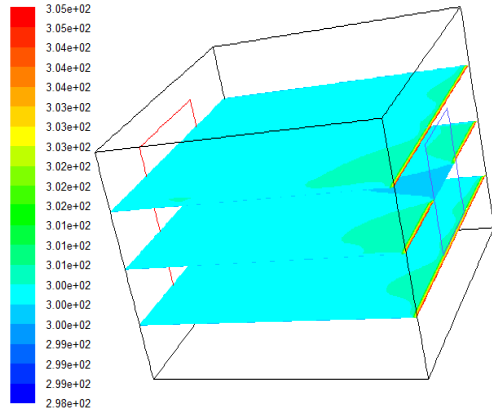


Figure 7 : Case 2: distribution of temperature in the room

For the second case, where the outlet opening is twice as large as the inlet, the dominant temperature is about 300k.

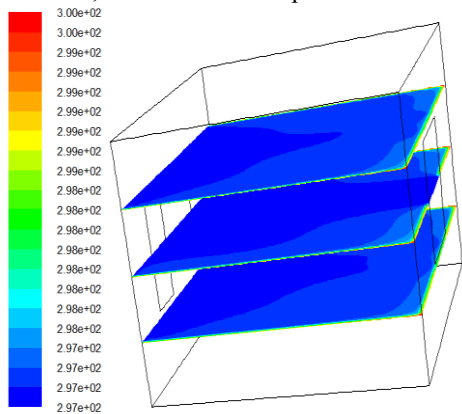


Figure 8 : Case 3: distribution of temperature in the room

For the third case, where the opening of exit is replaced by two openings, this temperature is estimated to be 297K.

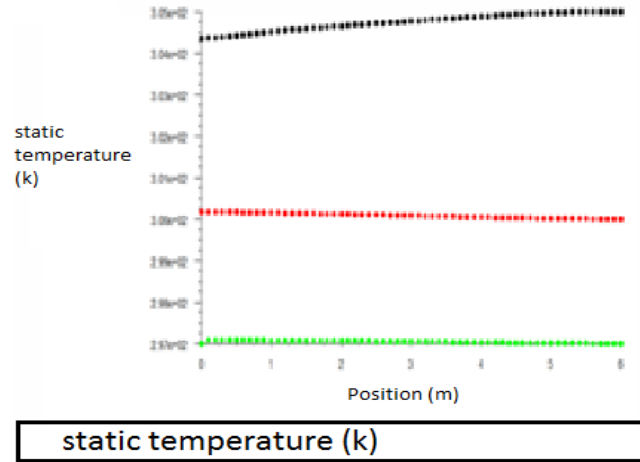


Figure 9 : comparison between the 3 cases: lines of temperature

The Figures 10, 11, 12 and 13 present the distribution of velocity in the room for the three cases, the figures show that there is no big different between the value of velocity for the three cases. However, the third case still has larger zones of thermal comfort. Consequently, it is obvious that the best geometry adapted to ensure good ventilation and therefore good comfort of the occupants is that chosen by the third case.

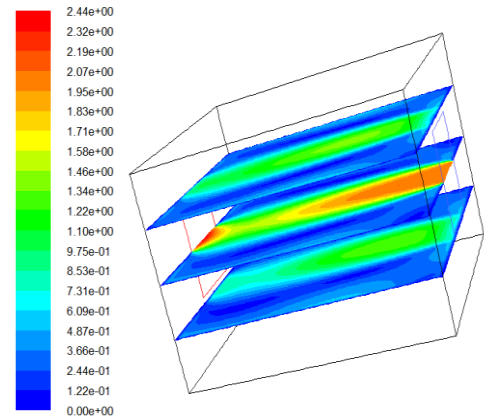


Figure 10 : Case 1: distribution of velocity in the room

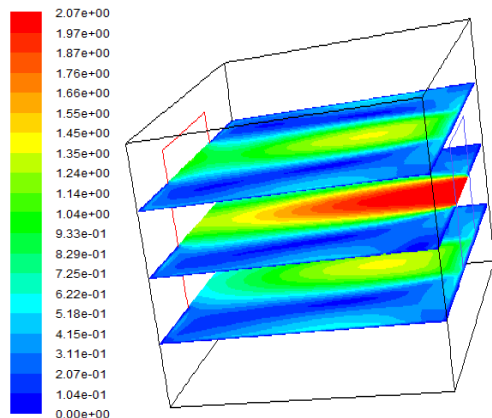


Figure 11 : Case 1: distribution of velocity in the room

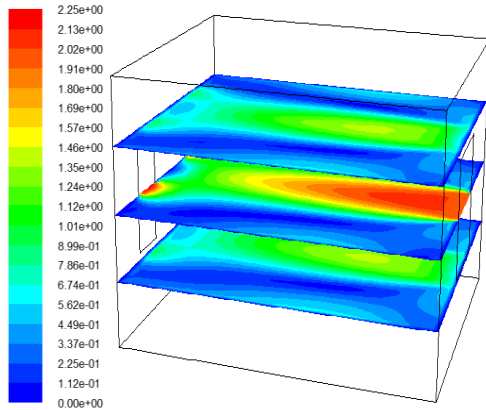


Figure 12 : Case 3: distribution of velocity in the room

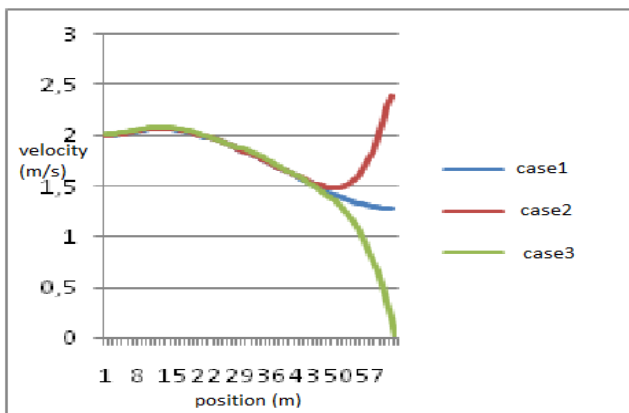


Figure 13: comparison between the 3 cases: lines of velocity

6. SECOND SIMULATION STUDY

In this part of our paper, we have studied the influence of the position and the surface of the outlet openings on the aeraulic behavior of the room.

Three cases have been chosen where we have changed the position of exits from the bottom of the wall to the top, also we have changed the surfaces, using a large surface in the first case and in the second case we made it smaller.

Figures 14, 15 and 16 represent respectively the three geometries that we have considered

6.1. Boundary conditions

For the three cases, the boundary conditions are the same:

- ❖ Bottom: $T=297K$,
- ❖ Ceiling: $T=303K$,
- ❖ East: $T=300K$,
- ❖ North: $T=298K$,
- ❖ South: $T=303K$,
- ❖ West: $T=303$.
- ❖ Inlet Velocity: $V=2m/s$,
- ❖ $T=303K$

These values are chosen arbitrary in order to allow us to predict the aeraulic behavior of the room.

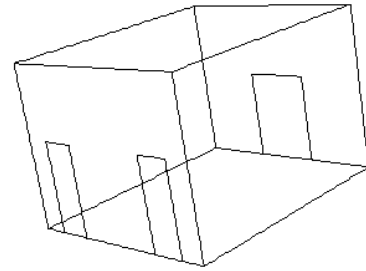


Figure 14: Two big exits

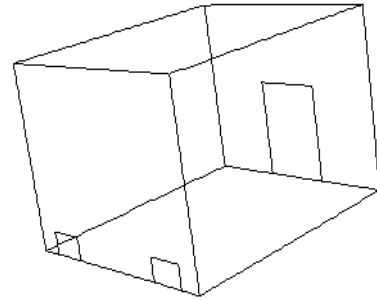


Figure 15: Two small exits on the bottom

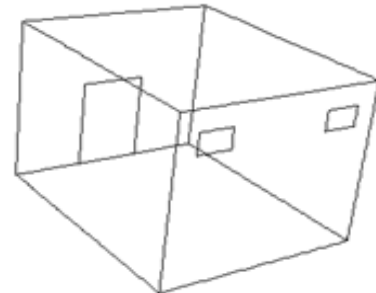


Figure 16: Two small exits on the top

6.2. Results

The figures represent the distribution of velocity and the temperature of the air for the three cases:

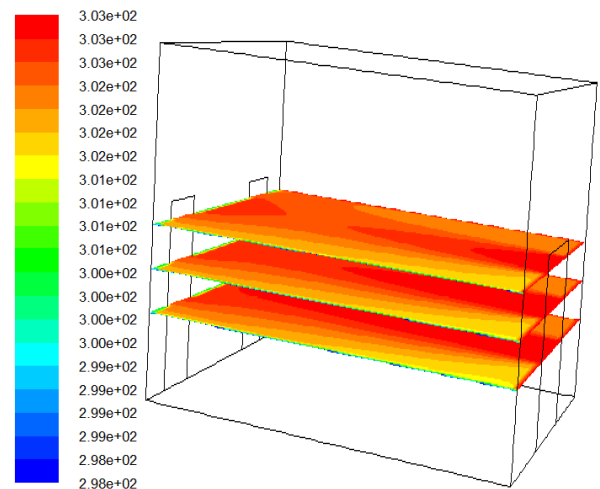


Figure 17: case1: distribution of Temperature in the room

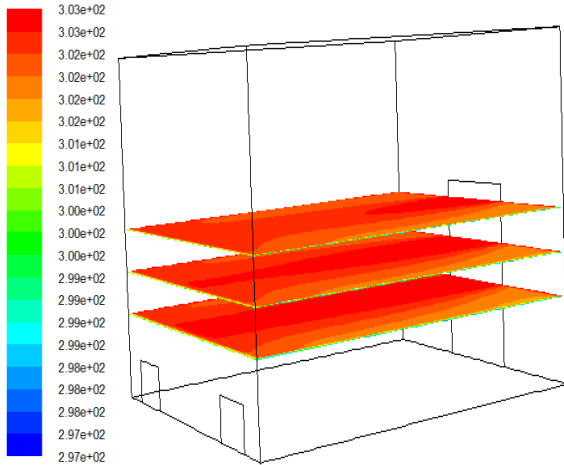


Figure 18: case2: distribution of temperature in the room

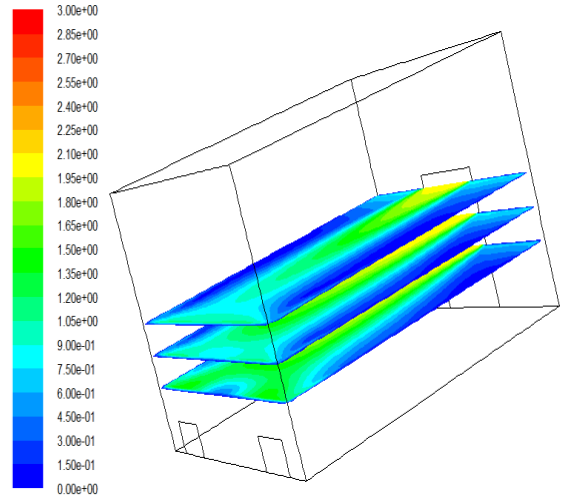


Figure 21: case2: distribution of the velocity in the room

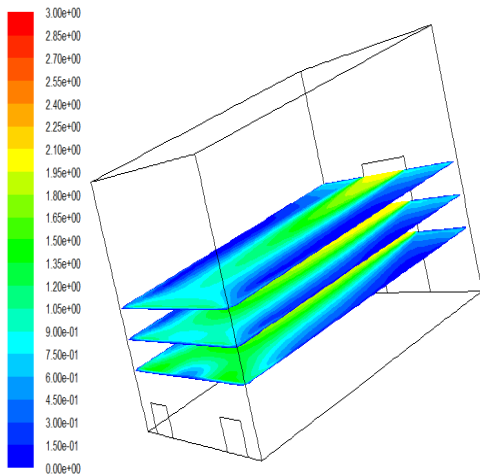


Figure 19: case2: distribution of the velocity in the room

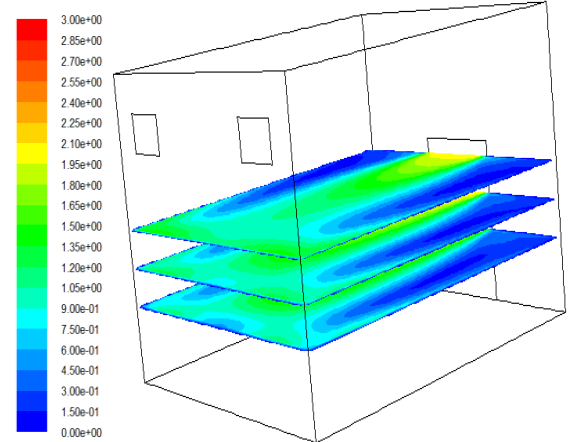


Figure 22: distribution of the velocity in the room

The figures 17, 18 and 19 represent the distribution of the temperature in the room for the three cases, these results show that for the three cases, the distribution was the same, we notice that the temperature has a value of 302K for the all cases with a small difference for the first case, where the exits are large, for this case we see some zones that have 301K.

The figures 20, 21 and 22 represent the distribution of velocity in the room for the three cases. These results show that the first case, where the exits have a big surface, gives a better distribution since the value of the velocity is situated in the range of 0.8 m/s-1.67 m/s in the most zones of the room; while the values of the velocity for the two other cases, where the exits are smaller, many zones with a value of 0 m/s show up. These zones called “dead zones” or “zones of discomfort”.

CONCLUSION AND FURTHER WORK

In this study, three geometries of naturally ventilate room were studied. The objective of this study was to compare the results obtained by these different geometries in order to achieve good thermal comfort.

The thermo-aerodynamic Simulation (study of the profile of temperature and air velocity in the room) was carried out using the FLUENT software. The numerical results enabled us to predict the thermal comfort by comparing three different cases. This simulation has shown that the room provided with two openings at the air exit generates a better thermal comfort compared to the two

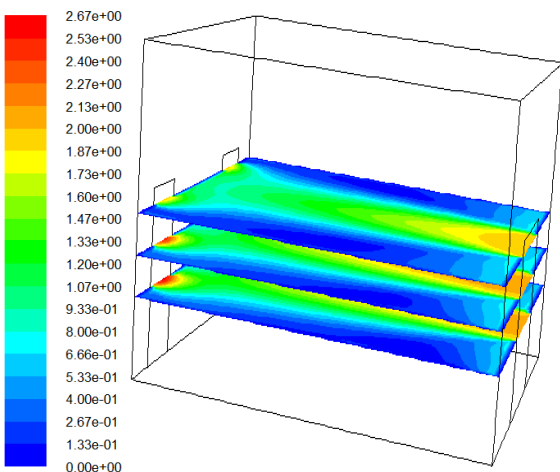


Figure 20: case1: distribution of the velocity in the room

other cases; where in the first case the exit has the same surface as the entry or in the second case where a larger surface is used at the exit.

From the results of this study, we can tell that the surface of openings has an important influence on the aerualic behavior of the room, we can see from both results, of temperature and velocity distribution, that the distribution is more interesting when the exits are big, The number of exits and their surfaces has an important influence on the thermal comfort in building. Every thermal study must take in consideration the numbers and the position of the exits of air flow. Knowing these two parameters allow us to reduce our need of energy.

For the future, we plan to pursue other studies that will allow us to find the ideal surface and number of openings that ensure the ideal comfort. We will also include in the study the humidity distribution in rooms of habitation.

NOMENCLATURE

- ρ Density (kg/m^3)
 C_p Specific heat ($W/kg.^\circ K$)
 B Thermal coefficient of expansion
 λ Thermal conductivity ($W/m^\circ K$)
 T Temperature ($^\circ K$)
 ν Dynamic viscosity ($kg/m.s$)
 u_i Air velocity (m/s)
 p Pressure (Pa)

REFERENCES

- [1] Guangyu.Cao, indoor air flow prediction by means of computational fluid dynamics, rapport scientifique; laboratoire de " heating, ventilating and air condition"; HELSINKI university of technology, (27/03/2006).
- [2] ZhiqiangZhai, Qingyan Chen 2001: Strategies For Coupling Energy Simulation And Computational Fluid Dynamic
- [3] Hee-Jin Park, Dale Holland (2001), The effect of location of a convective heat source on displacement ventilation: CFD study, Building and Environment 36 (2001) 883–889, Minneapolis, MN 55437-1075, USA
- [4] KjeldSvidt, Bjarne Bjerg, Thomas Dorf Nielsen (2001): Initial studies On Virtual Reality Visualization of 3D Airflow In Ventilated Livestock Buildings.
- [6] Tin-Tai Chow, Xiao-Yu Yang 2003 : Performance of ventilation system in a non-standard operating room.
- [5] Qingyan Chen, Ventilation performance prediction for buildings: A method overview and recent applications, School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, 2009.USA.
- [7] K.W.D. Cheong E. Djunaedy, Y.L. Chua, K.W. Tham, S.C. Sekhar, N.H. Wong, M.B. Ullah: (2001) thermal comfort study of an air-conditioned lecture theatre in the tropics. Building and environment (2001) .
- [8] K.Bennaser, R.Chouikh, C.Kerkeni, A.Guizani, 2006, Numerical study of the natural convection in cavity heated from the lower corner and cooled from the ceiling. Applied thermal engineering (2006) . vol26.Pages (772-775).
- [9] J. -J. A. A. Akoua, F. Allard, C. Beghein, B. Collignan: Experimental and numerical studies on indoor air quality in a real environment 2005
- [10] Liping Wang, Nyuk Hien Wong, 2008: Coupled simulations for naturally ventilated rooms between building simulation (BS) and computational fluid dynamics (CFD) for better prediction of indoor thermal environment.