NUMERICAL SIMULATION OF AIR TEMPERATURE AND VELOCITY IN A NATURALLY VENTILATED OFFICE

S. Shodiya*, M. B. Oumarou, G. M. Ngala and A. B. Muhammed
(Department of Mechanical Engineering, Faculty of Engineering, University of Maiduguri, Maiduguri, Borno State, Nigeria)

Corresponding author’s e-mail address: sulaimonshodiya@yahoo.com, Tel: +234(0)8169108486

Abstract
This paper presents a numerical simulation of air velocity and air temperature distribution in an office room of Computer Engineering Department of University of Maiduguri which is naturally ventilated. The office room under investigation with the dimension 5 m × 5 m × 4 m has a door in the East direction, and two windows, one in the East direction and the other in the South direction. For cost effectiveness, numerical solutions of steady-state airflow and heat transfer were done using a complete two-dimensional model. The results showed that the windows and the door could not undertake indoor heat load that can make the occupants to be thermally comfortable. In activity area where people sit and stand, the air velocity is moderate, this is about 0.98 m/s on the average. In addition, the temperature in this area is relatively high of about 302 K (29 °C) on the average. Based on the American Society of Heating, Refrigeration and Air-Conditioning Engineers (ASHRAE) standard for comfort environment in summer (air temperature: 293–299 K (20–26 °C); air velocity: 0.5–0.8 m/s), the natural ventilation for the office room cannot give a thermal comfort for the inhabitant of the room. However, a window, if installed opposite the door could improve the ventilation of the office.

Keywords: Ventilation, Modeling, Numerical simulation, Air-Velocity, Air-Temperature.

1. Introduction
Natural ventilation is a sustainable solution of retaining comfortable and healthy environmental conditions in buildings. Nevertheless, effective design, operation and construction of naturally ventilated buildings need a good technical knowledge of complex airflow patterns (air velocity) caused by the buoyancy, wind effects and air temperature distribution, working and studying, leisure activities takes place indoors. Therefore, it is important to offer healthy and comfortable conditions in buildings. As a result of energy crisis that happened in the 1970’s which has continued in the last few decades, many researchers have carried out studies in the indoor environment field to get a balance in the controversial relationship between air distribution and indoor air quality, thermal comfort and energy efficiency (Melikov et al., 2012). Heating, ventilation and air conditioning (HVAC) systems consumed almost half of the energy available in the buildings (Pérez-Lombard et al., 2008).

Well-designed building ventilation systems will give fresh and remove sour air, in order to provide a satisfactory indoor air quality into the building (Magdalena et al., 2013). The American Society of Heating, Refrigeration and Air-Conditioning Engineers (ASHRAE-55, 2013) states that the comfort zone for summer conditions needs the air temperature to be between 293 K and 299 K and air velocity between 0.5 and 0.8 m/s. The occupants’ comfort is strongly influenced by the ventilation systems such as the temperature differences or excessive
air movement. During building design and operation, reducing building energy use, adequate ventilation system should be provided (Magdalena et al., 2013).

Natural ventilation is gradually seen as a sustainable answer of maintaining healthy and comfortable environmental conditions in buildings. In natural ventilation systems, the airflow is driven through ventilation openings by naturally driving forces of wind (wind effect) and temperature (buoyancy effect). Although, buildings energy consumption is reduced if natural ventilation is used, however, there are many obstacles for its implementation. An efficient design of a naturally ventilated room requires a good understanding of complex airflow patterns caused by buoyancy and wind effects. As a result, without the sufficient instruments to predict and optimize environments in naturally ventilated buildings, the supply of healthy and comfortable conditions for occupants cannot be achieved. Design engineers must consider many factors when proposing natural ventilation. Most of these factors are not very important in mechanically ventilated buildings e.g. restrictions regarding outdoor conditions, such as location, wind patterns, air quality and noise) (Noe, et al., 2010).

At the design stage, design engineers often do not consider natural ventilation solutions probably as a result of lack of expertise to implement it. Even, where natural ventilation is successfully applied in the design, there is a danger of harming the project during the construction stage. This may be caused by contractors who do not fully realize or understand natural ventilation and make unilateral changes that can significantly affect the system (Mishra and Ramgopal, 2014). Thus, this lack of knowledge in the industry may result in poorly designed, constructed and operated naturally ventilated buildings.

There exists a major challenge to the present day engineers to create quality simulation models that can predict the performance of naturally ventilated spaces and show compliance with environmental requirements. Therefore, it is necessary to have simulation tools that can enable acceptable thermal performance of naturally ventilated buildings, through the design and optimization stages (John et al., 2011). The objectives of this investigation were to develop a simulation tool for thermal performance of naturally ventilated building and to evaluate the thermal comfort of an office room using natural ventilation. The paper used FLUENT CFD software package - FLUENT 6.3.26 (Anonymous, 2006).

2. Methodology
2.1 Description of CFD Model

The office shown in Figure 1 is a schematic diagram of an office room. The office room selected for the investigation was Computer Engineering Department of University of Maiduguri with two windows and a door; air distribution and thermal comfort in this room was analyzed and evaluated by using commercial Computational Fluid Dynamics (CFD) software (FLUENT). Maiduguri is located at latitude and longitude 11°50’N and 13°09’E respectively, belonging to subtropical climate (Shodiya et al., 2016).

Physical model to be calculated in the office room with two windows and a door in summer is shown in Table 1.
Figure 1: Schematic diagram of an office room

Table 1: The model parameters

<table>
<thead>
<tr>
<th>Name</th>
<th>Number</th>
<th>Size (Xm × Ym × Zm)</th>
<th>Model Type</th>
<th>Parameter value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Office</td>
<td>1</td>
<td>5 × 5 × 4</td>
<td>Room</td>
<td>-</td>
</tr>
<tr>
<td>East Window</td>
<td>1</td>
<td>1.2 × 0 × 1.2</td>
<td>Wall</td>
<td>-</td>
</tr>
<tr>
<td>South Window</td>
<td>1</td>
<td>1.2 × 0 × 1.2</td>
<td>Wall</td>
<td>-</td>
</tr>
<tr>
<td>Door</td>
<td>1</td>
<td>2.1 × 0 × 0.9</td>
<td>Opening</td>
<td>-</td>
</tr>
<tr>
<td>Persons</td>
<td>3</td>
<td>0.3 × 0.4 × 1.2</td>
<td>Persons</td>
<td>70 W/person</td>
</tr>
<tr>
<td>Computer</td>
<td>1</td>
<td>0.4 × 0.4 × 0.4</td>
<td>Blocks</td>
<td>250 W</td>
</tr>
<tr>
<td>Bulb</td>
<td>2</td>
<td>0.2 × 0.2 × 0.2</td>
<td>Blocks</td>
<td>20 W/bulb</td>
</tr>
</tbody>
</table>

2.2 Mathematical Model
Air flow in the ventilated Office room is an incompressible fluid flow. This flow must satisfy the basic conservation laws namely: conservation of mass, energy and momentum. Likewise, the conservation equations of the components in the room must also be satisfied.

2.3 Boundary Conditions
(i) Outdoor parameters: dry-bulb temperature at 33 °C; air velocity: 1 m/s (NIMET, 2014).
(ii) Indoor parameters: dry-bulb temperature at 27 °C, relative humidity of 40%.
(iii) External wall: South wall only was taken to simplify the calculation and heat transfer was assumed as constant heat flux boundary condition with heat transfer density being 16.8 W/m².
Outside window: on the external wall, heat transfer density was 16.8 W/m$^2$.

(v) Interior wall and floor can be regarded as adiabatic boundary condition without temperature difference.

The partial differential equations for the conservation of mass, momentum and energy were used. The paper also use the following assumptions (Zhang et al., 2004):

i. The indoor air is incompressible

ii. The indoor air at the entrances is uniform

iii. The indoor air pollutant quality is zero

The simulation was done along sectional plane of 0.5 m away from the South, West and North facing walls.

2.4 Numerical Method

The CFD code of the Fluent was used to simulate the temperature and velocity field of the office room. Based on the essential principle of solving the governing equations numerically, the temperature and velocity were first discretized (a tetrahedral mesh of about 37,500 cells) represented by a collection of variable value of series finite discrete points. From these mesh, numerical models was formed and the indoor air was considered to be incompressible and the entrance uniform. The first-order upwind differencing scheme and the SIMPLE algorithm was chosen to solve the resulting equations because it is widely used in thermal comfort and Air-conditioning system modeling and simulation. The Fluent software has also prove to be a powerful tool for simulating air temperature, velocity and humidity for thermal comfort purposes.

3. Results and Discussion

Fluent CFD software was used to simulate air distribution of velocity field and temperature field in the office room. Through the general assessment and analysis, the global design of the room is evaluated. Therefore, the results provide basis for creating a good environment for office work. The concept and techniques developed here also will enhance the process of achieving reliable CFD models for simulating indoor environments.

Figures 2 to 4 are temperature fields at 0.5 m away from the south wall, 0.5 m away from the west wall and 0.5 m away from the north wall respectively. Figure 2 is a selected plane section and the positions where the shelves are located, i.e., close to the window facing the south. This area is influenced directly by the warm air through the door from outside. The wall of this position is also direction of the outside sun. As a result of this external heat transfer, this position is a high temperature of about 300 K (27 °C) and at the extreme end is about 292 K (19 °C) as can be seen in the Figure.
Figure 2: Temperature fields at 0.5 m away from the South inner wall

Figure 3 is a selected plane section and the position that is close to the occupant sit. This plane section (area) is been influenced by the warm air passing through the window to the office and the adjacent wall face the sun. Consequently, this position is a high temperature of about 309 K (36 °C) due to this external heat transfer but lower at the extreme end of the cross sectional area.

Figure 3: Temperature fields at 0.5 m away from the West inner wall

Figure 4 is a selected plane section and the position that is also close to the occupant sitting area. This plane sectional area is also been influenced by the warm air passing through the window to the office but the adjacent wall is not exposed to the sun. As a result, these temperatures are moderately high of about 307 K (34 °C) and lower at the extreme end of about 303 K (30 °C).
Figure 4: Temperature fields at 0.5 m away from the North inner wall

Figures 5 to 7 are air velocity at 0.5 m away from the south wall, 0.5 m away from the West wall and 0.5 m away from the North wall respectively. Figure 5 is a selected plane section and the position where the shelves are located. This area is influenced directly by the warm air and wind speed through the door from outside. In the Figure, due to the office door’s ventilation, there is high air velocity of about 1.6 m/s in front of the door. However, low air velocity of about 0.2 m/s is experienced at the back.

Figure 5: Air velocity at 0.5 m away from the South inner wall

Figure 6 is also a selected plane sectional area and at the extreme end of the section is closer to the occupant sitting area. The part that is very close to the window has the air velocity of about 1.0 m/s and the part that is far away from the window has the air velocity of about 0.184 m/s. Also, due to the wall block, when incidence airflow is supplied along the X axis, the air flow formed vortex; as a result, the lower and upper part of the window has a high air velocity of about 1.2 m/s as can be seen in the Figure.
Figure 6: Air velocity at 0.5 m away from the West inner wall

Figure 7 is a selected plane sectional area and the extreme end of the window close to the occupant sitting area. This Figure is similar to Figure 6.0 but the air velocity at the extreme end of the cross sectional area is much higher, about 0.4 m/s.

Figure 7: Air velocity at 0.5 m away from the North inner wall

4. Conclusion

This paper developed a CFD numerical model and the model was used to evaluate the air temperature distribution and air velocity of an office room which is naturally ventilated in summer. The result showed that the natural ventilation could not undertake the heat load to provide good indoor thermal comfort. In the temperature distribution, temperature around the room is high as such not good for human comfort. Though, the air velocity is moderate around the office room, however, the room will not be comfortable for human according to ASHRAE standard. It is recommended that a window should be installed at the back, opposite the door, West direction, or the two existing window dimensions increased.
References