Analysis Of Ventilation Rate In Cross Ventilated Rooms By Varying Aperture Shape Of Windows Using Cfd

V. Sivakumar, Dr. K. Visagavel, A. Selvakumar

1Assistant Professor, Department of Mechanical Engineering, M.A.M College of Engineering, Trichy.
2Professor, Department of Mechanical Engineering, Knowledge institute of Technology, Salem.
3Assistant Professor, Department of Mechanical Engineering, Knowledge institute of Technology, Salem.

Accepted 22 April 2017; Available online 12 May 2017

ABSTRACT
On this era of increasing energy demand, natural ventilation appears to be the proper solution as it provides the necessary ventilation required for the room by means of the natural forces, such as wind. It ensures healthy and comfortable indoor climate through minimal energy consumption at minimal cost. Our project aims to identify the air velocities and air change per hour in a cross ventilated room through numerical predictions. The standards for natural ventilation are taken into account along with the heat transfer concepts and the analysis is carried out on different shapes of inlets and outlets at different heights. The models were analyzed using CFD software by using SIMPLE and SIMPLEC algorithm to solve the equations and the results are compared.

KEYWORDS:

INTRODUCTION

Natural ventilation is defined as using passive strategies to supply outdoor air to a building’s interior for ventilation and cooling. With proper design, appropriate to the building location and use, natural ventilation can replace all or part of a mechanical system. The need to reduce our consumption of energy and to give users more control over their immediate environments, are good reasons for people now to understand the role of natural ventilation in buildings and to become familiar with the basic principles involved. It serves the purpose of reducing not only the strain from your wallet but also on the environment.

On this era of increasing energy demand, natural ventilation appears to be the proper solution as it provides the necessary ventilation required for the room by means of the natural forces, such as wind without the use of any depleting resources. It also ensures healthy and comfortable indoor climate through minimal energy consumption at minimal cost. Buildings can be ventilated mainly by two methods: Single-sided ventilation and Cross-sided ventilation. Both these involves the process of air flowing into the room, circulating and then exiting through the apertures which happen due to pressure differences through buoyancy and wind forces. The physical processes taking place due to this type of ventilation can be computed using analytical and numerical methods.

Camille Allocca et al [3] have studied the effects of ventilation in a single-sided room due to buoyancy, wind and their combination using analytical and empirical methods and found. From this study, it has been found that the stack and wind effects can either reinforce or may even oppose each other. There is no guiding rule for predicting the counteraction of wind and stack effect. In the investigation of Chrysanthi Karagkouni et al [4], two facades consisting of elastically deformable vertical louvers, were implemented to enable experimentation and a
structure to improve the ventilation performance in a building was identified. Sinha et al. [5] have analyzed the distribution of air in a room with and without buoyancy effects by numerical means. A few works of K. Visagavel and P. S. Srinivasan [6-8] have been studied. A study in a single-sided ventilated room applied CFD analysis for both wind-driven and buoyancy cases by using rectangular and convergent-divergent nozzle type openings. It showed that the nozzle type opening increases effective depth of ventilation in terms of airflow patterns and had moderate effect on ACH. In a specific case, the atmospheric domain around the room has been taken into consideration and the analysis was carried. It was found that the increase in atmospheric velocity created recirculation zones in the room and the velocity in the room increased at different locations and it was suggested that the inlet and outlet locations has to be carefully placed inorder to get better comfort. Also, a comparison between cross and single ventilated rooms by changing the size of the opening brought the result that single-sided leeward ventilated room has the least effect and cross-ventilated rooms are most effective.

As cross-ventilation was found effective from the literature review and focusing in detail about cross ventilation would be more beneficial. Hence, this study uses computational fluid dynamics to identify the physical processes involved in a cross-sided ventilation room. This was done by taking three different openings—square shaped, convergent-divergent nozzle shaped and circular shaped opening. CFD analysis was used to solve the problem as the parameters can be varied effortlessly and the results can be obtained rapidly.

**Problem Investigated:**

The ventilation aspects in a cross ventilation room are investigated. Fig. 1 shows the computational domains (two-dimensional – 2D). A 10 x 10 x 10 ft room of height H= 10 ft and width L=10 ft is considered for investigation. The room has an opening at height (h1) for inlet and another opening at height (h2) for outlet from the floor and is located on the vertical (building) walls. As it is cross-sided naturally ventilated room, the air flow is through the inlet opening on the left side, passes through the room and flows out through the right opening using buoyancy effect. For this room, the opening shape is varied and the analysis is done. Also, analysis is made to compare the air change rate obtained from the analytical methods. The problem is investigated numerically using heat transfer concepts and analyzed by CFD softwares.

![Fig. 1: Computational domains](image)

**Solution methodology:**

**A. Analytical methods:**

**Load Calculations:**

We are taking a 10 x 10 x 10 ft room as a sample room and the results are investigated.

The area of the ventilation opening used here is 2 x 2 ft opening.

A building or room gains heat from many sources. Inside occupants, computers, copiers, machinery, and lighting all produce heat. Warm air from outside enters through open doors and windows, or as ‘leakage’ though the structure. However the biggest source of heat is solar radiation from the sun, beating down on the roof and walls, and pouring through the windows, heating internal surfaces. So the major heating loads for a room are the envelope load, people load and fresh air load.

**Envelope load:**

We know that, heat transfer \( Q = UA\Delta T \)

The value of overall heat transfer coefficient \( U \) can be obtained from ASHRAE 62.1 standards [1].

Considering the outdoor temperature of the room to be 30°C and the desired indoor temperature to be 25°C, the change in temperature appears to be 5°C.

Total envelope load, \( Q = (Q_1 + Q_2 + Q_3 + Q_4 + Q_5 + Q_6) / 6 \)

Where \( Q_1 = \) Heat generated on wall with inlet
\( Q_2 = \) Heat generated on wall with outlet
\( Q_3 = \) Heat generated on ceiling
\( Q_4 = \) Heat generated on the floor
Q5 = Heat generated on other wall
Q6 = Heat generated on other wall
Total envelope load, Q = 1.146 kW

People load:
The internal cooling load due to occupants consists of both sensible and latent heat components. The rate at which the sensible and latent heat transfer take place depends mainly on the population and activity level of the occupants. Hence a portion of the heat transferred by the occupants in the form of radiation.

As per the standards, 20-50 ft / person floor area is required for comfortable occupation and hence the no. of occupants for a 10 x 10 ft room is to be 5.
The sensible heat and latent heat of the occupants can be obtained from ASHRAE 62.1 standards [1].
People load = Total sensible load + Total latent load
= 0.55 kW

Fresh air rate:
As per ASHRAE standard 62.1:6.2.2.1 [2], the design outdoor airflow (or) ventilation rate required in the occupiable space is given by
Ventilation rate, \( V = R_p P_z + R_a A_z \)
\( V = 31 \) cfm

Fresh air load:
The fresh outdoor air enters the room through the openings. A load is added by the incoming fresh air which can be calculated using the sensible and latent heat of the air.
Fresh air load = 0.18 kW
Total load in the room = Envelope load + People load
+ Fresh air load
= 1.146 + 0.55 + 0.18
= 1.876 kW

Amount of heat required to raise the temperature of the room:
We know that, heat transfer can also be expressed as \( Q = C_p m \Delta T \)
For a room of volume 1000 ft3,
\( \Delta T = 40.6 \) K

Required air flow rate:
In order to find the air change per hour it is essential to find the fresh air rate necessary for comfortable occupancy of the occupants.
So, the required air flow necessary for the desired ventilation level can be found using the equation,
Required air flow rate, \( L = \frac{Q}{C_p \rho (\text{th}-\text{tr})} \)
Fresh air rate, \( L = 79.45 \) ft3/min

Ventilation Rate (m³/h):
As per CIBSE Guide B2, The rate at which air is exchanged is an important property for the purpose of ventilation design and heat loss calculations. This property is expressed in ventilation rate (m³/s).
Ventilation rate (m³/h) = Air change rate (/h) x Room volume (m³)
\( \text{Ach} = \frac{\text{fresh air rate x 100}}{1000} \times \text{volume of room} \)
\( \text{Ach} = 4.76 \) h⁻¹
Ventilation rate (m³/s) = 4.76 x 1000/3600 = 1.32
Therefore the air change rate for the required room has been calculated using the standards for ventilation and heat transfer concepts.

B. Numerical methods:
The given analysis involves the following assumptions:
a) The analysis is in a two-dimensional Cartesian coordinate system and steady state condition.
b) The fluid properties are constant and flow is isothermal.
c) The flow is turbulent.
d) Flow is incompressible and Newtonian.
Under these assumptions, the governing equations to be solved are as follows:
a) Continuity equation
\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 1
\]

b) Momentum equation in x and y direction

\[
u \frac{\partial v}{\partial x} + v \frac{\partial u}{\partial y} = \frac{1}{\rho} \frac{\partial p}{\partial x} + \gamma \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
\]

\[
u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \frac{1}{\rho} \frac{\partial p}{\partial y} + \gamma \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)
\]

where \( \rho \) is the density, \( u \) the velocity in x-direction, \( v \) the velocity in y-direction, and \( p \) is the pressure. For modeling the turbulent quantity, standard two-equation k-\( \varepsilon \) model is used. The above equations are solved numerically using the FLUENT 6.0 CFD software.

The boundary conditions to solve the problem are:

a) The air flow is given in the inlet,

b) No slip boundary condition is applied on the floor

c) The building surface and the outlet has zero pressure boundary condition (p = 0).

The model is created using the SOLIDWORKS software. Boundary layer type mesh is applied on the floor and building surfaces. The uniform or expanding grid spacing is applied over the remains regions. The entire region is treated as fluid (air) continuum. The basic conservation equations are solved numerically using the FLUENT 6.0 CFD software. Steady state solver is activated.

Air as fluid is defined and the Boussinesq’s model is selected. Gravity is activated. In boundary condition, value for the inlet-velocity magnitude has given. Iterated still the solution is calculated. The grid display, contour of properties, velocity vector and stream function are viewed from the ‘Display’ option.

**Case 1:**

First, the room is set with a square shaped opening for both the inlet and outlet passages. The opening area is assumed to have an area of 2 x 2 ft. The openings are kept at heights \( h_1 = 4 \) ft and \( h_2 = 6 \) ft from the floor and the results are taken.

**Case 2:**

Secondly, a convergent-divergent type of apertures is used. The inlet has a convergent shaped aperture and the outlet has a divergent shaped aperture as these structures could increase the velocity of the air passing through the room. Here in the inlet, the opening has an area of 2 x 2 ft and drafts about 25° and then converges. The outlet has the exact opposite dimensions on its both sides. The apertures are kept at heights \( h_1 = 4 \) ft and \( h_2 = 6 \) ft from the floor and the results are analyzed.

**Case 3:**

Finally, a third cross-section was selected to have a circular shape. Here, the openings have a radius of 2 ft and were placed at different heights on the opposite walls. The openings are assumed at heights \( h_1 = 4 \) ft and \( h_2 = 6 \) ft from the floor and the results are observed.

**RESULTS AND DISCUSSION**

From the theoretical calculations, we can find that to have an air change rate of 4.76h\(^{-1}\) we must have an inlet velocity of 0.11 m/s. An analysis was made with this inlet velocity and the results were taken. The results obtained are shown below:
Fig. 2: (a) pressure (b) velocity (c) vector pressure (d) vector velocity

Case 1:
Now, the results taken for the analysis of the square–shaped aperture using buoyancy effect are shown and are then compared with the other cases. The inlet velocity is given as 1.66 m/s which was taken from weather data.
Fig. 3: (a)

Fig. 3: (b)

Fig. 3: (c)

Fig. 3: (d)

Fig. 3: CFD results for room with square – shaped aperture (a) pressure (b) velocity (c) vector-velocity (d) density
Case 2:

The analysis made with the convergent inlet and divergent outlet is shown with their results. The analysis starts with an inlet velocity of 1.66 m/s and the flows emerge out of the room through the outlet. The flow process can be found below:

Fig. 4: (a)

Fig. 4: (b)

Fig. 4: (c)

Fig. 4: (d)

Fig. 4: CFD results for room with convergent - divergent aperture (a) pressure (b) velocity (c) vector-velocity (d) density
Case 3:
The analysis with inlet velocity 1.66 m/s given through the circular opening is carried out and the final results are shown below. The results of all three cases are compared and the optimum design is found.

Fig. 5: (a)

Fig. 5: (b)

Fig. 5: (c)
Fig. 5: CFD results for room with circular aperture (a) pressure (b) velocity (c) density.

Conclusion:
This study draws attention to the physical procedures governing air movement during the natural ventilation through the description of both analytical and computational methods. It can be observed that the solutions to the ventilation problems can be solved easily and effectively using numerical solutions. The pressure and velocity changes in the room for three different aperture shapes are noted. From the results, it can be found that the square–shaped apertures take in the maximum amount of air intake and the air flows along the entire room. Also, this type has the good balance between high velocity and the low pressure. Hence, among these three cases, square–shaped apertures would give better ventilation rate to the room. For future research, the size of the window of cross–sided ventilated room may be changed and analyzed for better air flow.
REFERENCES

4. Chrysanthi (Sandy) Karagkouni, Ava Fatah gen Schieck, Martha Tsigkari, and Angelos Chronis, “Façade apertures optimization: Integrating cross-ventilation performance analysis in fluid dynamics simulation”.