NATURAL VENTILATION OF BUILDING USING CFD ANALYSIS

VENKATESH P M¹, KANNAN M², BHARATH KUMAR N³, RAO D S N M⁴

¹,³,⁴ Associate Professor, Department of Electrical and Electronics Engineering, VFSTR, Guntur
²ME Energy Engineering, Department of Mechanical Engineering, PSG College of Technology, Coimbatore

https://doi.org/10.26782/jmcms.2019.12.00047

Abstract

The natural ventilation is flow through a closed surfaces. The internal air quality is good when the process of flow is continuous state of inlet and exhaust. By these way this paper described the room with close surfaces and flow direction comparison. Two open window with similar dimension are taken with top and bottom. The velocity of the flow inside are measured and determined a per measurements. By estimating the natural flow inside building the computational fluid dynamics process taken placed by using both concepts such as RANS method and LES method such as capturing the large eddy and by following governing Navier stokes equations

Keywords : natural ventilation, fluid dynamics process, RANS method, Navier stokes equations

I. Introduction

Natural ventilation is the process of entering and leaving the air through an indoor space without using forced system. In most of countries, the buildings are responsible for one third of all energy consumption e.g. in US, about thirty of total energy consumption is used in non-domestic buildings, and of that fraction about thirty percentage is used in heating and cooling [I], [VII], [III].

One-sided ventilation has best adaptive comfort hours than two-sided ventilation and produces much low ventilation volume [XII] is used when there is non-availability of other options.

The most reliable methods of getting information about the air flow and pressure distribution around and inside the building are through full scale measurements [VI], scale model testing using wind tunnel [X] and computational fluid dynamics (CFD).

The common CFD techniques are direct numerical simulation (DNS), large-eddy simulation (LES) and Reynolds averaged Navier-Stokes (RANS) equation with turbulence models. Each technique handles turbulence in different ways [IX, VIII].
Among those techniques, RANS is widely used by most CFD software [XIII], however, LES should be more suitable than the RANS approach to study highly three-dimensional or separated flows, especially those in which the gradient transport hypothesis, and consequently one and two-equation models of turbulence, fails [XI]. It was introduced by Dearsdoff in the early 1970s for meteorological applications.

The present investigation is focused on the application of three dimensional RANS and LES modelling on wind driven natural ventilation of Two openings at One-sided buildings.

II. Methodology

This study involves 4 configurations models of leeward double opening (Figure 1). The dimensions of the building-like models are 6m length and 6m height which is closed box room. The computational domain constructed at a definite spaces of height and length sufficiently large to avoid disturbance of air flow around the building. A logarithmic wind profile upwind of the building is employed. The commercial software package CFD-ACE from the ESI group is used for the computation.

The parameters are airflow velocity, air temperature, mean radiant temperature and air relative humidity. They are also used as the boundary conditions for the CFD flow analyses and for validating the CFD model. At each location, the probe of the measuring instrument was placed at the height from the floor, as shown. This height represents an average breathing level of the occupants when they are in a standing position.

III. Computational Fluid Dynamics (CFD) software

Computational fluid dynamic (CFD) method was employed in this study to predict the distribution of airflow velocity, air temperature and humidity at the data collection locations inside the hall, under the present ventilation condition and when exhaust fans were virtually installed at the hall envelope. The results were then used to calculate the thermal comfort indices at these locations. The indices were then compared with the corresponding values determined based on data obtained from the field measurement.

ANSYS FLUENT CFD software was used to develop the computational domain based on the geometry of the mosque’s hall and to perform all flow analyses. A parametric study was carried out to examine the effects of installing exhaust fans at different locations of the hall envelope, on the thermal comfort indices.

The goal was to find out the suitable number of fans to use and their location that would give the best improvement in the thermal comfort indices inside the hall.

The CFD flow analysis involves the following steps: construction of a simplified model of the hall (the computational domain), meshing the model, prescribing the boundary conditions, choosing a suitable air flow model based on the Reynolds Average Navier-Stoke (RANS) family models, setting the solution methods, and finally specifying the convergence criteria of the solution. The analyses were carried
out in a transient mode until a satisfactory convergence of all the residuals was attained

IV. Result & Discussion

The result is based on the flow at interior side of building by comparison the best model of LES and RANS. The velocity components have been determined, along stream wise and vertical direction, respectively. LES is used to compare with RANS as LES is produced more precise results but more time-consuming method.

Figure 2 shows the air flow pattern inside the building. Based on the figure the configuration is taken into account and calculated for both RANS and LES models. The opening is marked as A and B, where the air enters the building through the lower opening A and comes out through the upper opening B. If the level of the opening is at the same height, the air will enter through the lower area and come out through the upper area of both openings.

Table 2 shows numerical values of air flow rate of simulation that being performed for the given building. Each configuration gives different flow rate and it is found that the highest flow rate occurs at configuration a(iv) for RANS and a(ii) for LES, whereas the lowest flow rate occurred at configuration a(i) for both RANS and LES.

Figure 1  Mesh generation using CFD software
V. Conclusion

In this recent study, RANS and LES approaches have been applied to wind driven natural ventilation in a cubic building. Both RANS & LES scheme gives almost the same air flow rate with the difference between four to eight percent which are discussed. The location & arrangement of the opening influences, the air flow rate and air flow pattern.

Table 1: Numerical values of airflow rate (m$^3$/s)

| CONFIGURATIONS | RANS       | LES       |
|----------------|------------|-----------|
| a(i)           | $6.67 \times 10^{-1}$ | $6.08 \times 10^{-1}$ |
| a(ii)          | $6.91 \times 10^{-3}$ | $7.56 \times 10^{-3}$ |
| a(iii)         | $6.42 \times 10^{-1}$ | $7.20 \times 10^{-1}$ |
| a(iv)          | $7.15 \times 10^{-1}$ | $7.13 \times 10^{-1}$ |
| System | Configuration | 2D-Airflow Pattern |
|--------|---------------|--------------------|
| a(i)   | ![Diagram](image) | ![Diagram](image) |
| a(ii)  | ![Diagram](image) | ![Diagram](image) |
| a(iii) | ![Diagram](image) | ![Diagram](image) |
| a(iv)  | ![Diagram](image) | ![Diagram](image) |

Table 2: Double Opening on Leeward Wall
References

I. A.R.VijayBabu, Ch. Umamaheswara Rao, L. Tirupathaiah, Energy Conservation, Green House Gas Emission Reduction and Management Strategies of VFSTR University: A Case Study, Journal of Applied Research in Dynamical & Control Systems, Volume 9, Issue 4, PP. 21-27, 2017.

II. G. Gan, "Effective depth of fresh air distribution in rooms with single-sided natural ventilation," Energy and Buildings, vol. 31, pp. 65-73, 1// 2000.

III. J. Morrissey, T. Moore, and R. E. Horne, "Affordable passive solar design in a temperate climate: An experiment in residential building orientation," Renewable Energy, vol. 36, pp. 568-577, 2// 2011.

IV. Kavitha, M., et al. "Evaluation of Antimitotic Activity of Mukiamaderaspatana L. Leaf Extract in Allium cepa Root Model." International Journal 4.1 (2014): 65-68.

V. K. Visagavel and P. S. S. Srinivasan, "Analysis of single side ventilated and cross ventilated rooms by varying the width of the window opening using CFD," Solar Energy, vol. 83, pp. 2-5, 1// 2009.

VI. M. M. Eftekhari, L. D. Marjanovic, and D. J. Pinnock, "Air flow distribution in and around a single-sided naturally ventilated room," Building and Environment, vol. 38, pp. 389397, 3// 2003.

VII. P. F. Linden, "The fluid mechanics of natural ventilation," Annual Review of Fluid Mechanics, vol. 31, pp. 201-238, 1999.

VIII. Q. Chen and L. Glicksman. Application of computational fluid dynamics for indoor air quality studies. Available: http://www.accessengineeringlibrary.com/mgh_pdf/0071450076_ar059.pdf.

IX. Q. Chen, "Ventilation performance prediction for buildings: A method overview and recent applications," Building and Environment, vol. 44, pp. 848-858, 4// 2009.

X. T. S. Larsen and P. Heiselberg, "Single-sided natural ventilation driven by wind pressure and temperature difference," Energy and Buildings, vol. 40, pp. 1031-1040, 6// 2008.

XI. U. Piomelli, "Large-eddy simulation: achievements and challenges," Progress in Aerospace Sciences, vol. 35, pp. 335-362, 5// 1999.

XII. Y. Wei, Z. Guo-qiang, W. Xiao, L. Jing, and X. San-xian, "Potential model for single-sided naturally ventilated buildings in China," Solar Energy, vol. 84, pp. 1595-1600, 9// 2010.

XIII. Yakhot, S. A. Orszag, S. Thangam, T. B. Gatski, and C. G. Speziale, "Development of turbulence models for shear flows by a double expansion technique," Physics of Fluids A: Fluid Dynamics (1989-1993), vol. 4, pp. 15101520, 1992.