Indian Journal of Science and Technology, *Vol 8(17), DOI: 10.17485/ijst/2015/v8i17/68171, August 2015* *ISSN (Print) : 0974-6846*
ISSN (Online) : 0974-5645

An Analysis of Effect of Wing Walls on Natural Ventilation

T. V. K. Sushil Kumar[1*], J. Chandrasekhar[1], Shriram K. Vasudevan[2] and A. Balachandran[3]

[1]Department of Aerospace Engineering, Amrita School of Engineering, Amrita Vishwa Vidyapeetham (University), Coimbatore - 641112, Tamil Nadu, India; tvk_sushilkumar@cb.amrita.edu, jc@amrita.edu
[2]Department of Computer Science and Engineering, Amrita School of Engineering, Amrita Vishwa Vidyapeetham (University), Coimbatore - 641112, Tamil Nadu, India; kv_shriram@cb.amrita.edu
[3]Corporate and Industry Relations, Amrita Vishwa Vidyapeetham (University), Coimbatore - 641112, Tamil Nadu, India; a_balachandran@cb.amrita.edu

Abstract

Background/Objectives: The air flow inside a building due to natural ventilation consists of recirculation regions and a main flow between the inlet and outlet. Recirculation regions are characterized by low velocities resulting in thermal discomfort for the occupants. Winged walls are basically protrusions from external walls to improve the process of natural ventilation. Existing literature on wing walls is on main air flow pattern only. **Methods/Statistical Analysis**: In this paper, CFD was used to compare effectiveness of configurations suggested in the literature. Information was also obtained about the recirculation regions. Further conventional wing walls were modified and its effectiveness presented. **Findings**: Valuable information on recirculation regions can be accomplished through this innovation. Optimum dimension of wing walls to improve natural ventilation has also been achieved through the research. More than all these, the information obtained from this research can be used for designing energy efficient buildings.

Keywords: Air-Conditioning, CFD, Energy Efficiency Green Buildings, Natural Ventilation, Wing Walls

1. Introduction

The importance of going green and saving energy has been mantra for many upcoming countries and India is no different[1]. Natural ventilation can be used as an alternative to conventional air conditioners to reduce energy consumption. However natural ventilation depends on wind velocity outside the building. Wind velocity at a site shows both seasonal and diurnal variations and is usually less than 10 m/s. Since the wind velocities are low and seasonal, the building plan has to be optimized for effectively utilizing natural ventilation. Such optimization requires study of both internal and external air flow patterns. Wind tunnel experiments to visualize the flow field for both inside and outside a building was carried out in 60's. Guidelines for effective use of natural ventilation in tropical regions, empirical formulae to measure the pressure distribution on external building surfaces and information on external flow and direct flow between the inlet and outlet was also formulated during this timeframe. G.C. Da Graca[2] on the other hand focused on internal flow in terms of both the main jet and recirculation regions. His simulations were applicable for rooms with geometrically opposite inlet and outlet windows.

Wing walls are external projections to effectively improve the natural ventilation. Mark Dekay, 2014[3] gave broad guidelines on the direct flow pattern between the inlet and outlet windows. Cheuk Ming Mak, 2007[4] carried out a study on wing walls for two windows placed on the same side. Windows should be placed on adjacent walls for households having multiple rooms. Hence this study focused on a house with wing walls and windows placed on adjacent sides. Further information on recirculation regions inside the room is also required to study thermal comfort levels.

This study was carried out using Autodesk simulation CFD software, CFD 2013 and Revit. The CFD model was

**Author for correspondence*

validated with results obtained by G.C. Da Graca[2]. Inlet velocities for buildings with and without wing walls were studied. Guidelines given by Mark Dekay, 2014[3] were simulated and additional information on recirculation regions was obtained. Parametric study on wing wall lengths was also carried out. Wing walls might not be economically feasible for high rise buildings. Hence a study on shorter walls and an inverted 'L' configuration were carried out. Results of this study on natural ventilation will be useful for designing energy efficient buildings.

2. Validation

Results given in G.C. Da Graca, 2003[2] dissertation were used to validate our CFD model proposed in this paper. His dissertation has given formulae for air velocities in both the main jet and recirculation regions, for a rectangular room plan form. A room with a square plan form of 4.5 m and a height of 3.5 m used by G.C. Da Graca, was used for our simulations. Both the inlet and outlet windows are placed at the center of geometrically opposite walls and are of 1m^2 area each. G.C. Da Graca[2] has not given details about the external volume used in his simulations. It was felt that suitable external volume should be defined, so that real world applications can be studied. Hence the room was enclosed by an external volume. Best practices suggested in Autodesk Simulation CFD 2013, for architectural applications are used for deciding the external volume. CAD model used for our study is as shown in Figure 1. All dimensions used in this paper follow SI units.

Wind velocity and zero gauge pressure were used as Inlet and outlet boundary conditions for the external volume. To simulate a free space environment, slip condition was assigned to the top and sides of the external volume. Adaptive meshing was used to ensure finer mesh in high gradient regions and coarser meshes in rest of the regions. Comparison between velocities obtained using G.C. Da Graca[2] formulae and our simulations are listed as Table 1.

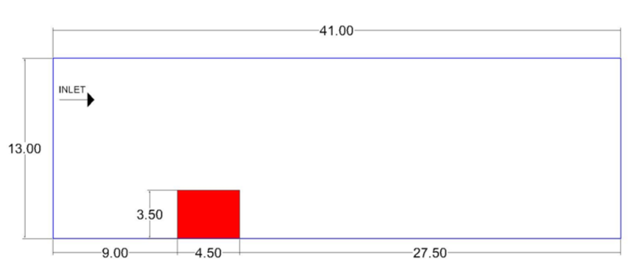

Figure 1. CAD model used for simulation.

Table 1. Comparison of velocities between G.C. Da Graca[2] and our simulations

Inlet velocity (m/s)	Main jet velocity (m/s)		Recirculation velocity (m/s)	
	G.C. Da Graca	Simulation	G.C. Da Graca	Simulation
1.0	0.78	0.80	0.17	0.17
1.5	1.17	1.10	0.26	0.26
2.0	1.56	1.50	0.34	0.34
2.5	1.96	1.90	0.43	0.40

It is evident from the Table 1 that there is a close match between the values predicted by G.C. Da Graca and our CFD simulations. Hence our CFD model was validated and used in our studies explained in subsequent paragraphs.

3. Effect of Wing Walls on Natural Ventilation

Wing walls if positioned properly, produce high and low pressure regions on the inlet and outlet windows respectively. Hence they are capable of producing cross ventilation even during low external wind velocity conditions. In this study, a room of length 4.87m and width of 3.33 m (16 ft X 11ft) was used. These dimensions are close to the golden ratio, which is the ratio of length by width used by architects, Francis D.K. Ching, 2007[5]. Square windows of side 1.22 m (4 ft) were placed on adjacent walls, because windows on geometrically opposite walls are difficult to construct inside a house. The winged walls were of 1.22 m in length and 4.27 m in height. Four different configurations were studied. The results are as shown in Figure 2. A free air stream velocity of 5 m/s at the inlet of the external volume was used throughout the study subsequently. Dark shades in Figure 2 indicate low velocity regions.

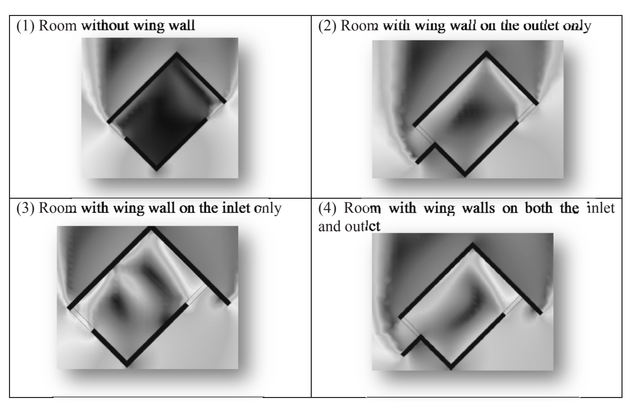

Figure 2. Effect of Wing walls on natural ventilation.

Table 2. Effect of Wing walls on cross ventilation

Configuration	Average inlet velocity (m/s)
1	2.64
2	4.1
3	4.7
4	5.12

Average inlet velocities for different configurations are given in Table 2 below. It was observed that a single wing wall on the inlet side (third configuration) can result in 92% of the velocity achievable with two wing walls.

4. Study of Air Flow Pattern for Different Wing Wall Positions

Rooms with an inlet and outlet on adjacent walls experience two regions of flow namely a wall jet and regions of recirculation. Velocity in the wall jet region is much higher compared to the recirculation region, where the flow is almost stagnant. Such recirculation areas result in reduced thermal comfort. Mark Dekay et al, 2014[3] has given guidelines for streamlines between the inlet and outlet only, for different wing wall configurations. In addition to obtaining information on the flow between the inlet and outlet, this study identified recirculation regions. These regions are characterized by very low velocities and are shaded black in Figure 3 below.

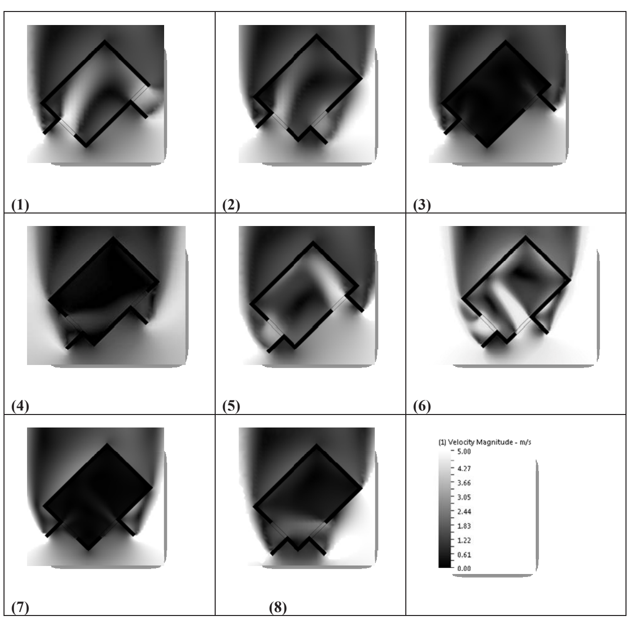

Figure 3. Flow visualization for different Wing wall positions.

Table 3. Comparison between Mark Dekay[3] rating and our CFD results

Configuration	Mark Dekay rating	CFD results (inlet velocities) m/s
1	Best	3.96
2	Good	3.0
3	Poor	0
4	Poor	1.0
5	Best	3.78
6	Good	3.3
7	Poor	0
8	Poor	1.9

Average inlet velocities obtained for eight configurations shown in Figure 3 above is tabulated as Table 3.

Figure 3 and Table 3 indicates that the results obtained through CFD simulations are in agreement with the guidelines given by Mark Dekay et al[3].

5. Effect of Wing Length on Average Inlet Velocities

A study on effect of wing wall length 'L' on inlet velocity was carried out, for a given position of inlet and outlet as shown in Figure 4.

Inlet velocities with increasing wing wall lengths and its construction cost is as shown in Figure 5. Construction cost is taken as Rs. 3225-/- for every square meter.

It was observed that, a 0.45 m wing wall length, was sufficient to achieve 90 % of the velocity obtained using a wing wall length of 1.8 m, at 25 % cost.

6. Wing Wall Height

Above mentioned simulations were carried out using a wing wall of height 4.27 m, which is the height from the ground level to the top of the parapet wall. Construction

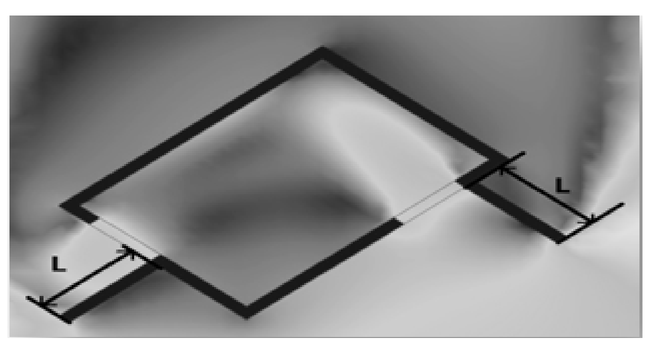

Figure 4. Effect of wing wall length on flow pattern.

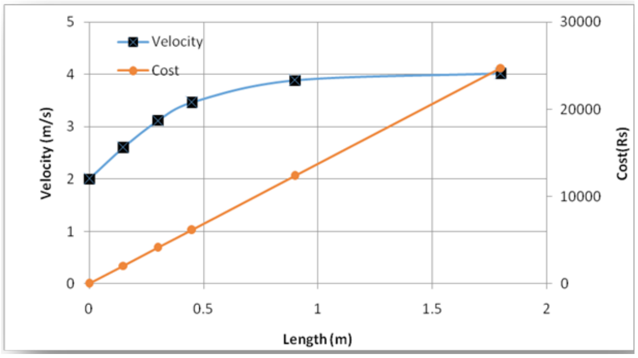

Figure 5. Effect of wing wall length on inlet velocity and construction cost.

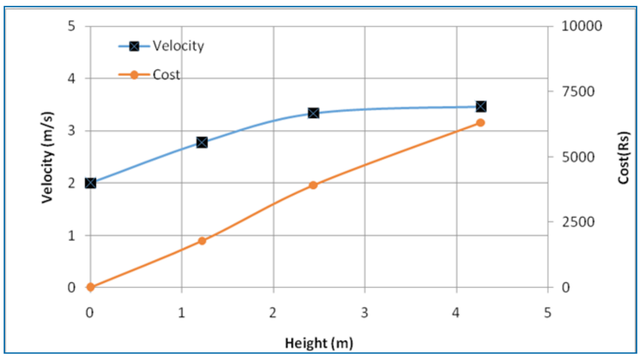

Figure 6. Effect wing wall height on inlet velocity and construction cost.

of such high walls may not be economical. Hence a study was carried out on the effect of wing wall height on inlet velocity. In this study the wing wall length was maintained constant at 0.45 m. Results are as shown in Figure 6 above. It is evident that a wing wall of same height as that of the window (1.22 m) results in 80 % of the inlet velocity obtained using a full height wing wall (4.27 m), at 28% cost.

7. Inverted 'L' Type Wing Wall

An inverted 'L' type wing wall not only improves the aesthetics but also provides shading. Hence a study was carried out on an inverted "L" wing wall of dimensions shown in Figure 7. An average inlet velocity of 3.17 m/s was observed, which is 90% of the velocity of a full height wing wall. It was felt that an inverted 'L' type wing wall is a good option compared to a full height wing wall.

8. Conclusion

Effect of wing walls on natural ventilation was carried out in this study. Existing literature focuses on flow pattern

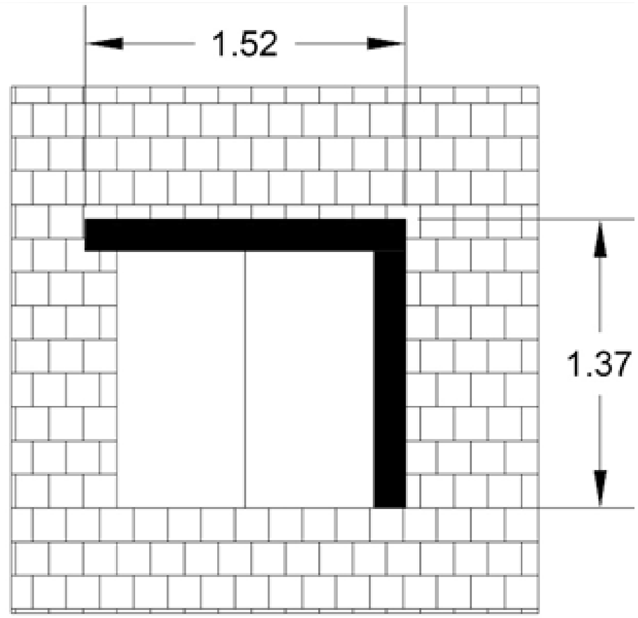

Figure 7. Front view of Inverted 'L' type wall.

between the inlet and outlet windows with wing walls. Guidelines given in the literature are in agreement with the results of this study. In addition recirculation regions were identified. It was also found out that a wing wall length above 0.5 m does not result in appreciable increase in inlet velocities. Further construction of long and full height wing walls is expensive. This study indicates that a wing wall of window height, is sufficient for achieving 80 % of the inlet velocity achievable with a full height wing wall, at 28% cost. To improve the aesthetics of such short walls and to provide shading, an inverted 'L' configuration was studied. This configuration resulted in another 9% increase in average inlet velocity.

9. References

1. Dhanda U, Gupta S. Exploring the Go Green Mantra in India. Indian Journal of Science and Technology. 2015 Feb; 8(S4):247–58.
2. Graca DGC. Simplified models for heat transfer in rooms [PhD dissertation]. San Diego: University of California;
3. Dekay M, Brown GZ. Sun, wind and light-architectural design strategies. 3rd ed. New Jersey: Wiely; 2014. p. E122.
4. Mak CM, Niu JL, Lee CT, Chan KF. A numerical simulation of wing walls using computational fluid dynamics. Energy and Buildings. 2007; 39(9):995–1002.
5. Ching FDK Architecture-Form, space and order. 3rd ed. New Jersey: John Wiley; 2007. p. 302.