International Journal of Fluids Engineering.
ISSN 0974-3138 Volume 11, Number 1 (2019), pp. 1-23

http://www.irphouse.com

Numerical Investigation of Wind Driven Natural Ventilation in a Mega Warehouse Building

Shrinivas Hulsure[1] R.S.Maurya[2]

Sardar Patel College of Engineering, Mumbai, India.

Abstract

This study investigated present condition of wind driven natural ventilation rate and air flow distribution in a low rise gable roof mega warehouse building, located at Embassy Industrial Park Bilaspur, Gurugram, India, using Coupled CFD Approach.

A suitable turbulence model was selected by comparing CFD results for external surface distribution of mean pressure coefficient on a gable roof building with full scale experimental data from literature. The wind driven cross ventilation rate predicted by Coupled CFD approach is also compared with Network Method. The effect of ridge ventilator on wind driven natural ventilation is investigated and new ridge design is suggested for better airflow distribution. In order to improve the indoor ventilation in the warehouse an alternative configuration of louver windows and ridge ventilator is suggested.

Key words: Wind driven natural ventilation, Mega warehouse gable roof building, Ridge Ventilator, Coupled CFD Approach, Network Method

1. INTRODUCTION

E-commerce sites such as Amazon, Flipkart, Myntra are booming in India. The online consumers increasingly expect not just endless choices, but also instant gratification. Hence it is not enough to carry an item, a retailer has to be able to deliver it quickly, too. In order to meet these demands retailers looks towards extra-large storage exceeding 113,600 cubic meters known as 'Mega Warehouses'. Mega Warehouses are distributive warehouses which are different from traditional storage warehouses. Mega warehouses serve as all in one fulfilment centres for **storage, packaging and shipping** with a direct connection to the consumers.

Ventilation refers to the changing of air in an enclosed space. The objective of ventilation is to provide fresh air. Ventilation involves two processes. One is '**air change rate'** by which air is replaced with fresh air and second is '**air distribution** 'throughout the enclosure. Proper ventilation provides both. One without the other is not adequate ventilation. Ventilation can be achieved by either mechanical or natural ventilation system.

Natural ventilation is the flow of air through open windows, doors, grilles and other planned building envelope penetration.[1] Natural ventilation is driven by wind force and buoyancy force.[1]Natural ventilation occurs primarily because of the pressure difference in wind pressure across a building and due to difference in inside and outside temperature. There are two types of wind driven natural ventilation: single sided and cross ventilation.[2] In single sided ventilation openings are on one side of building whereas in cross ventilation openings are located on two opposite sides of the building.

The relative importance of wind and buoyancy forces is obtained using the **Archimedes Number** (Ar).It is the ratio of Buoyancy Force and Wind Force.

$$A_r = \frac{Grashof}{Reynold^2} = \frac{\beta g H^3 \Delta T}{U^2 D^2}$$

When $A_r >> 1$, Buoyancy Force (natural convection) dominates

When $A_r << 1$, Wind Force (forced convection) dominates

Though the buildings employing NV relies mainly on wind driven ventilation, stack ventilation has advantage in moderate and cold climate. An ideal design for naturally ventilated building should include both.Natural ventilation is often expressed in "Air Change per Hour" (ACH).It is defined as number of volume of air in an enclosure replaced by fresh air in one hour which is given as:

$$ACH = 3600\ Q/V$$

where Q is volumetric flow rate in m^3/sec and V is Volume of building

As the Mega Warehouse buildings are very large they are generally not air conditioned, but often have sufficient heat and ventilation to provide a tolerable working environment. In today's world with global warming and energy cost increasing at an unprecedented rate air conditioning in a mega warehouse is not cost effective. Hence natural ventilation will be an energy efficient and healthy ventilation strategy.

Thermal Comfort Criterion

- Acc.to National Building Code of India 2016 Air Change Per Hour for Stores and Warehouse should be 3-6 [3]
- Acc.to ASHRAE 62.1-2010 Minimum Ventilation Rates for warehouse are : 5 L/s / person and 0.3 L/s /m^2 [4]

- Acc.to ASHRAE 55-2004 Thermal comfort can be extended from 26^0 C to29^0C at indoor air velocity of 0.7 m/s and up to 33.5 ^{0}C,at indoor air velocity of 1.5 m/s as the upper possible limit. [5]
- Acc.to Kang and Carillo, 2007 air velocity 0.25 to 0.5 m/s. is pleasant and can be extended up to 1 m/s. Possible maximum indoor air velocity in 1.5 m/s.[6]

Wind driven ventilation depends on pressure difference. Hence study of internal and external pressure distribution is important. Natural ventilation can be studied by different methods [7] such as analytical, empirical, small scale experiments, full scale experiments or numerical methods. CFD is more popular than other methods. CFD simulation of natural ventilation in a building is carried out using two methods- **Decoupled [8] and Coupled [9]** approaches. Analytically ventilation rate can be predicted by using Network method [10]

Though CFD models are extensively used for natural ventilation study their validation using experiments is also important. It has been suggested that in the absence of experimental data, ventilation rate by Network method can be used for validation of CFD predictions [11]. Network Method is an analytical method to predict a ventilation rate inside a building using external pressure distribution data. This method is based on Orifice equation and used under the following assumptions: the sealed body assumption ie. the pressure distribution on the building surface is not affected due to the presence of openings, airflow is due to static pressure difference alone and effect of the dynamic pressure due to the velocity component parallel to the wall containing inlet opening is insignificant (for oblique wind it may be significant) and can be neglected, stack effect is negligible, the static pressure inside the building is uniform, and the discharge coefficient of opening is assumed to be constant

For CFD simulation appropriate boundary conditions should be used to model inlet flow and Atmospheric Boundary Layer (ABL) [12].As CFD study of natural ventilation is very sensitive to various parameters such as turbulence model, computational domain, boundary conditions, one can follow the general guidelines provided for Computational Wind Engineering (CWE) [13]. In order to resolve wall boundary condition at ground, various solutions are suggested but there are contradictions [14]. The k-ε RNG model is found to be suitable to predict the performance of natural ventilation.[25,26]

Wind driven natural ventilation can be achieved by passive techniques (windows, atria, wind tower etc.) or active techniques (turbine ventilator, solar assisted turbine ventilator)[15].For stack ventilation roof ventilators and ridge ventilators are common for large industrial buildings.[15,16,17]

Wind driven ventilation depends on pressure difference. Hence study of internal and external pressure distribution is important. Some full scale experiments on gable roof buildings can be found in literature [18, 19].For wind driven cross ventilation the building geometry plays an important role. The length (in the direction of wind) of building should be less than five times of ceiling height.[2,20]Steeper roofs are better for ventilation.[21].The opening size,shape,and its configuration plays important role in cross ventilation.[22]

Many authors investigated cross ventilation in small flat roof [23-26] buildings. CFD study of ridge ventilator on cross ventilation is not dealt. Though some full scale CFD study in a large space [27] are observed ,the natural ventilation in a large gable roof mega warehouse building with multiple opening and ridge ventilator is not observed in the literature.

2. PROBLEM DESCRIPTION

Present work is a case study of a Mega Warehouse located at Embassy Industrial Park Bilaspur, India. It is a low rise gable roof building. The issue is related to the building ventilation. An internal and external view of this warehouse is shown in fig.1.It is of size 145 m x 76 m with longer surfaces facing North-South direction with eave height of 13 m and ridge height 15 m. Warehouse has louver windows of size 0.8 m in height all around at sill height 6 m. There are 15 doors on south wall having dimensions 5m×4m–2 nos. and 3m×2.5m – 13 nos. A ridge ventilator of throat size 1.5 m is used for rooftop ventilation. Annual weather report say that local winds speed during summer and monsoon is in the range of 3.4 to 5.5 m/s, and it changes to 1.6 to 3.4 m/s during winter season. Throughout the year the warehouse is not meeting the standard of ventilation rate. It is desired to assess the ventilation characteristics of the warehouse with existing provisions and recommend methods to improve it without major structural change.

(a) External view

(b) Internal View

Fig.1 Actual Warehouse

Therefore, the objective of the work is to assess the wind driven ventilation inside the warehouse by estimating the rate of ventilation under different wind conditions. The work is proposed to be executed in two parts.

Part I: Assessment of performance of existing provisions for the ventilation

To investigate wind driven natural ventilation (a) Without Ridge Ventilator, (b) With Ridge Ventilator

Part II: Assessment of performance of recommended provisions for enhanced ventilation

Investigation of flow structure developing outside and inside of such a big warehouse is complex, costly and time consuming from experimental view point. So a numerical investigation through a commercial software i.e. 'ANSYS Fluent' is selected for its execution. To assess the performance of cross flow natural ventilation, a Coupled CFD approach and Network method is also used for few cases.

3. MATHEMATICAL MODEL

Wind driven ventilation depends on wind behaviour and its interactions with the building envelopes. The openings and other air exchange devices-inlets, ventilators, chimneys etc. also play important role to decide the efficiency of the system. The flow evolves to be of low intensity isothermal flow with turbulence. The flow parameters can be captured mathematically by using mass and momentum balance considering steady, incompressible flow with negligible body force.

Mass Balance Equation:

$$div(\rho u) = 0 \tag{1}$$

Momentum Balance Equation:

$$div(\rho uu) = -\frac{\partial p}{\partial x} + div\left(\mu\, grad\, u\right) \tag{2a}$$

$$div(\rho vu) = -\frac{\partial p}{\partial y} + div\left(\mu\, grad\, v\right) \tag{2b}$$

$$div(\rho wu) = -\frac{\partial p}{\partial z} + div\left(\mu\, grad\, w\right) \tag{2c}$$

Turbulence Model:

It is observed from the literature review that the physics of such wind driven ventilation has been successfully captured by using K-ε turbulence model. [2, 28]

Turbulent Kinetic Energy (k)Equation:

$$\frac{\partial}{\partial t}(\rho k) + div(\rho k U) = div\left(\Gamma_k grad\ k\right) + G_k - \rho\varepsilon \quad (3)$$

$\Gamma_k = \mu + \frac{\mu_T}{\sigma_k}$ is turbulent diffusion coefficient of kinetic energy

$G_k = \mu_T \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\frac{\partial U_i}{\partial x_j}$ is rate of generation of turbulent energy

Dissipation rate is given by following transport equation:

$$\frac{\partial}{\partial t}(\rho\varepsilon) + div(\rho\varepsilon U) = div\left(\Gamma_\varepsilon grad\ \varepsilon\right) + C_{\varepsilon 1} G_k \frac{\varepsilon}{k} - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \quad (4)$$

$\Gamma_\varepsilon = \mu + \frac{\mu_T}{\sigma_\varepsilon}$ is turbulent diffusion coefficient of dissipation rate.

Boundary conditions

Boundary condition applicable to the present investigation are as follows.

Inlet: At inlet velocity of the flow is assumed to be logarithmic in nature due to the expected turbulence cause by ground [12]. It is given by, $U(y) = \frac{u_{ABL}^*}{k}\ln\left(\frac{y+y_0}{y_0}\right)$.Other inputs is computed by, Kinetic Energy, $K(y) = \frac{u^{*2}{}_{ABL}}{\sqrt{C_\mu}}$, Dissipation Rate :$\varepsilon(y) = \frac{u^{*3}{}_{ABL}}{k(y+y_0)}$.

Where y is the height co-ordinate, Aerodynamic roughness height y_0 =0.01m for open terrain, u_{ABL}* the atmospheric boundary layer friction velocity, k the von Karman constant (0.40) and C_μ=0.09 a model constant for k and ε . Velocity at ridge height H is U_{ref} which is calculated using y_0 =0.01 and U_{10}.

Outlet: Zero static pressure

Walls: No slip – smooth wall.

Ground: No slip with roughness k_s= 0.01m, Cs=0.5 [14]

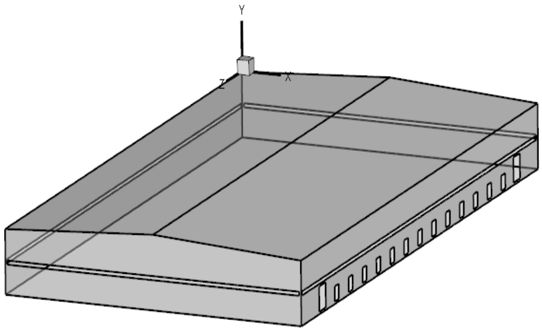
Other open boundaries: Symmetry condition

4. NUMERICAL EXECUTION

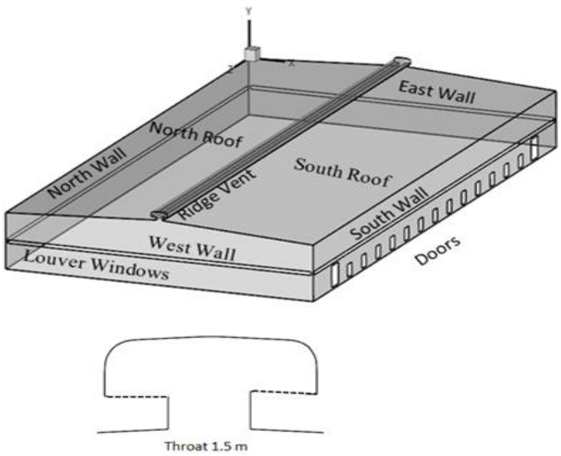
To execute the investigation actual warehouse is assumed to be an empty single zone space with wall of zero thickness. Ridge ventilator is assumed continuous. The effect of louver windows are equivalent to 50% of the open windows and only wind driven ventilation considered. To understand effect of ridge ventilator present condition of ventilation is studied using two cases.

Case1: Warehouse without Ridge Ventilator

Case 2: Warehouse with Ridge Ventilator



(a) Without Ridge Ventilator



(b) With Ridge Ventilator

Fig.2. Simplified CFD model of Mega Warehouse

Size of warehouse: 145m×76 m in plan with eave height of 13 m and ridge height 15m

Computational Domain:

In order to investigate the flow structure developing in and around full scale of the building, a computational domain as shown in Fig.3 is considered based on the best practice guidelines suggested by Franke [14]. The distances were 5H from the building model to the sides and top of the domain, and 20 H for the downstream length to care of the vortices, where H is the ridge height of building. The upstream length of the domain is reduced from 5H to 4H to decrease the stream wise gradient.

Whole domain is discretized using polyhedral and prism elements. Prism elements of first height 0.02 m are imposed near solid boundaries to capture boundary layer effect

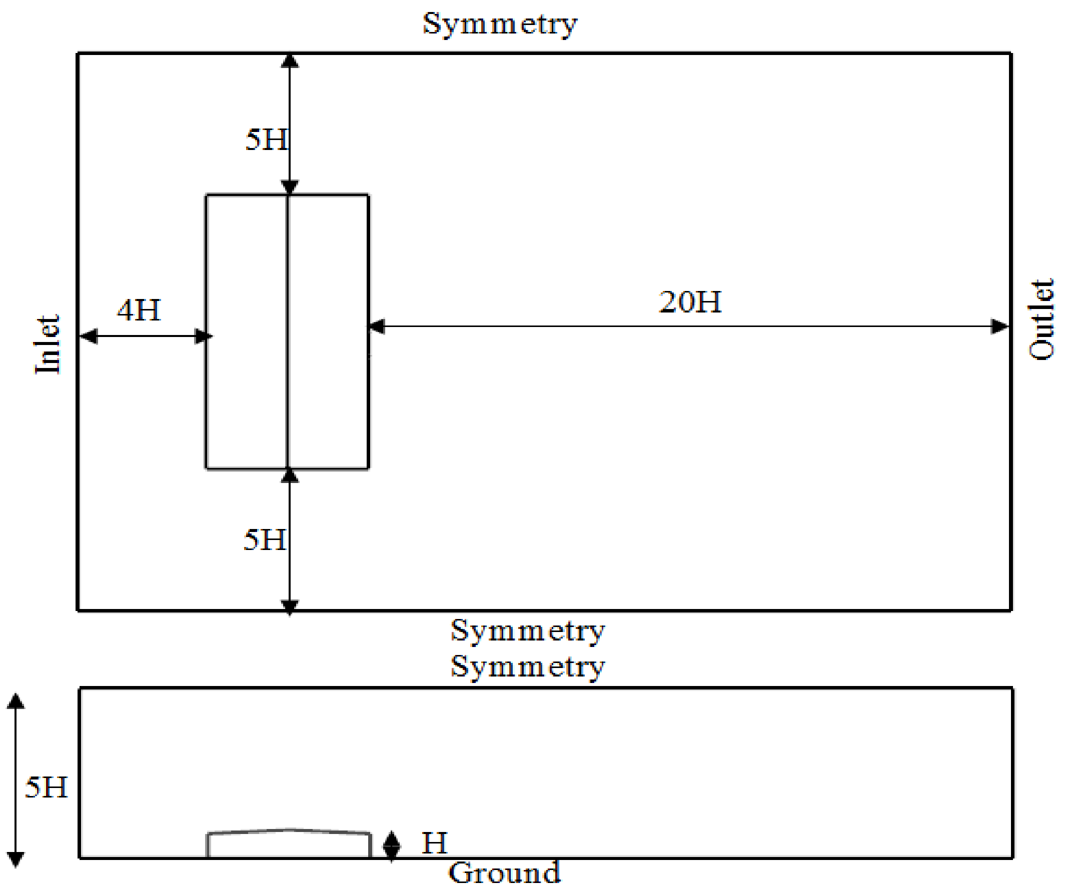


Fig.3. Computational domain

Solution Method:

Material properties of air at STP is considered for investigation, Using finite volume approach the variation of pressure and velocity is treated by SIMPLE algorithm. The input boundary condition is implemented through an UDF. Discretized data of pressure is interpolated by using standard scheme. Second order upwind scheme is used to treat the convective effect of momentum, turbulent kinetic energy and turbulent dissipation rate. The convergence criterion is set to be 10^{-4}.

5. RESULT AND DISCUSSION

This section present results in two parts - Part I: Assessment of performance of existing provisions for the ventilation, and Part II: Assessment of performance of recommended provisions for enhanced ventilation.

Before executing the investigation of the above two parts as discussed in the problem definition, the computational models of flow for proposed cases is authenticated by an experimental data of distribution of external pressure coefficients collected by doing investigation on similar kind of gable roof building byR.P.Hoxey et.al. [18]. The author investigated distribution of external pressure coefficients on a gable roof building of size 24m×13m×4m with 10^0 gable roof angle with reference velocity 8m/s. The authors used both methods- full scale experiment and numerical. Author also noted the drawbacks of k-**ε** standard turbulence model. Hence present numerical investigation is carried out using k-**ε** RNG turbulence model.

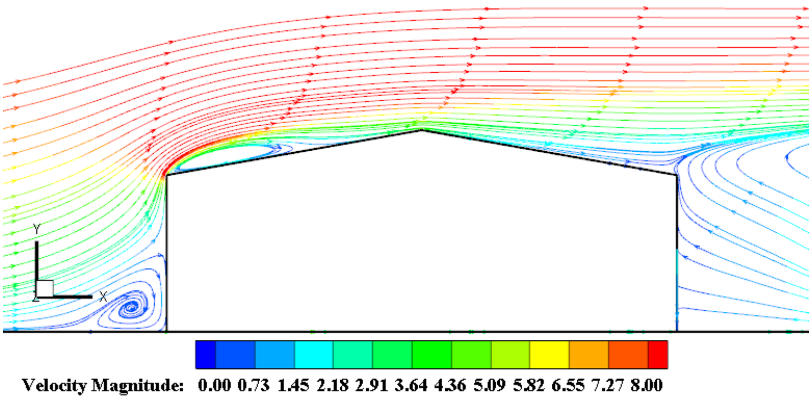


Fig.4 (a). Streamlines around the building,

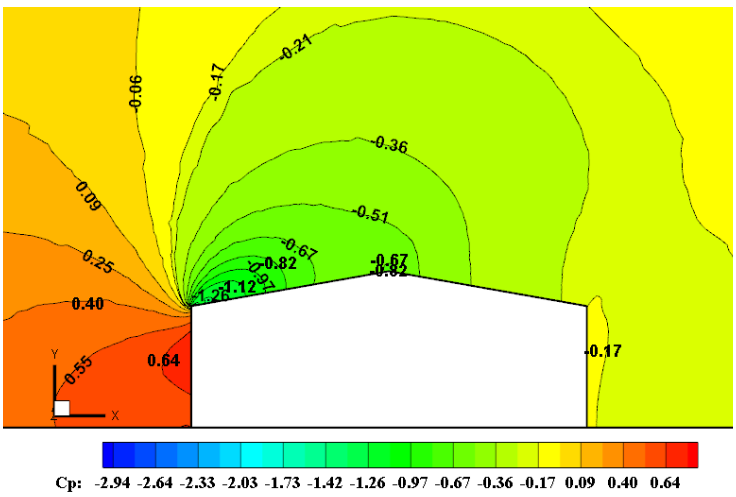


Fig.4 b) Distribution of Cp on mid plane (Z=12m)

Outcome of the investigations are depicted in fig.4. through the flow structure evolved around the building and distribution of pressure coefficient on a mid-vertical plane. Stagnation zone, flow separation and its reattachment along with the vortices can be observed clearly which finally effects the variation of pressure coefficient.

Figure.5 shows a comparison of C_p value of present numerical work with experimental work of R.P.Hoxey et.al. [18] plotted for the vertical mid plane (Z=12m). A close agreement with the experimental and present numerical work can be observed.

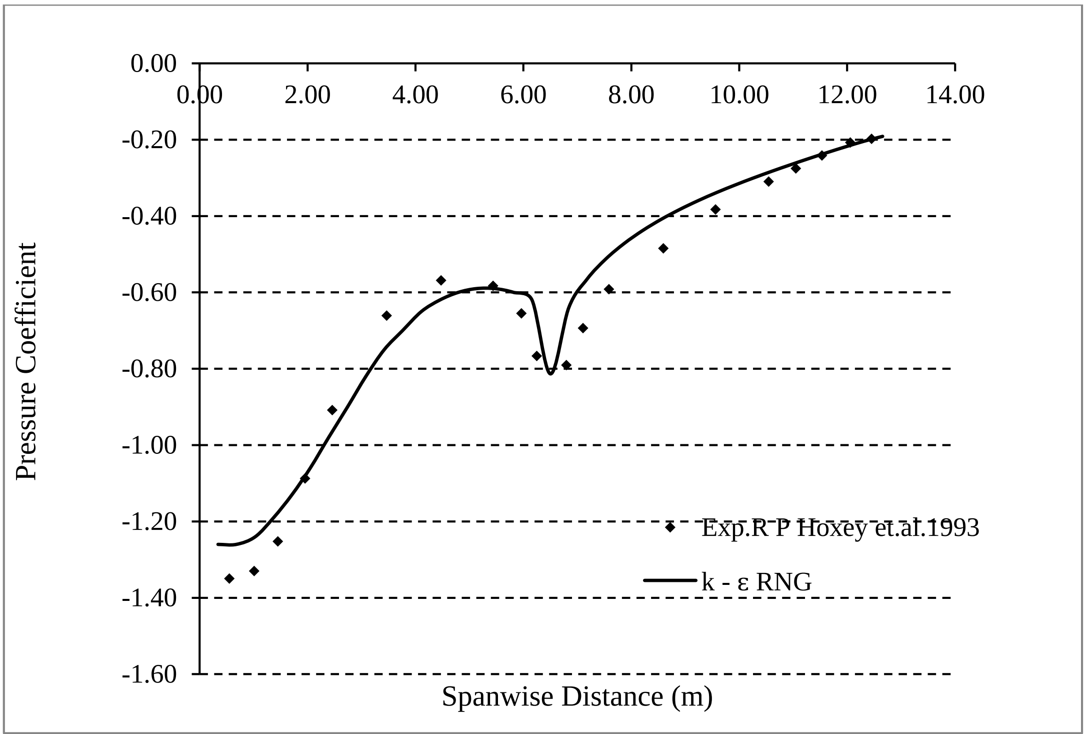


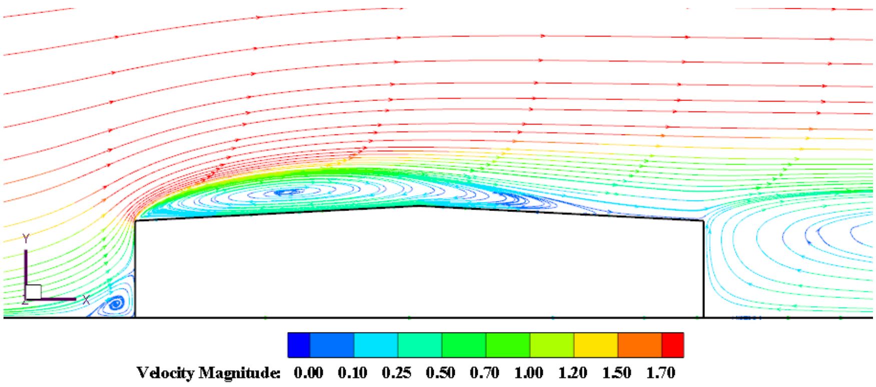
Fig.5. Validation of numerical model

After validation of mathematical and numerical models, following sections deals with results and discussion of the case under investigation.

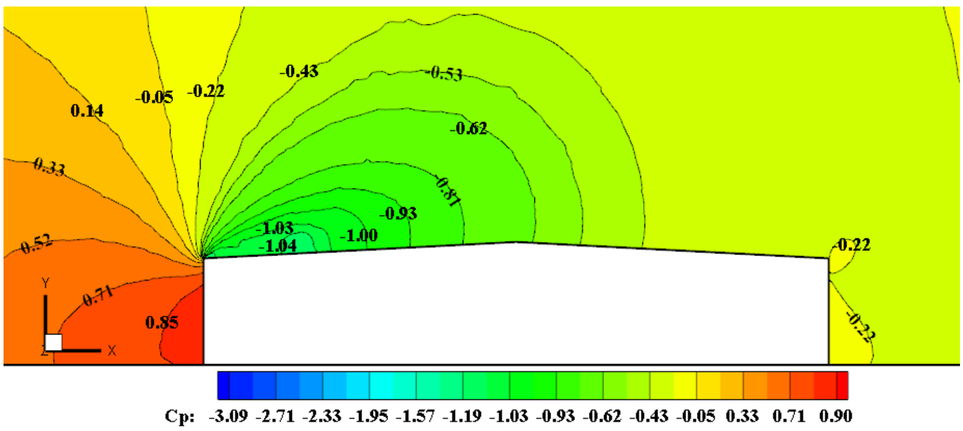
Part I: Assessment of performance of existing provisions for the ventilation

a. Predicting ventilation rate (without ridge ventilator) with the help of external flow analysis

Network method is used to predict the ventilation rate in the warehouse with the help of the data obtained from external flow analysis carried out using CFD approach. A distribution of streamlines is presented Fig. 6(a). It can be seen from the figure that there is only one flow separation at windward corner and there is no flow separation at ridge as the gable angle is less than 10 degrees. There is reattachment of flow towards the leeward wall. The distribution of pressure coefficient on a mid-vertical plane for a reference wind speed of 1.6 m/s is shown in Fig. 6(b). Using the data (C_p) obtained from external CFD analysis, the network method computes the ventilation rate. The calculation of ventilation rate at wind velocity 1.6 and 3.4 m/s comes to be 124.39 and 263.99 m^3/s respectively.



(a) Streamlines

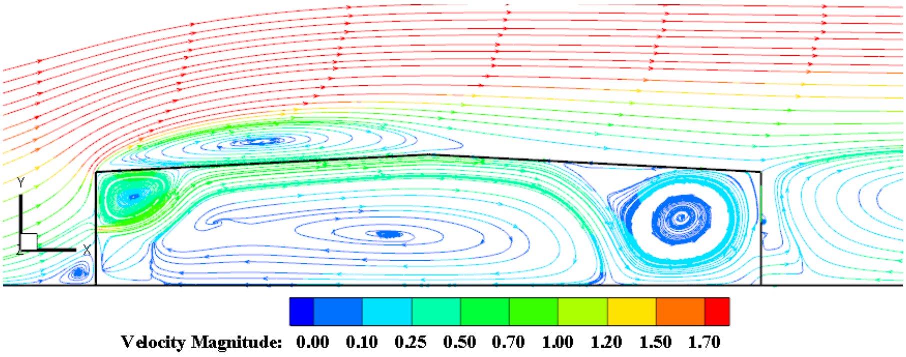


(b)Pressure coefficient

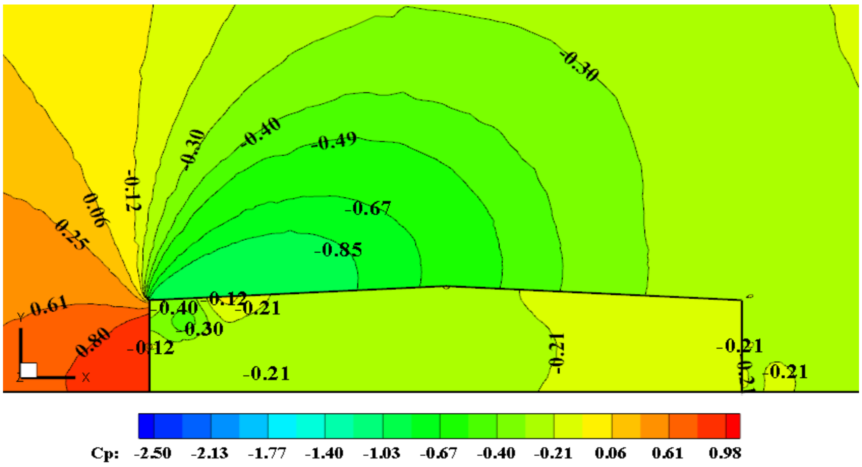
Fig.6. CFD prediction for external flow analysis at U_{10} = 1.6 m/s

b. Predicting ventilation rate of warehouse (without ridge ventilator) with internal-external flow analysis

The results of coupled flow analysis (external and internal together) is presented here. The distribution of velocity and pressure can be observed in fig. 7. The wind reaches to windward face of the building. Some part of it enters to interior causing ventilation and remaining flow moves over the building causing flow separation at front and back end of the building like previous case. Flow through the internal space shows the formation of two big internal vortices along good flow in major part of the space. The recirculation zone behind the building creates a low pressure zone which is main cause of cross ventilation inside the building.



(a) Streamlines



(b) Pressure Coefficient

Fig. 7.CFD prediction for coupled flow analysis for U_{10} = 1.6 m/s

The distribution of pressure coefficient around and inside the warehouse at vertical mid plane is presented. The external pressure coefficient distribution is same as predicted by external flow analysis. This is due to low porosity of wall (window to wall ratio is less than10 %). Though the internal pressure coefficient is more (-0.30 to – 0.41) at eave corner near openings, pressure distribution inside the warehouse is nearly uniform (-0.21to-0.22). This agrees with assumption made in Network method. Table 1 compares the cross ventilation rate predicted by Network Method and Coupled CFD Approach. It can be observed that there is an excellent agreement between both.

Table 1: Comparison of ventilation rate using Network Method with Coupled CFD Approach

Wind speed	Q [m^3/s] (Network Method)	Q [m^3/s] (CFD Coupled Approach)	Discrepancy %
U_{10} = 1.6 m/s North	124.39	124.85	-0.0037 %
U_{10} = 3.4 m/s North	263.99	265.14	-0.0043 %

In order to get better understanding of flow pattern developing inside the warehouse due to wind driven ventilation, the velocity contours at different planes along the direction of flow is shown in fig. 8. It could be seen that the bottom region of the warehouse has weaker intensity of flow compared to upper regions. It indicates as poor ventilation pattern.

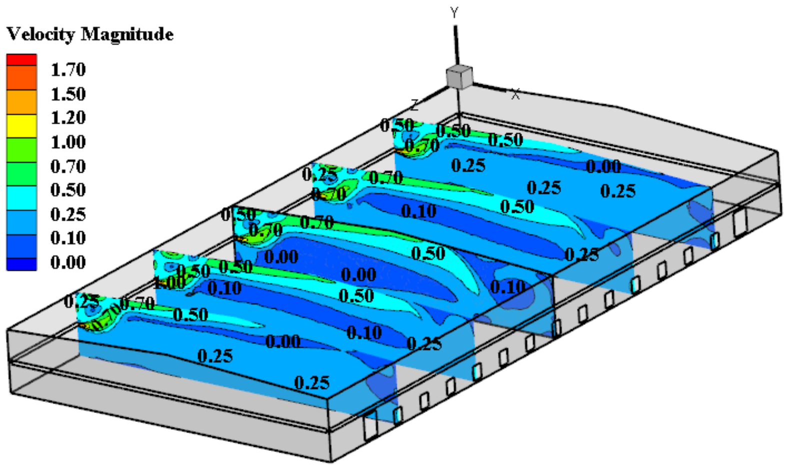
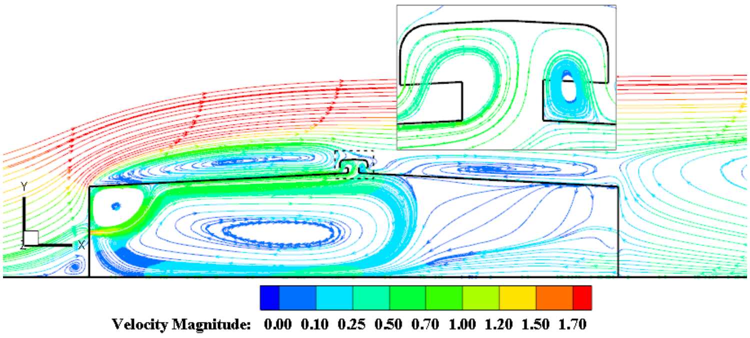


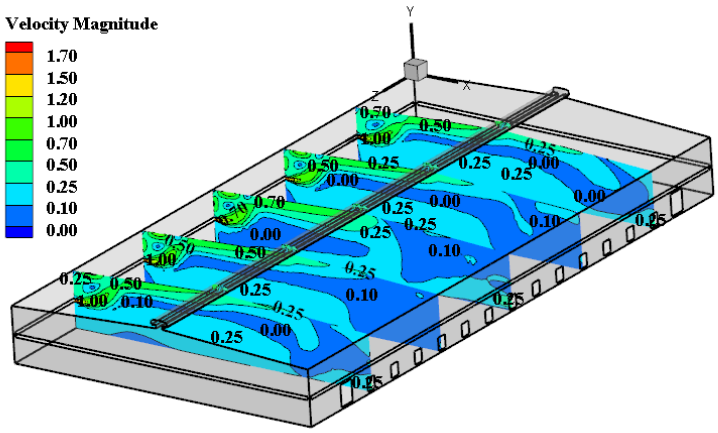
Fig.8. Velocity contour on different vertical planes for U_{10} = 1.6 m/s

c. Predicting ventilation rate of warehouse with ridge ventilator

To assess the wind driven ventilation in presence of ridge ventilator as depicted in Fig.1, a simulation is performed at wind speed U_{10} = 1.6 m/s. Streamlines of flow outside and inside with ventilator is shown in Fig.9. The air enters through the inlet and directly goes outside through ridge vent. The ridge vent form virtual stream tubes due to which there is no diffusion of air towards the leeward side. As compared to case without ridge the bottom low velocity recirculation zone is shifted to centre but the velocity towards leeward side remains low.



(a) Streamlines around and inside the warehouse,



b) Velocity Contour on different locations

Fig.9 CFD prediction for coupled flow analysis with ridge ventilator for U_{10} = 1.6 m/s

Internal flow distribution on different vertical planes along the length of warehouse is shown in Fig.10. Major portion of the warehouse particularly at the bottom remain stagnated. The rate of ventilation with ridge ventilator increases from 125 m^3/s to 135 m^3/s.

Considering 50% blockage area for Louver windows the actual ventilation rate is half of the ventilation rate achieved with open windows. Hence the actual ACH is 1.58 which is far less than the recommended minimum ventilation rate by NBC India. The airflow distribution is also poor. Hence the ventilation rate as well as airflow distribution need to be improved.

Part II : Alternative configuration for ventilation improvement

Part II consists of three sections. Section A and B presents results of effect of ridge vent design and effect of louver location on ventilation respectively. Improvements in ventilation with alternative configuration are presented in section C.

A) Effect of Ridge ventilator design on ventilation

Although summer ventilation is provided by wind driven cross ventilation with large sidewall openings, the ridge vent is important in winter for stack ventilation. The warm air naturally flows upward and not inclined to move downward. For better ventilation in winter the ridge ventilator should have open design and minimal interference to natural rise of hot air. The ridge vent with cap prevents warm air from naturally flowing upwards when there is low wind speed in winter.

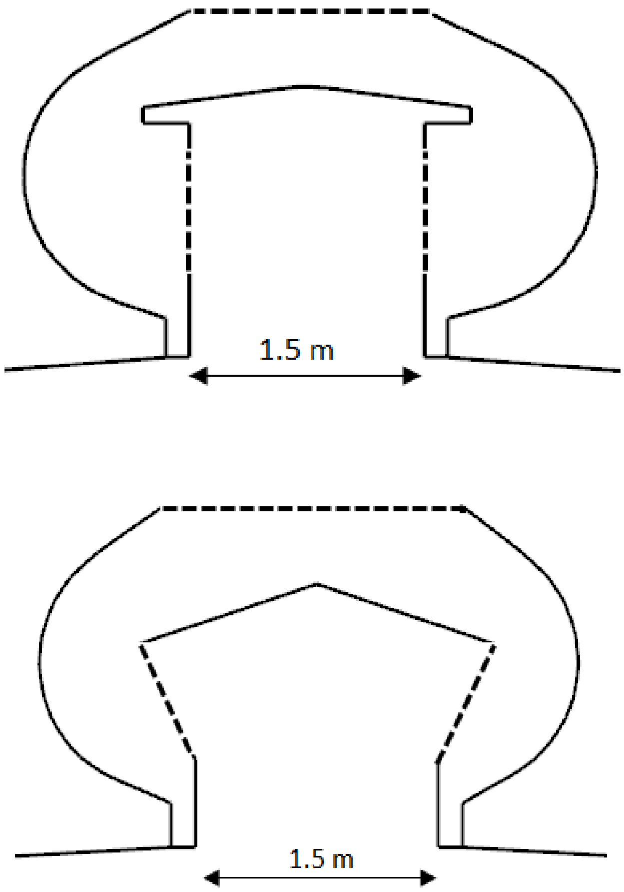
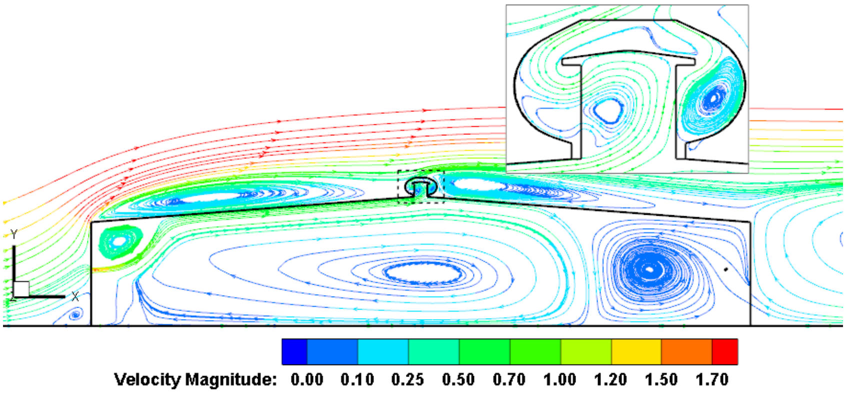
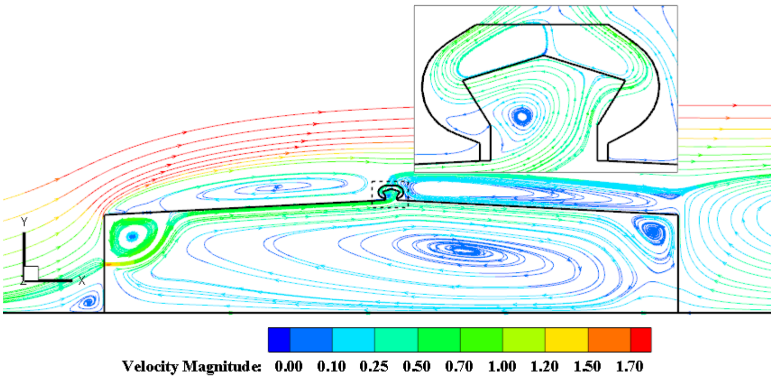


Fig.10 Ridge vents design 1 and 2

As open ridge cannot be used for warehouse due to rain penetration, two new ridge ventilator designs are considered. CFD simulations with new ridge vents are presented in fig.11.The performance of both new ridge vents are better than the existing ridge vent. The bigger middle recirculating zone indicates better ventilation as compared with existing ventilator. The new ridge vents does not form virtual stream tubes hence airflow diffuses towards the leeward side. It could be seen from fig.11 second ridge vent design has better performance than first one. Hence second ridge vent is considered for new configuration.



(a) Ridge vent 1



(b) Ridge vent 2

Fig.11 CFD prediction of airflow distribution with new ridge vents for U_{10} = 1.6 m/s

B) Effect of louver window location on ventilation

Summer ventilation is provided by cross ventilation as the temperature difference between inside and outside is less as compared to winter for stack ventilation. For cross ventilation openings of a building plays important role. The position, size and shape of openings have significant effect on ventilation rate and air distribution inside the building. As per ECBC [29] Window to Wall ratio <40% and as per IGBC [30] percentage of open area to the total carpet area is for regularly occupied area (> 100 sq. m.) should be 12 %. In the existing structure the ventilation rate is very less due to less wall porosity. The window to wall ratio is only 6.15% and less percentage of openable area to the total carpet area i.e. 6.4% only.

In order to improve ventilation rate an adjustable louver openings of height 2 m (existing 0.8m) is proposed all around the warehouse. The effect of this change in opening on ventilation and air flow distribution at sill heights of 3m, 6m and 9 m with new ridge vent are investigated numerically. CFD simulation results for three cases with wind speeds 1.6 m/s are presented as follows:

Table 2. Ventilation rate for louver window at different sill height

Louver Sill Height	Volume flow rate (m^3/s)	ACH with open window	ACH with Louver window [50 %]
3m	341	7.96	3.98
6m	335	7.82	3.91
9m	327	7.63	3.82

Table 2 shows CFD predicted ventilation rate for louver window at different sill height. It is evident from the table that ventilation rate decreases with increase in sill height. With increase in opening size ventilation rate is improved from 1.58 ACH to 3.98 ACH which just satisfies the minimum requirement of 3 ACH.

Fig 12-14 shows airflow distribution and velocity contour for three cases. It could be seen from figures that the airflow distribution for three cases is approximately same but the magnitude of velocity changes with location of louver window. In all three cases central zone remains stagnant and velocity at given sill height ranges from 0.5 m/s to1.2 m/s. Louver opening at sill height 3 m provide better ventilation for bottom occupied zone.

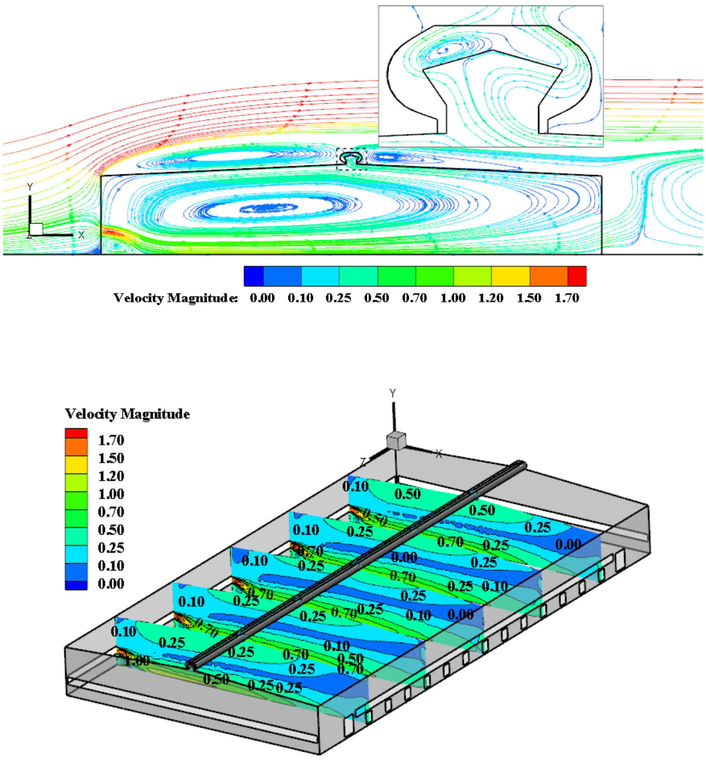


Fig.12 Streamlines and Velocity Contour with sill height 3m for U_{10} = 1.6 m/s

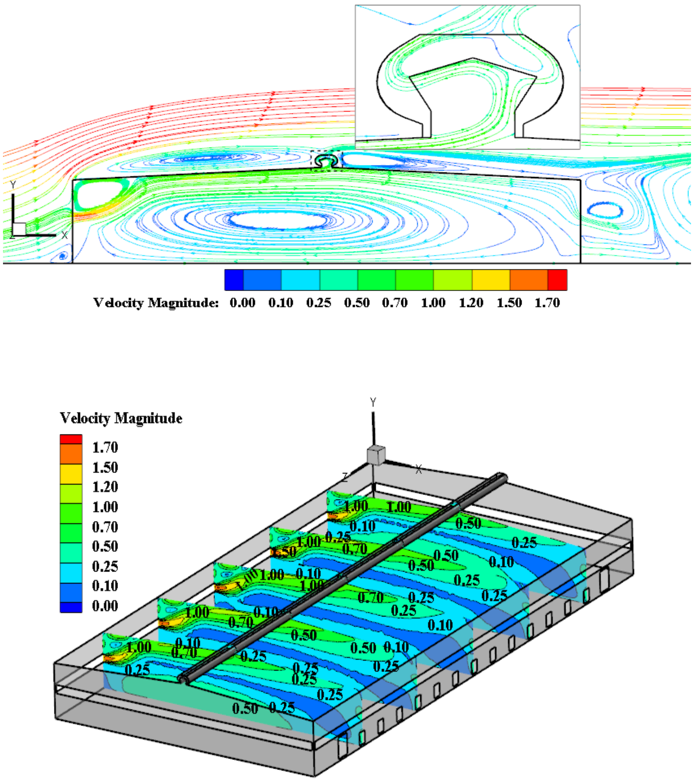


Fig.13. Streamlines and Velocity Contour with sill height 6m for U_{10} = 1.6 m/s

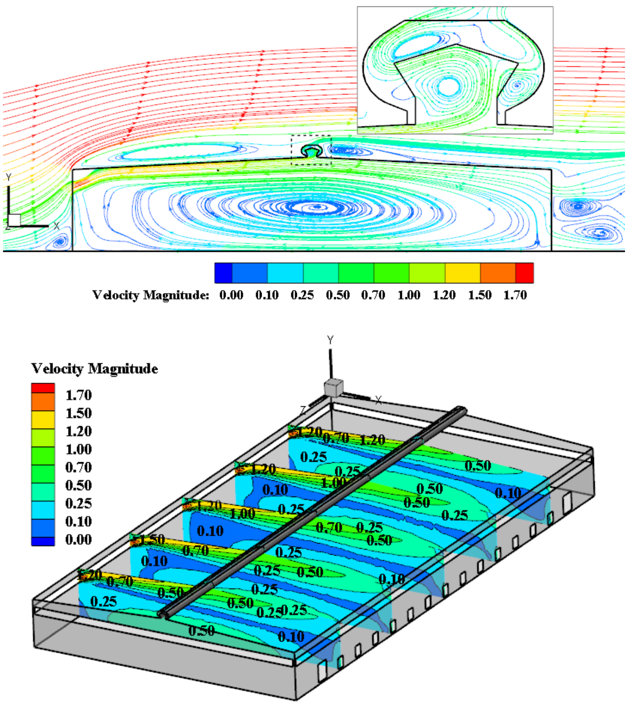


Fig.14. Streamlines and Velocity Contour with sill height 9 m for U_{10} = 1.6 m/s

C) Ventilation with alternative configuration for three wind speeds

Further, in order to improve ventilation rate and airflow distribution, two louver openings are provided on north and south wall. One at sill height 3 m of 2 m height and other eave opening at sill height 9.5 m of 1 m height. CFD simulation results for three wind speeds 1.6 m/s 3.4 m/s 5.5 m/s are presented as follows:

Table 3. Ventilation rate for three wind speeds with two louver openings

Wind speed	Volume flow rate (m^3/s)	ACH with open window	ACH with Louver window [50 %]
1.6	504	11.76	5.88
3.4	1070	24.97	12.48
5.5	1727	40.29	20.15

It is evident from table 3 the ventilation rate increases from 5.88 ACH to 20.15 ACH. Though the ventilation rate for 3.4 m/s and 5.5 m/s are high as compared to the required, these ventilation rates are predicted for empty warehouse. Fig 15, 16 and 17 shows airflow distribution and velocity contour for 1.6 3.4 and 5.5 respectively. The velocity distribution in lower half is better as compared to upper half. The velocity at bottom reaches up to 3 m/s for wind speed 5.5 m/s which may cause disturbance. But with stacking in warehouse velocity inside the warehouse will decreases.

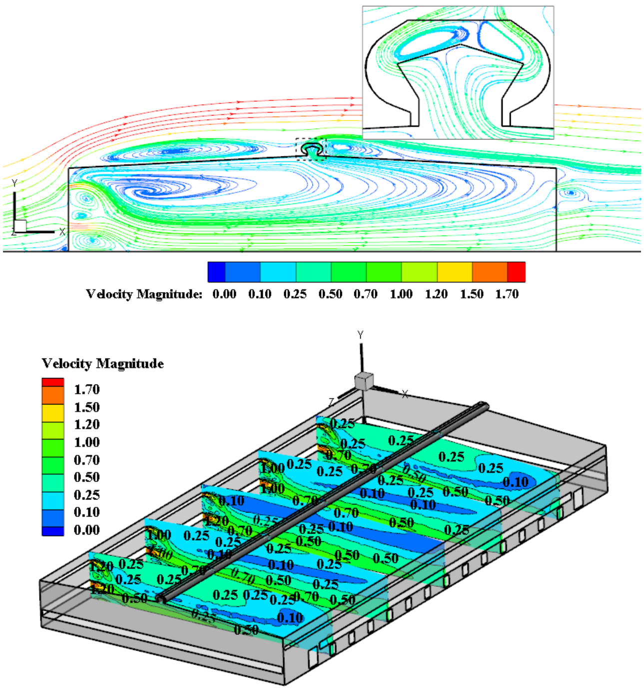


Fig.15. Streamlines and Velocity Contour for U_{10} = 1.6 m/s

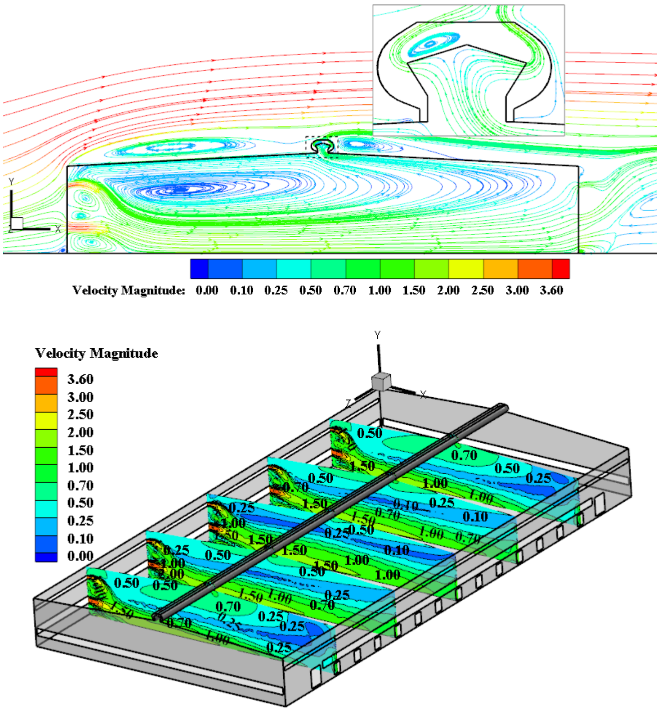


Fig.16. Streamlines and Velocity Contour for U_{10} = 3.4 m/s

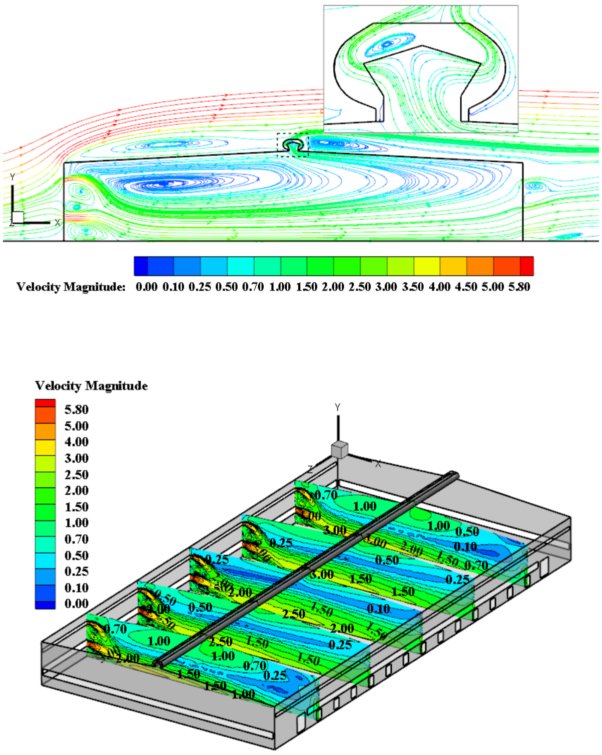


Fig.17. Streamlines and Velocity Contour for U_{10} = 5.5 m/s

6. CONCLUSION

This study presents a case study to improve the natural ventilation and air distribution in the existing structure of the warehouse. The conclusions are as follows:

1) For original warehouse building without any modifications with minimum wind speed 1.6 m/s the air change rate (ACH) of 1.58 is possible. Compared to the standard it is very low which is found to be because of low porosity of wall ie. insufficient opening for natural ventilation.
2) An increase in ventilation rate of about 7 to 8 % is observed by installing a ridge ventilator, but with existing configuration of ridge ventilator, most of the entrained air is observed to be directly reaching to ridge vent without effecting the internal space significantly
3) Increasing the size of louver window and decreasing sill height with one more eave opening on north and south wall significantly improves the ventilation. With these modifications ACH for minimum wind speed 1.6 m/s increased to 5.88 which satisfy the requirement. For wind speed 3.4 m/s and 5.5 m/s attained ACH is 12.48 and 20.15 respectively. Also inside velocity in most of the region is 0.25 m/s which is better than the present condition. For wind speed 3.4 m/s inside air velocity ranges from 0.25 m/s to 0.70 m/velocity at ridge height reaches up to 1.5 m/s. For wind speed 5.5 m/s inside velocity is little bit higher reaches up to 3 m/s which may cause some disruptions.

ACKNOWLEDGEMENT

I am grateful to Mr. Amol Banpatte, AGM at Tata Consulting Engineers Ltd (TCE), Pune for providing me an opportunity to work on this project. I must also express my special gratitude to Mr. Anand Jambhale, Manager-Mechanical (TCE) for his time to time help.

REFERENCES

[1] ASHRAE Handbook of Fundamentals, American Society of Heating, Refrigeration and Air-Conditioning Engineers, Atlanta, 2001 (Chapter 26).

[2] Awbi H.B. Ventilation of buildings. Spon Press, London, 2003.

[3] National Building Code of India 2016, PART 8 Building Services, Section1 Lighting And Ventilation, Sub -Section 5 – Ventilation

[4] ASHRAE 62.1. Ventilation for Acceptable Indoor Air Quality 2010

[5] ASHRAE (Std-55,2004), American Society of Heating, Refrigeration and Air-Conditioning Engineers, Atlanta.

[6] Kang, B. and Carillo, L. Indirect Passive air flow system, University of Texas at Austin, School of Architecture, Seminar in sustainable architecture,UTSOA,2007.

[7] Q. Chen, "Ventilation Performance Prediction for Buildings: A Method Overview and Recent Applications," Building and Environment, 44(4), 2009, pp. 848–858.

[8] Meroney RN. CFD Prediction of airflow in buildings for natural ventilation. Proceedings 11th Americas Conference on Wind Engineering, San Juan, Puerto Rico 2009:1-11

[9] R. Ramponi, B. Blocken, CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters, Build. Environ. 53 (2012) 34–48

[10] Bahadori M, Haghighat F. Passive cooling in hot arid regions in developing countries by employing domed roofs and reducing the temperature of internal spaces. Building and Environment 1985;20(2):103–13

[11] O.S. Asfour, M.B. Gadi, A comparison between CFD and network models for predicting wind-driven ventilation in buildings, Building and Environment 42 (12) (2007) 4079–4085.

[12] Richards, P.J.,Hoxey,R.P.,1993.Appropriate boundary conditions for computational wind engineering models using the k–ε turbulence model. Journal of Wind Engineering and Industrial Aerodynamics 46 and 47, 145–153

[13] J Franke, C Hirsch, AG Jensen, HW Krüs, M Schatzmann, PS Westbury, Recommendations on the use of CFD in wind engineering Cost action C 14, C1, 2004.

[14] Blocken, B.,Stathopoulos, T., Carmeliet, J., 2007. CFD simulation of the atmospheric boundary layer: wall function problems. Atmospheric Environment 41, 238–252.

[15] Khan N,SuY,Riffat SB. A review on wind driven ventilation techniques. Energy Build 2008;40:1586–1604

[16] Riskowski GL, Ford SE, Mankell KO. Laboratory measurement of wind effects on ridge vent performance. ASHRAE Trans 1998;104(1):495e501.

[17] T. Kobayashi, Tomoyuki Chikamoto Evaluation of Ventilation Performance of Monitor Roof in Residential Area based on Simplified Estimation and CFD Analysis Building and Environment 63 20 - 30 2013.05

[18] Hoxey R.P., Richards, P.J., 1993. Flow patterns and pressure field around a full-scale building. J. Wind Eng. Ind. Aerodyn.

[19] Stathopoulos, T., Surry, D., and Davenport, A. G. (1979). Internal pressure characteristics of low-rise buildings due to wind action.'' *Proc., 5th Int.Conf. on Wind Engrg.*, Vol. 1, 451–463

[20] Chia-Ren Chu, Bo-Fan Chiang. Wind-driven cross ventilation in long buildings. Building and Environment. 80, 2014, 150-158.

[21] Perén JI, van Hooff T, Leite BCC, Blocken B. CFD analysis of cross-ventilation of a generic isolated building with asymmetric opening positions: impact of roof angle and opening location. Building Environment 2015;85:263–76. doi:10.1016/j.buildenv.2014.12.007

[22] B. Givoni, Laboratory study of the effect of window size and location on indoor air motion. *Architectural Science Reaiew 8, 4245 (1965).*

[23] Karava P, Stathopoulos T, Athienitis AK. Impact of internal pressure coefficients on wind-driven ventilation analysis. Int J Vent 2006;5:53-66.

[24] Karava, P., Stathopoulas, T., Athienitis, A., K., Airflow assement in cross-ventilated buildings with operable facade elements, Building and environment, 2011

[25] Evola, G. and Popov, V.: Computational analysis of wind driven natural ventilation in buildings, Energy and Buildings, 38(5), 491-501,2006

[26] M. Z. I. Bangalee, S. Y. Lin, and J. J. Miau, “Wind driven natural ventilation through multiple windows of a building: a computational approach,” Energy and Buildings, vol. 45,pp. 317– 325, 2012.

[27] Van Hooff T, Blocken B. Coupled urban wind flow and indoor natural ventilation modelling on a high resolution grid: A case study for the Amsterdam Arena stadium. Environ Modell Softw 2010;25:51-65.

[28] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method ~Longman Scientific and Technical, 2007

[29] Energy Conservation Building Code 2017 ,Bureau of Energy Efficiency, New Delhi, India.

[30] Indian Green Building Council (IGBC).