VOL. 7, NO. 11, NOVEMBER 2012 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

SIMULATION OF INDOOR AIR FLOW FOR A ROOM WITH WINDOWS AT THEIR ADJACENT WALLS UNDER VARIOUS WIND FLOW DIRECTION USING CFD

D. Prakash[1] and P. Ravikumar[2]
[1]Department of Mechanical Engineering, Anna University, Trichy, Tamil Nadu, India
[2]Department of Mechanical Engineering, St. Joseph College of Engineering and Technology, Thanjavur, India
E-Mail: dee_prakas@yahoo.com

ABSTRACT

The present paper is about to study the effect of wind direction on thermal comfort in an office room with windows at their adjacent walls. As a preliminary work, a suitable turbulence model was selected by analyzing the various turbulence models like Standard K-ε, Renormalization-group (RNG) K-ε and Realizable K-ε model and their closeness to the experimental results are predicted. From the preliminary study, Standard K-ε model is selected as a suitable turbulence model for this thermal comfort study. The CFD simulation is also checked for grid independence test. Secondly, the effect of wind direction on thermal comfort was analyzed by including the fluctuation of wind direction. The CFD simulated mass flow rate for all wind directions are compared with the network work model and the maximum discrepancy obtained is 9.97% which is within the accepted level. The indoor air flow pattern, temperature distributions, are predicted and these results are very much useful to identify the most comfort and un comfort zones prevailed inside the room for various wind directions.

Keywords: natural ventilation, indoor air, CFD, K-ε turbulence model.

1. INTRODUCTION

Ventilation involves the supply of fresh air in to the building and increases the thermal comfort for the occupants. Ventilation can be achieved by natural provisions and by mechanical systems. Natural ventilation using ambient air movement is an economic option for enhancing indoor room air circulation and it reduces energy requirement and associated cost. Natural ventilation is enhanced by wind pressure and buoyancy force. Mendel and Fisk [1] identified an increased risk of health symptoms among building occupants that was attributable to the use of mechanical ventilation. Fisk and Rose field [2] found that natural ventilation improves both the indoor environment and ultimately worker's productivity. The creation of good natural ventilation provides, comfortable living and working conditions without consumption of electrical energy. Because of these benefits, natural ventilation is coming popular and many research works have been encouraged in this area. In this contest, present work is focused on the natural ventilation in the room with windows at their adjacent walls. Most of the existing literature pertains only to single -sided ventilation or cross ventilation (Window openings at the opposite walls). However in real case, locating the windows at the opposite wall in a room attached with the building is not possible. Instead of that, the windows are located only at their adjacent walls and hence there is a need to study the indoor airflow characteristics in a room with windows at the adjacent walls by natural ventilation [3]. Natural ventilation was driven by wind pressure and temperature difference. Among the two driving factors, wind pressure contributes more for the indoor ventilation and hence the effect of wind pressure on thermal comfort was included in this paper. The wind pressure is fluctuating continuously by its direction of impact over the wall surface and its magnitude. The wind (breezes) enters through a window at the windward side window, brings fresh air and pulling the staled air to exit through adjacent opening of the room. This is due to the positive pressure on the wind ward side and negative pressure on the leeward side. Knowing of wind flowing pattern and correct orientation of house, placement of windows, doors and other openings is very crucial for providing good ventilation. The air flow pattern inside the room is complex and is characterized by multi-flow features such as laminar boundary layers, highly turbulent diffuser jets and low turbulent flow [4]. The techniques available to study the air flow are based on theoretical approach, experimental technique, and numerical or computational approach. The theoretical approaches will only deal with simple cases of flow. Some simplified empirical models are also available to calculate the air flow pattern. Dascalaki *et al.*, [5] referred some models such as AIRNET, BREEZE, COMIS, ESP, NORMA and PASSPORT-AIR to analyse the air flow. But these models are limited to simple case of analysis that includes laminar flow only. Walker and White [6], Nielsen and Olsen [7], Dascalaki *et al.*, [5], Zeidler and Fitzner [8], used tracer gas measurement technique for their air flow studies. The study of complex flow patterns by experimental approach is highly infeasible and it provides flow pattern details only at a specified location. But, CFD investigates complex flow structure and provides detailed result at every location in the flow domain. With the aid of CFD, Hoang *et al.*, [9], Huo *et al.*, [10], Oliver rouaud and Michel Havet [11], Doosam song and Shinsuke kato [12] performed the air flow analysis in the buildings. With all these information, the present article attempts to implement CFD technique to study the effect of wind

VOL. 7, NO. 11, NOVEMBER 2012 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

direction on the thermal comfort prevailed inside the room with windows at the adjacent walls.

2. PRELIMINARY INVESTIGATION AND CFD VALIDATION

In the preliminary investigation, selection of suitable turbulence model and result independent gird size was identified. A cube of size 5m x 5m x 5m with external atmospheric zone of 30m x 20m x 30m was modelled in the gambit software. Tetrahedral T grid element was used to mesh the entire fluid domain. Structured grid of size ranging from 0.8 to 0.4 was tested in the grid independency check with standard k-ε turbulence model. The pressure coefficients along the cube face are predicted for all the grid sizes and are shown in Figure-1.

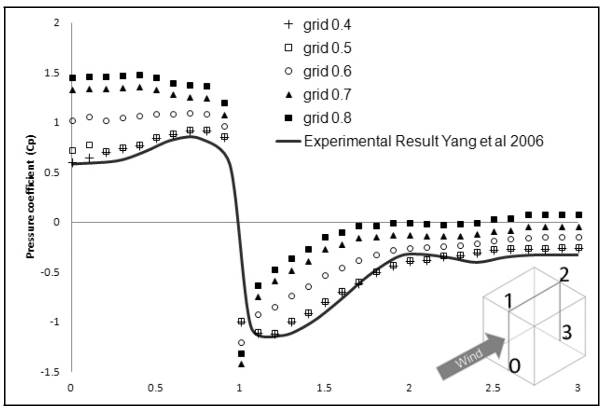

Figure-1. Grid independence check.

From this figure it is evident that, for grid of size 0.8 the pressure coefficient values are having more discrepancy with experimental values [13]. Further reducing the grid size to 0.7 and 0.6, the pressure coefficient (Cp) values are converging towards the experimental values. Even though, the pressure coefficient values are converging towards the experimental values but their discrepancy with experimental value are still significant. Again reducing the grid size to 0.5 and 0.4, the pressure coefficient values are having close agreement with the experimental values. Also, both grid size 0.5 and 0.4 gives the same value of pressure coefficient along the wall surface with a negligible difference in the order of 10^{-3}. Since the CFD simulated pressure coefficient values are independent for the grid sizes 0.5 and 0.4, these mesh sizes are identified as grid independent. By considering the time consumption for solving the fluid domain equations, grid size of 0.5 is identified as the suitable mesh size which is employed in the future analysis.

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of fluid flow problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of turbulence model, a detailed investigation on the effect of various turbulence model in the building ventilation is required. Hence four separate analysis are performed on the same cube with various turbulence models like Standard K-ε, Renormalization-group (RNG) K-ε, Realizable K-ε and Large eddy simulation (LES) model. The pressure coefficient values along the cube faces are predicted for the above four turbulence model cases and their closeness with the experimental values are shown in Figure-2.

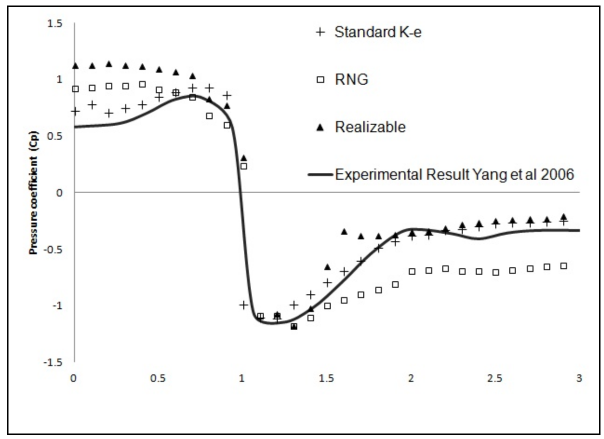

Figure-2. Comparison of turbulence models.

From this figure, a wide variation in the pressure coefficient values with the experimental values are identified throughout the wall surface of the cube for Realizable K-ε and RNG turbulence model. On the other hand, standard K-ε models are having close agreement with the experimental values. Hence, the standard K-ε model is identified as the suitable turbulence model for this study and the same was employed in the future analysis. After identifying the grid independent mesh size and a suitable turbulence model, the main aim of paper was focussed and discussed in the forth coming section. The governing equations for fluid flow are conservation of mass, momentum and energy. These governing equations are in the partial differential equation form and are converted in to suitable algebraic equation by discretization process. In the present research work Finite volume method of discretization process was employed in which the fluid domain was subdivided in to cell with a central node. The fluid flow variables at the central node were predicted and by interpolation method the same was calculated at the cell surface.

3. TEST CASE ROOM SPECIFICATION AND CFD SOLUTION METHODOLOGY

A typical office room of size 5 x 5 x 4 m (L x W x H) with two window openings on the adjacent walls are modelled. The size of the window opening is 1 x 1 m, thickness of the roof and side walls are 0.2m and 0.25m respectively. This isolated room model was placed inside a three dimensional box of size 30 x 30 x 20m.

This 3-D box was considered as an external atmospheric zone around the room model. The fluid domain was created in the GAMBIT software and is shown in Figure-3(c). The effect of wind fluctuation on the thermal comfort was studied by varying the wind angle form -67.5° to 67.5° with an increment of 22.5° is also shown in the same Figure-3(b).

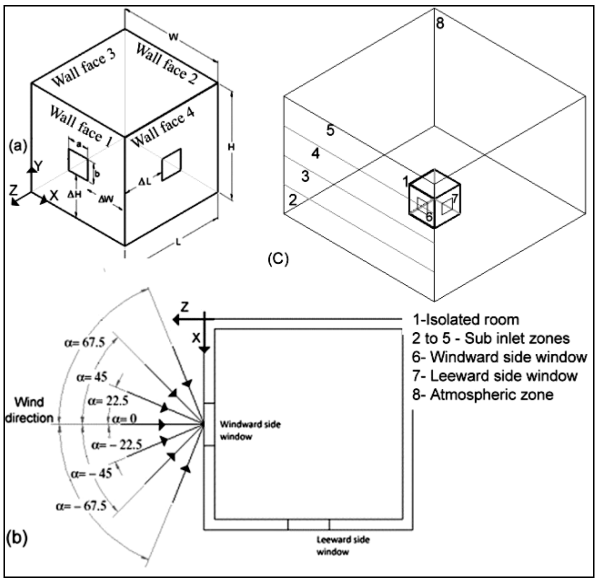

Figure-3. Test case room with CFD model.

3.1. Boundary conditions

The velocity of wind at the inlet of fluid domain is the function of height from the ground level. This variation of velocity along the building height can be defined either by logarithmic profile [14] or by dividing the velocity inlet into many sub inlet zones [15]. In this paper, the wind inlet zone was divided in to 4 sub zones. The wind velocity at these sub zones are predicted from the equation 1 [14].

$$V = V_r c H^a \tag{1}$$

where V is the wind speed at ground level. (m/s), Vr is the reference wind speed measured experimentally, H is the height of the building c is the parameter relating wind speed to terrain nature (0.68 in the open country terrain), and a is the exponent relating wind speed to the height above the ground (0.17 in the open county terrain). The values of turbulent kinetic energy, K and turbulent kinetic energy dissipation rate, ε at the inlet region are calculated from the equations 2 and 3 respectively [16].

$$K = \frac{3}{2}\left(V_{avg} \times T_i\right)^2 \tag{2}$$

$$\varepsilon = C_{\mu}^{\frac{3}{4}} \frac{k^{3/2}}{l_t} \tag{3}$$

Where V_{avg} is the average flow velocity, T_i the turbulence intensity and l_t is the turbulence scale length. T_i and l_t are taken as 4% [17, 18, 19] and 0.4m [20], respectively. The thermal heat gains are measured on the same hot day at noon time. The temperature of the ambient air, sidewalls, roof and floor are measured as 306K, 312K, 325K and 303K respectively. The heat generated from the various electrical devices and human beings are approximated to 25 w/m^2.

Following assumptions are considered to analyse the further work:

a) Ventilation due to wind force was only considered.
b) An isolated room was analyzed.
c) Flow equations are solved by continuity, momentum and energy equations.
d) Air properties are assumed as a constant with reference to atmospheric temperature.

The tetrahedral hybrid T grid type element is used for meshing the flow domain. Approximately 42 cells in the x direction, 33 cells in the y direction and 42 cells in z directions are consumed for the given mesh of size 0.5m. This grid size was independent to the results as verified by repeating the test with a greater number of cells. The fluid model shown in Figure-3(c) was solved in FLUENT 6.1 software by segregated solver. Second order upwind method was employed and the iterations are continued up to the convergence level of 10^{-6}. After solving the fluid domain to the required convergence level, the mass flow rate of air passing through the window opening at the wind ward sidewall are predicted. The mass flow rate predicted from the CFD simulation is compared with the mass flow rate calculated from the network model given in equation 4.

$$Q_{network} = A_{Effective} \sqrt{\left(2\Delta p / \rho\right)} \tag{4}$$

Δp is calculated from the equation 5 [15].

$$\Delta p = 0.5 \rho V^2 \left| (C_{pn} - C_{pi} \right| \tag{5}$$

The Cpi is determined from the law of conservation and is given in the equation 6 [15].

$$\left| (C_{pn} - C_{pi} \right|^{-0.5} + \left| (C_{p(n+1)} - C_{pi} \right|^{-0.5} = 0 \tag{6}$$

The pressure coefficient, C_{pn}, is the function of wind flow direction. The predicted mass flow rate from CFD code is compared with the network model and their deviation with the network model was given in Table-1. The mass flow rate values at the wind ward side for all the wind direction are having the discrepancy in the agreeable level.

Table-1. Discrepancy of CFD with network model.

Wind angle	Mass flow rate (kg/s)		
	Network model	CFD prediction	Discrepancy %
-67.5°	0.331	0.31	6.34
-45°	0.61	0.58	4.92
-22.5°	0.7	0.675	3.57
0°	0.774	0.762	1.55
22.5°	0.7	0.687	1.86
45°	0.61	0.578	5.25
67.5°	0.331	0.298	9.97

4. EFFECT OF WIND DIRECTION ON INDOOR AIR FLOW CHARACTERISTICS

The effect of wind direction on indoor thermal comfort was studied. The direction of wind passing the windward side window was varied from -67.5° to 67.5° as specified in the Figure-3(b). The indoor velocity and temperature trend along the midlines of the test room was predicted and were shown in Figures 4 and 5, respectively.

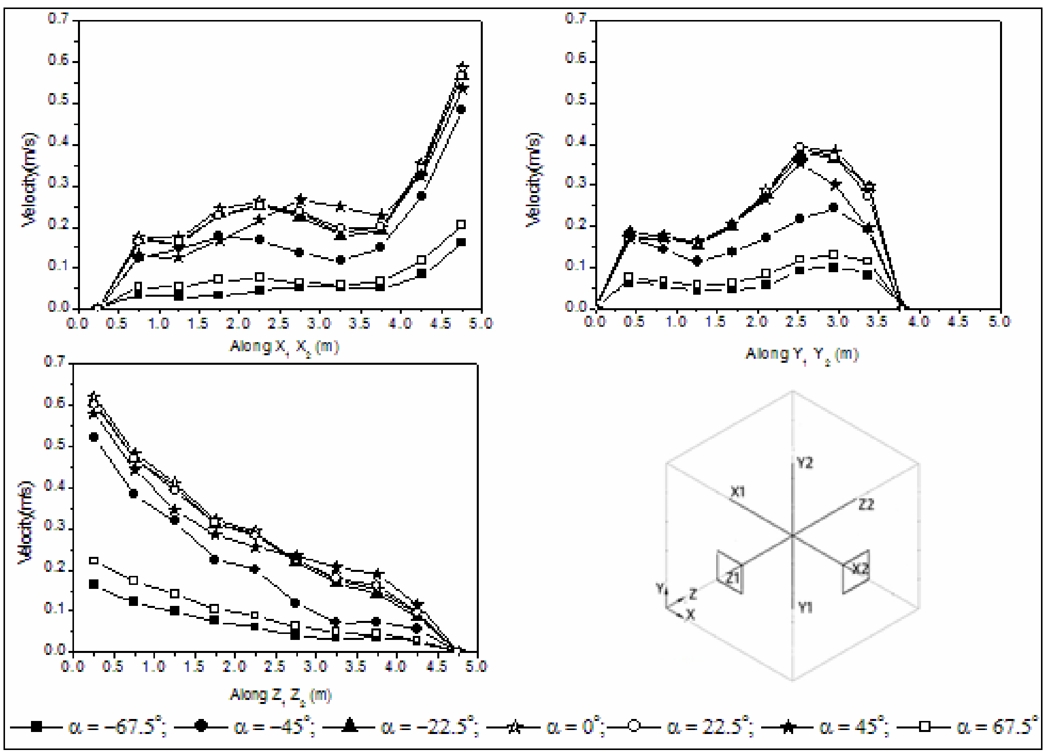

Figure-4. Velocity trend along midlines.

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

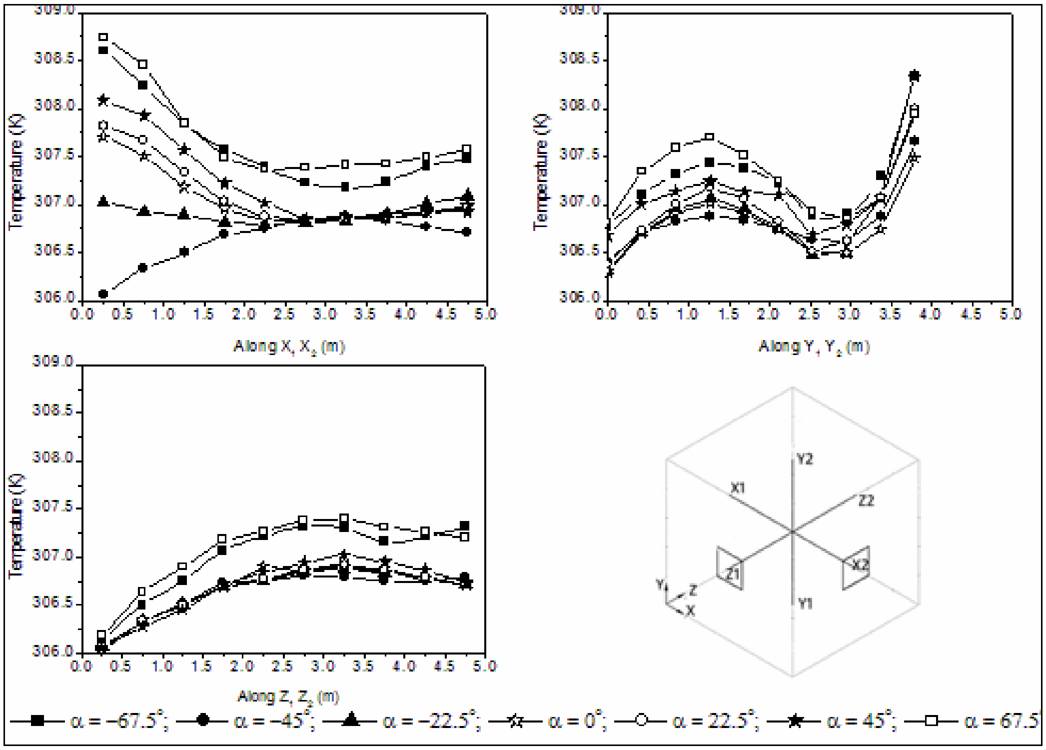

Figure-5. Temperature trend along midlines.

For the given wind velocity of 1m/s, the variation of velocity along the midlines X1X2, Y1Y2 and Z1Z2 are shown in Figure-5. The indoor air velocity is constant up to 3.75m and increasing drastically towards X2 for all wind directions. This drastic increase in air velocity in the X1X2 midline from 3rd m to X2 is due to the presence of both windows at this location. The velocity trend along Y1Y2 shows a slight fall up to 1.25m from the ground surface and above 1.25m to 2.5m the velocity was increasing significantly and again from 2.5th m to roof surface, the velocity gets decreases. The velocity trend along Z1Z2 shows gradual decrease from Z1 to Z2. Among the analysed wind directions, α = -67.5° and 67.5° causes low velocity for indoor air. For this angle the magnitude of air velocity at the wind ward side window is 0.05m/s, which is not having sufficient strength to stale out the hot indoor air. For all the mid lines, the air velocity is maximum for α = 0°, i.e., the wind entering in the normal direction to the windward side window. In comparison, the indoor air velocity is higher for positive wind angles than negative angles. This is because, if α is positive, the entrapped air immediately vents out through the leeward side window without travelling the entire zone of the room. But for wind angles in negative angles, the entrapped air tries to travel along the face 3 and then vents out through the leeward window at the face 4. Due to this phenomenon, the air entering the room at a negative angle is having the possibility to cool the maximum regions of the room which is also evident from the temperature trend. The temperature trend along all the midlines showed in Figure-6 shows elevated temperature for the wind angle α = –67.5° and 67.5°. For α = 45°, the temperature value up to 2m in the X1X2 mid line is high. This reveals that the positive wind angle does not allow the air toward the face 3 and hence the temperature nearer to the face 3 is high. For the negative wind angles like α = –45° and -22.5°, the temperature is almost constant along the X1X2. For all the wind angles, the temperature from 3m to 5m in the X1X2 is about to constant because of the location of windward side window and leeward side window at the walls face 1 and 4, respectively. This causes the air to flow predominantly near to the face 4. The temperature along the Y1Y2 increases up to 1.2m from the ground surface and later it decreases up to 3rd m and again increases towards the roof surface. This decrease in temperature is due to the presence of window opening at the same level. The temperature trend along trend Y1Y2 also shows the lower temperature value for the negative wind angles between α = -45° to 0°. The temperature trend for both α= -65° and 65° runs at the higher level. About 2.5°C temperature increase was identified inside the room with reference to the ambient air for α = -67.5° and 67.5°, while the other wind angle cases increases the indoor air temperature by 1.5°C. The temperature trend Z1Z2 is also follows the similar trend for all the wind directions. Here, the temperature value increases linearly and significantly up to 3m from the windward side and beyond that, a marginal decrease in temperature was identified up to the wall face2. About 1.2°C temperature rise was identified for α = -67.5° and 67.5° and only 0.7°C was identified for the wind angle in between -45° and 45°. The velocity and temperature plot for the plane XZ at a height of 2m from the ground surface is shown in Figure-6.

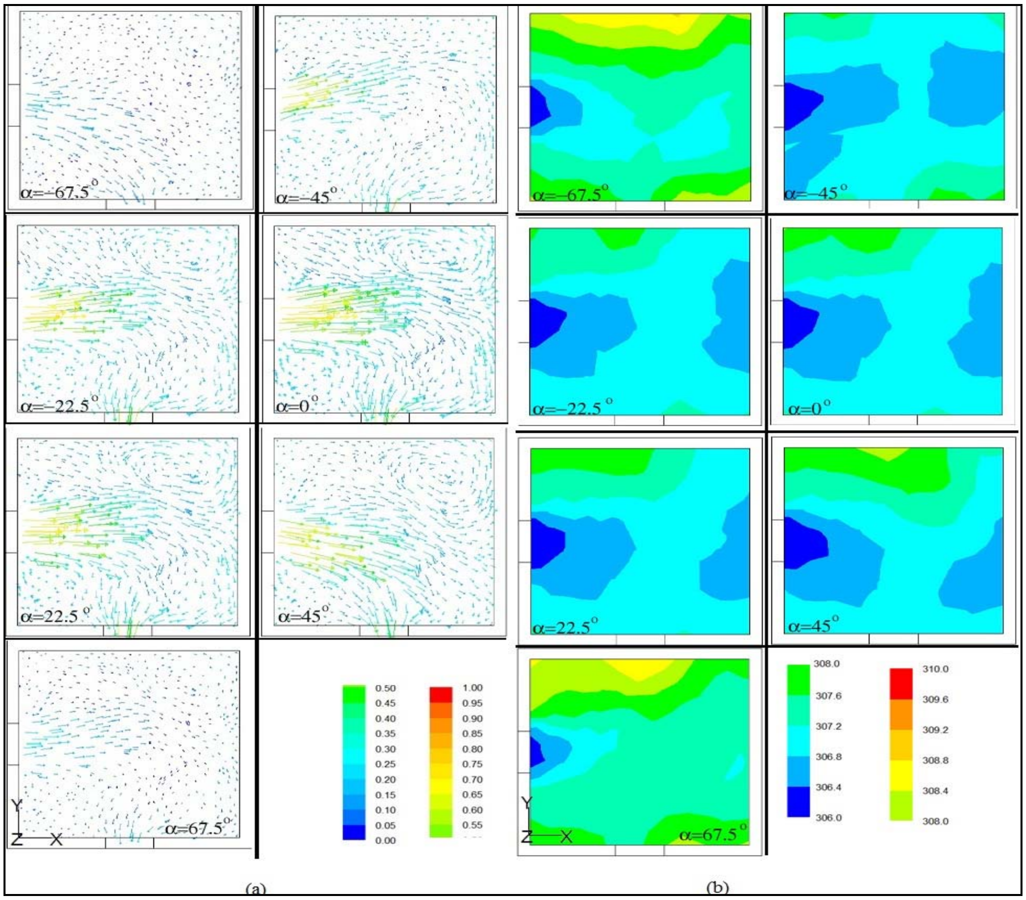

Figure-6. (a) Velocity vector plot (b) Temperature plot at XZ plane at a height of 2m from ground.

In the velocity vector plot for wind angle $\alpha = -67.5°$ and 67.5°, the air particles flow inside the room with the magnitude less than 0.05m/s. This air velocity is not enough to travel the entire portions of the room and their by causes local heating by increasing the temperature to a value of 308.8 K from 306.4 K. Most of the portions nearer to the wall face 3 have high temperature. Also all the room corners are having peak temperature. This shows that, the quantity of air entrapped is not enough to absorb the heat from the indoor surface and it was not properly vented out. So the wind angle in the range $67.5° < \alpha < 90$ and $-67.5° > \alpha > -90°$ may be considered as dead zone direction for ventilation. For wind angle between -45° to -22.5°, the entrapped air moved toward the wall face 3 and cools the portion nearer to the wall face 3 which is evident from the temperature plot. The portion nearer to wall face 3 is having the temperature in the range of 306.8 to 307.6 K and major portions of room interior are having temperature below 307.2K. But a small recirculation zone was identified on the right end of the face 3. After the air gets recirculated, travel towards the right end of the face 4 and their by keeps the room at lower temperature in comparison with other wind angles. For the wind angle of 0°, the maximum amount of air tries to impact the wall face 2 which is opposite to the wind ward side wall. This again causes a small recirculation zone nearer to the right end of wall face 3. Due to this impact, the fluid particle looses their kinetic energy and creates only a small recirculation zone and later vents out through the leeward side window. Hence the portion nearer to the wall face3 is experiencing a slight high temperature as 308. K and the left end of face w3 have not cooled well. For wind angles greater than 0° like 22.5° and 45°, the entrapped air immediately tries to vent out through leeward side window. In these cases also, a small recirculation zone was identified at the right end of the wall face3. But the air particles in the recirculation zone cannot able to vent out completely. However, some amount of the air particles continuously involved in recirculation and causes local heating in the same zone. Hence the portions nearer to the wall face 3 were experiencing a high temperature in the range of 307.2 to 308 K. With all these information it was identified that if the entrapped wind is in the direction between -22.5° to 0°, helps the air to travel near the face 3 and keeps maximum portions of room interior with lower temperature. On the other hand, if the wind was entering in the direction between 0° to 45°, the entrapped air leaves the room immediately through the leeward wind and creates a permanent recirculation zone near the right end of the wall opposite to the leeward window. In this contest, a new design in the window pattern was required which makes the air to travel near the face 3 under various wind direction especially at α –22.5° τo 0°.

Figure-7 shows the percentage of low temperature zone ($\%_{lt}$) predicted for the analyzed wind flow directions. Low temperature zone is the region having temperature between 306 K and 307 K. For wind direction α = -67.5° and 67.5° the $\%_{lt}$ was very low as 5%. However, $\%_{lt}$ increases suddenly to a value of 63.9% for α = -45° and gradually reduces to 51% for α = 45°. From this it is very clear that the percentage of low temperature zone was maximum when the wind flows in the direction of negative angle.

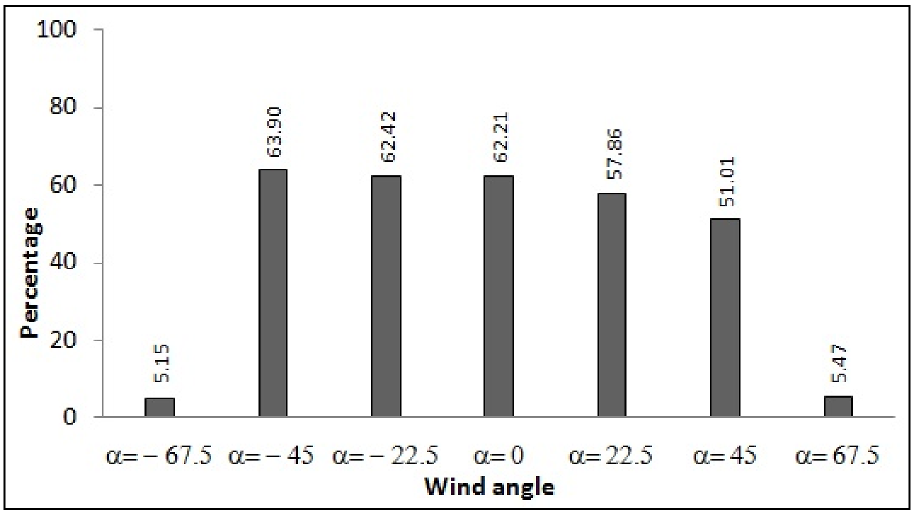

Figure-7. Percentage of low temperature zone predicted for various wind flow directions.

5. CONCLUSIONS

In this study the effect of wind direction on thermal comfort inside the room with window opening at their adjacent walls was studied. CFD technique was used to simulate the internal air flow pattern. Initially, the CFD simulated result was checked for grid independency and standard k- ε model is selected as a suitable turbulence model. Also the CFD simulated pressure coefficients along the external surface of the room were compared with the experimental results for its validation. The effect of wind direction was studied by varying the wind direction from -67.5° to 67.5° with respect to the windward side. For all the wind directions, the CFD simulated mass flow rate of air passing through the windward side window opening was predicted and compared with the mass flow rate calculated from the network model. This deviation was found as 9.97% which is in the agreeable level. Wind direction made greater effect on the indoor thermal comfort and air flow characteristics. Wind direction between -45° to 45° took active participation in the ventilation process. From this study, the most un comfort zone was identified as the portions nearer to the walls which are opposite to the wind ward and lee ward side.

REFERENCES

[1] M.J. Mendell and W.J. Fisk. 1996. Elevated symptom prevalence associated with ventilation type in office buildings. Epidemiology. 7: 583-589.

[2] W.J. Fisk and A.H. Rosenfeld. 1997. Estimates of improved productivity and health from better indoor environments. Indoor Air. 7: 158-172.

[3] Paramasivam Ravikumar and D. Prakash. 2009. Analysis of thermal comfort in an office room by varying the dimensions of the windows on adjacent walls using CFD: A case study based on numerical simulation. Building simulation: 2. 187-196.

[4] Lingying Zhao, Riskowzki G L, Yuanhui Zhang, Christianson. L and Xinlei Wang. 1999. Devolopment of PIV techniques to measure air flow pattern in ventilated airspace's. ASHRAE Transaction. pp. 1088-1107.

[5] E. Dascalaki, M. Santamouris, A. Argiriou, C. Helmis, D.N. Asimakopoulos, K. Papadopoulos and A. Silemes. 1995. Predicting single sided natural

ventilation rates in buildings. Solar Energy. 5: 327-341.

[6] R. R. Walker and M. K. White. 1992. Single sided natural ventilation-how deep an office. Building Services Engineering Research and Technology. 13: 231-236.

[7] A. Nielsen and E. Olsen. 1993. Measurements of if air change and energy loss with large open outer doors. Building physics'93. Lyngby, Denmark, Bjarne Saxhof.

[8] O. Zeidler and K. Fitzner. 1997. Investigation of the impact of natural ventilation through windows on the thermal comfort. Proc. Clima 2000, Brussels. 126.

[9] M. L. Hoang, P. Verboven, J. De Baerdemaeker and B. M. Nocolai. 2000. Analysis of the air flow in a cold store by means of computational fluid dynamics. International Journal of refrigeration. 23: 127-140.

[10] Y. Huo, F. Haghighat, J.S. Zhang and C.Y. Shaw. 2000. A systematic approach to describe the air terminal device in CFD simulation for room air distribution analysis. Building and Environment. 35: 563-576.

[11] Oliver Rouaud and Michel Havet. 2002. Computation of the airflow in a pilot scale clean room using k-ε turbulence models. International Journal of refrigeration. 25: 351-361.

[12] Doosam song and Shinsuke Kato. 2003. Radiational panel cooling system with continuous natural cross ventilation for hot and humid regions. Energy and Buildings. 36: 1273-1280.

[13] T. Yang, N.G. Wright, D.W. Etheridge and A.D. Quinn. 2006. A comparison of CFD and full-scale measurement for analysis of natural ventilation. International Journal of Ventilation. 4(4): 337-348.

[14] G. Evalo and V. Popov. 2006. Computational analysis of wind driven natural ventilation in buildings. Energyand Building. 38: 491-501.

[15] O.S. Asfour and M.B. Gadi. 2007. A comparison between CFD and network models for predicting wind-driven ventilation in buildings. Building and Environment. 42: 4079-4085.

[16] H.G. Versteeg and W. Malalasekera. 1995. An introduction to Computational Fluid dynamics: The Finite volume method approach, Wiley, New York, USA.

[17] P. D. Gaspar, R.F. Barroca and R.A. Pitarma. 2003. Performance evaluation of CFD code in building energy and environmental analysis. Proceedings of Eight International IBPSA conference, Eindhoven, The Netherlands. pp. 371-378.

[18] J. D. Posner, C. R. Buchanan and D. Dunn-Rankin. 2003. Measurement and prediction of indoor air flow in a model room. Energy and Buildings. 35: 515-526.

[19] D. Lakehal and W. Rodi. 1997. Calculation of the flow past a surface mounted cube with two-layer turbulence models. Journal of Wind engineering and Industrial aerodynamics. pp. 65-78.

[20] AIRPAK 2.1 User guide, Fluent Inc. 2001.