Journal of Engg. Research, ICIPPSD Special Issue

THERMODYNAMIC ANALYSIS OF WIND CATCHER WITH COOLING PADS USING SSG REYNOLDS STRESS TURBULENCE MODEL.

DOI : 10.36909/jer.ICIPPSD.15507

Agarwal A.*, Pitso I.*, Letsatsi M.T.**

*Department of Mechanical Engineering, FET, University of Botswana, Gaborone, Botswana.

**Department of Industrial Design & technology, FET, University of Botswana, Gaborone, Botswana.

*Corresponding Author: agarwala@ub.ac.bw

ABSTRACT

With increase in energy requirement, the researchers are looking for energy efficient passive ventilation techniques. The current design concept is based on environment sustainability and use of renewable energy sources is preferred over conventional energy sources. The current research investigates the wind catcher design with cooling pads using techniques of Computational Fluid Dynamics. The CAD model of wind catcher is designed using Creo design software and CFD analysis is conducted using ANSYS CFX software. The CFD analysis is directed at different air inlet velocities and SSG Reynolds stress turbulence model under steady state thermal conditions for both side open design and singe side open designs. The cooling pads have successfully reduced temperature up to 1.65 degrees for single side opening and 2.86 degrees for double side opening design. Maximum air flow rate is achieved with higher air inlet velocities for both design types.

Key words: Wind catcher; computational fluid dynamics; cooling pad; temperature; ventilation.

INTRODUCTION

With more emphasis on construction of energy efficient buildings, the demand for utilization of natural ventilation techniques has increased manifold. The natural ventilation technique employs buoyancy and wind forces to generate air flow inside buildings. Adequate ventilation facilitates proper breathing air, ventilation of contaminants and thermal conditioning (Shwehdi et al., 2015). The effective air ventilation can be achieved with proper site environment and building architecture. Besides these factors the sunlight, geographical location, prevailing wind speed also affects the efficiency of natural ventilation. The natural ventilation methods use stack effect for driving air and is ecofriendly. Natural ventilation in buildings offers following advantages.

- The running cost is much lower than mechanical air circulation systems.

- The maintenance requirement of natural ventilation system is much lower than that of mechanical ventilation systems and has lower installation cost also. Natural ventilation incurs nominal installation expenses and can operate without AC plant as shown in figure 1.

Figure 1 Natural Ventilation

Li and Mak (2007) has conducted experimental and numerical analysis of wind catcher under different air velocities. The experimental and numerical results were in close agreements and findings have shown that wind catcher efficiency was significantly affected by air velocity and direction (i.e., the incident angle). Hughes and Ghani (2008) conducted analysis on passive ventilation device known as wind vent using experimental and numerical techniques. The wind vent had been very effective even at low external air speed and its viability is well proven for urbanized regions. Montazeri and Azizian (2009) investigated the effect of number of openings in wind catchers on its performance. The investigation was carried out using smoke visualization, CFD and experimental techniques for 10m/s, 20m/s and 25m/s. The results have shown decrease in efficiency with increased number of openings. The rectangular shaped wind catcher has higher efficiency as compared to other shapes (Montazeri, 2011). Lakhdari, Krim, and Borni (2019) conducted the research on wind generation system using PMSG. Simulink simulation-based study shows effectiveness of the system. Asfour and Gadi (2006) has investigated the effect of rood type on performance of natural ventilation of room using Computational Fluid Dynamics technique. The findings have shown that best roof shape for improving internal air distribution is of arched type. Alshayji and Ebrahim (2020) conducted a numerical analysis on inclined rook using COMSOL software package to enhance the ventilation through convection heat transfer. The results were tested with an experimental work & shows maximum 89% reduction in heat gain. Asfour and Gadi (2007) have conducted investigation of wind catchers using techniques of Computational Fluid Dynamics and various network models. The author emphasized the importance of CFD technique in determining fluid flow characteristics of wind catcher at normal and oblique wind conditions. The findings have shown that CFD results of natural ventilation are in close agreements with analytical results. Zile (2019) performed the study based on solar and wind-based hybrid plant. The concept and thus results, provide an impressive support in selecting the location thorough the world. Elmualim (2006) conducted CFD simulation on wind catchers with square plan and circular plan. The sharp edges of square resulted in increased flow separation which helped to achieve higher ventilation and is therefore more efficient. Elmualim and Awbi (2002) has conducted CFD and experimental testing of wind catchers having square type section and circular type section. The testing was conducted to determine air flow characteristics and findings have shown that the performance was largely dependent on wind flow direction and magnitude. Bahadori, Mazidi, and Dehghani (2008) conducted experimental testing of wind catcher (scaled model) to determine wind pressure coefficients at different openings. Different wind angles are considered for open country and suburban location. The wind pressure coefficients are used to determine air flow rate. The findings have shown that air flow rates can be significantly affected by restricting air leaving through apertures. Bouchahm, Bourbia, and Belhamri (2011) conducted experimental testing to determine the effect of external climatic conditions on air flow characteristics of wind catcher. The effect on internal air humidity and temperature is thoroughly studied and comparison is made from respective values in outdoor conditions. Zhang et al. (2019) investigated the thermal plume based VRF System for an office building using CFD simulation. Results shows that inlet temperature can be compacted by 22% which provides a guidance to the optimization design layout. The main objective of this study is to investigate the application of cooling pad in wind catchers using CFD. The software used for CAD modelling is done using Creo design software and CFD simulation is conducted using ANSYS CFX (Agarwal and Mthembu, 2020). CFD analysis using ANSYS software on double sided opening wind catcher at different velocities to study temperature plot, velocity plot, pressure plot.

METHODOLOGY

The first step in methodology involves CAD modeling which is developed in Creo design package with dimensions as shown in figure 2.

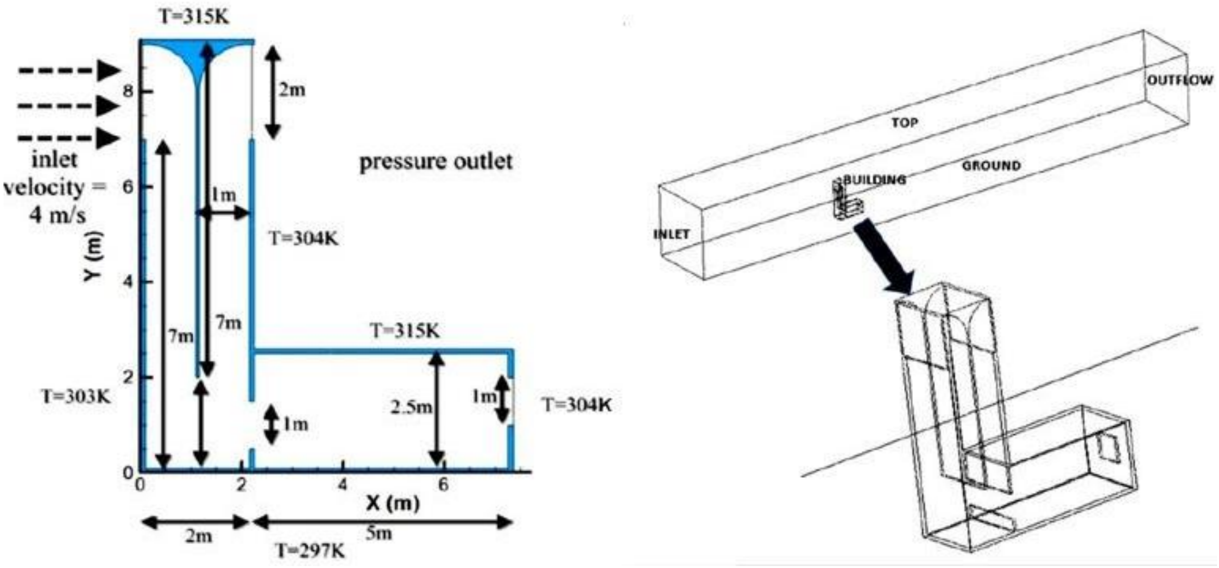
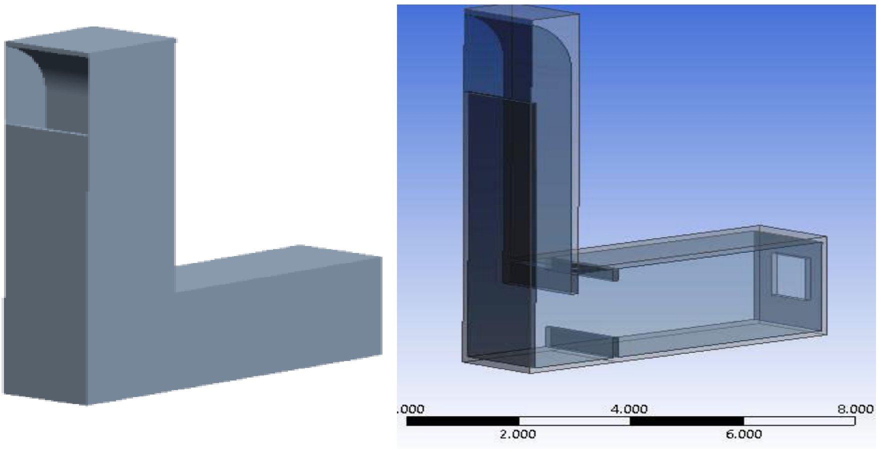


Figure 2 Interpretation of wind catcher (Ahmed Kabir, Kanagalingam, and Safiyullah, 2017)

The schematic of windcatcher is shown in figure 2 shows length, width and height of wind catcher placed inside wind tunnel. The CAD model is developed as per schematic is shown in fig 3 (a). The model using Creo design software is imported in ANSYS as shown in fig. 3 (b)



(a) **(b)**

Figure 3 wind catcher CAD model (a) and Imported CAD model (b)

The model is further tested for symmetrical errors and clean-up operations are executed for reducing hard edges, surfaces, and point detection (Agarwal and Mthembu, 2020). The CAD model is meshed with tetrahedral elements (Agarwal, Molwane, and Pitso, 2020). The element sizing is set to fine, size function set to curvature and relevance to 0. Inflation also set to zero and transition to smooth. The total number of elements generated is 61513 and total quantity of nodes generated is 14845 shown in figure 4.

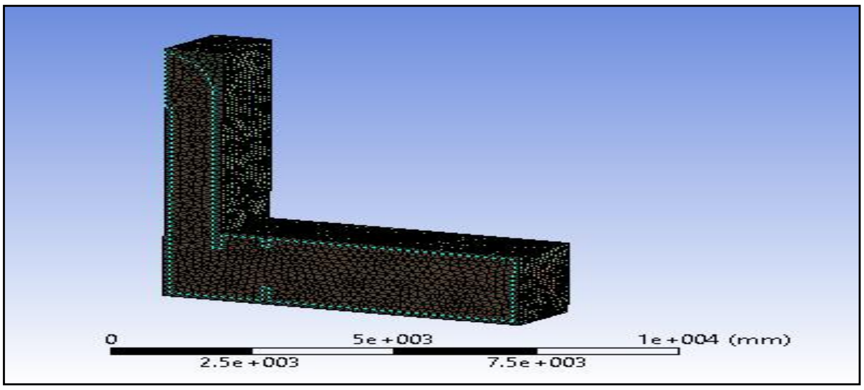


Figure 4 Meshed CAD model of wind catcher

In this stage the domains are defined which is fluid and solid both. The inner domain will be fluid through which air is flowing and outer will be solid which represents wall. For fluid domain the reference pressure is set to 1atm and material is taken as air ideal gas. The turbulence model for analysis is selected SSG Reynolds stress turbulence model as it certainly includes the effects of streamline curvature, sudden changes in the strain rate, secondary flows or buoyancy compared to turbulence models using the eddy-viscosity approximation (Pawar, Mujumdar and Thorat, 2012). The Reynolds averaged momentum equations for the mean velocity are (Lee, Lee, and Kim, 2017)-

$$\frac{\partial \rho U_i}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho U_i U_j\right)-\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\right]=-\frac{\partial p''}{\partial x_i}-\frac{\partial}{\partial x_j}\left(\rho \overline{u_i u_j}\right)+S_{M_i} \tag{1}$$

Here revised pressure is the sum of body forces and the fluctuating Reynolds stress contribution. Loads and boundary conditions are shown in figure 5. Unlike eddy viscosity models, the modified pressure has no turbulence contribution and is related to the static (thermodynamic) pressure by:

$$p''=p+\frac{2}{3}\mu\frac{\partial U_k}{\partial x_k} \tag{2}$$

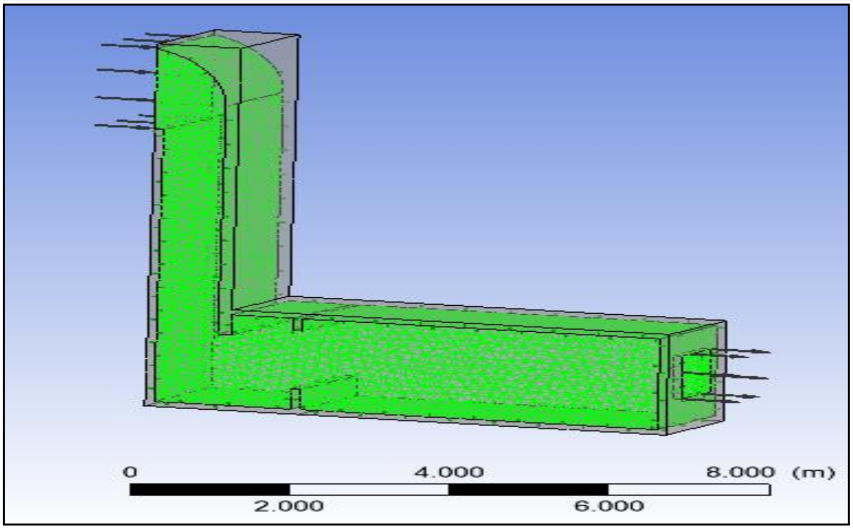
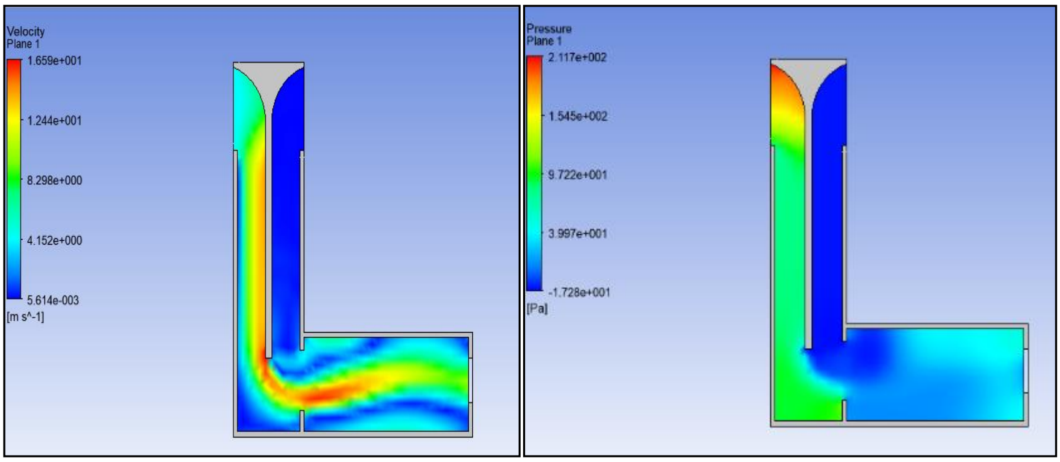


Figure 5 Loads and boundary conditions of wind catcher

RESULT AND DISCUSSION

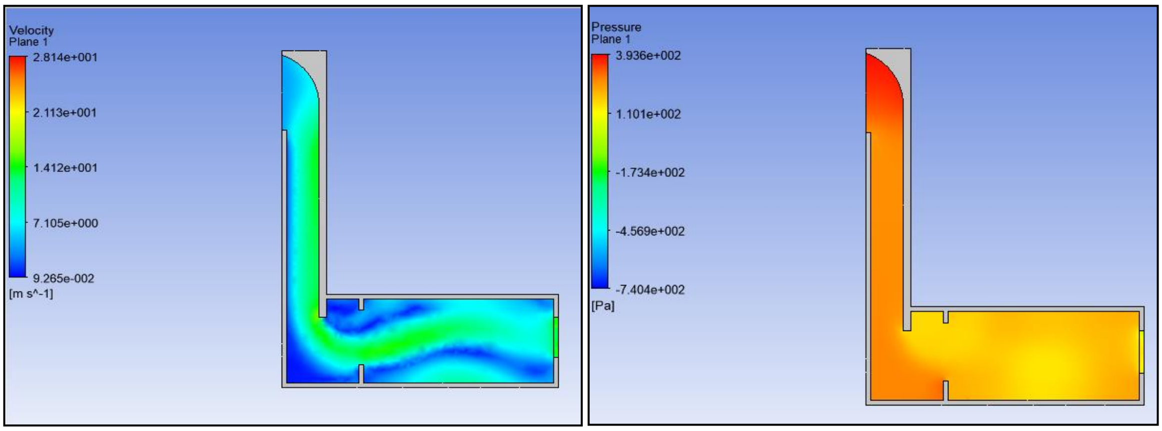
Preliminary CFD evaluation is conducted keeping both sides open to study the effect of air flow characteristics. The analysis is conducted under isothermal conditions. Further analysis is conducted using wet clay conduit to reduce temperature. The velocity contour plotted as shown in fig.6 (a) shows higher velocities near wall partition with magnitude of 16.5 m/sec which reduces and again increases. The design shows considerable good flow of air in room. The pressure contour plotted as shown in fig.6 (b) shows higher pressure near wall inlet and reduces on moving towards exit of the room.



(a) **(b)**

Figure 6 With both sides open velocity contour (a) and pressure contour (b).

The pressure in vertical stack is higher as compared to rest of room. CFD analysis is conducted with single opening through window without cooling pads. The results of analysis are shown in fig. 7(a) and fig.7 (b).

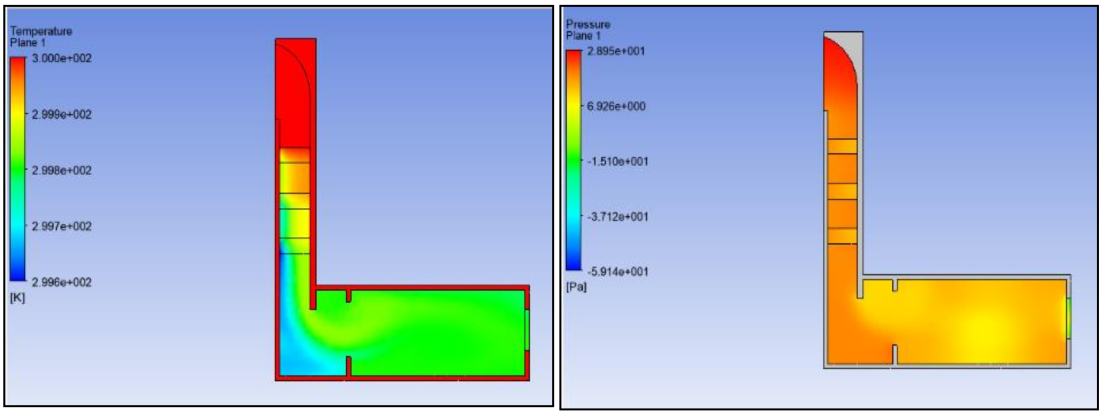


(a) **(b)**

Figure 7 With both sides open velocity contour (a) and pressure contour (b).

The velocity contour in fig.7 (a) shows higher velocities near wall partition with magnitude of 14 m/sec which reduces and again increases. The pressure contour plotted in fig.7 (b) shows almost constant pressure in room and high pressure near air inlet. The pressure is 110 Pa across room. Further CFD analysis is conducted using cooling pads (3 no's) with 1.1 m/sec inlet velocity as shown in fig. 8 (a).

Considerable reduction in temperature is observed with the use of cooling pads. The temperature near inlet is 300K as defined and temperature air after passing through cooling pads reduces to nearly 299.6K with more uniform temperature distribution. The pressure plot is shown in fig. 8 (b). The pressure plot shows higher pressure near inlet and constant pressure in room with air opening from side of tower.



(a) **(b)**

Figure 8 With single sides open temperature contour (a) and pressure contour (b).

The velocity plot is shown in figure 9 shows higher magnitude of air velocity on inlet and outlet of tower and reduced velocity in room with magnitude of average velocity of 1.97 m/sec.

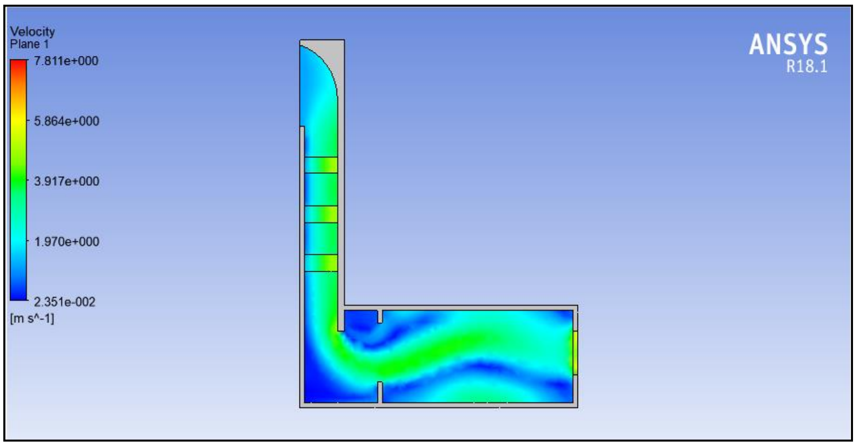


Figure 9 Velocity contour with single sides open

The ventilation performance of wind-catchers is usually projected by establishing the pressure coefficients.

Determination of Air Flow rate

The pressure coefficient describes the ratio of the local surface pressure and the dynamic pressure and defined as follows:

$$C_p = dP / (\rho\ v^2 / 2) \qquad (3)$$

Where- C_p = pressure coefficient, dp = pressure difference (N); ρ = fluid density (kg/m^3)
v = flow velocity (m/s); dh = head (m).
Due to comparation concerning the passing flow through an orifice in a pipe and the airflow through the opening of a wind-catcher, the orifice equation is widely used to calculate the airflow rate (m^3/s) in designing of natural aeration systems.

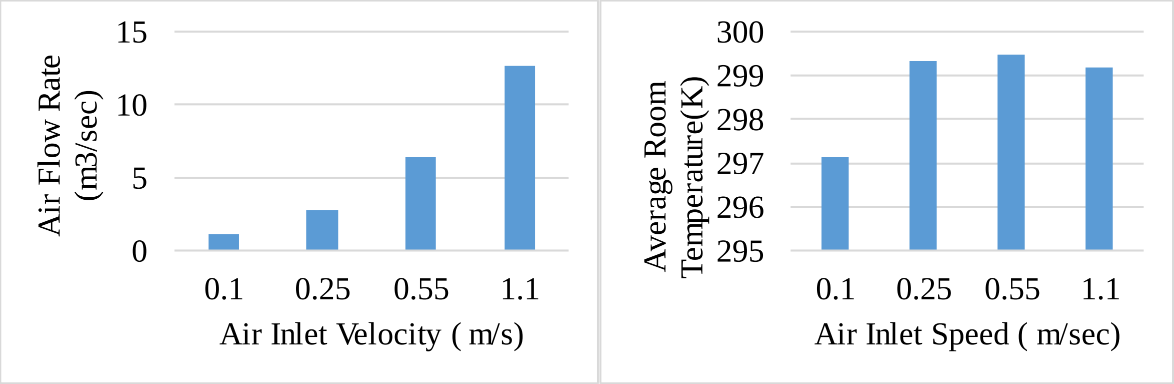
$$Q = A\ V_{ref}\ C_d\ (Cp)^{½} \qquad (4)$$

Where, A is the area of opening (m^2) = $3.6m^2$, V_{ref} is the free stream velocity (m/s), ΔC_p is the pressure coefficient difference across the inlet and outlet openings and C_d the discharge coefficient that describes the flow resistance of the opening. The discharge coefficients given in the literature are in the range of 0.6–0.65 for sharp-edged openings as shown in table 1.

Table 1 Ventilation rate for both side opening

Inlet Velocity (m/s)	**V_{avg}(m/s)**	**Avg Temp (K)**	**Pressure Diff (Pa)**	**Pressure Coefficient C_p**	**Q (Air Flow Rate) m^3/s**
.1	.11	297.15	.15	24.48	1.17
.25	.31	299.33	1.01	17.41	2.79
.55	.69	299.48	5.27	18.17	6.35
1.1	1.49	299.18	20.95	15.51	12.67

Air Flow rate with both sides open design is shown in figure 10. From fig. 10 (a) the average room velocity increases with increase in air inlet velocity. The average velocity and ventilation air flow rate is highest for 1.1 m/s air inlet speed as is evident from fig. 10 (b).



(a) **(b)**

Figure 10 With both sides open air flow rate (a) and average room temperature (b).

The average room temperature is lowest for air inlet velocity of .1m/s with magnitude of 297.15K as shown in figure 11.

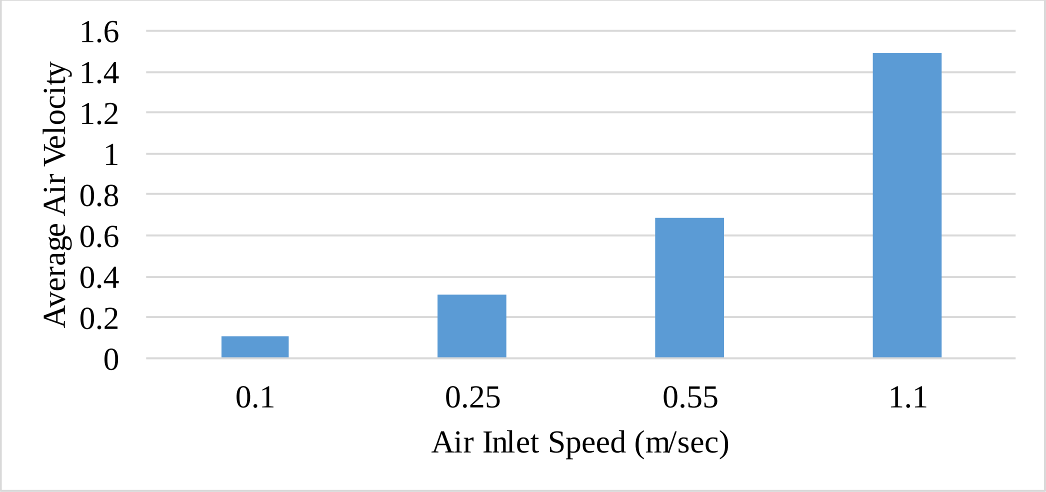


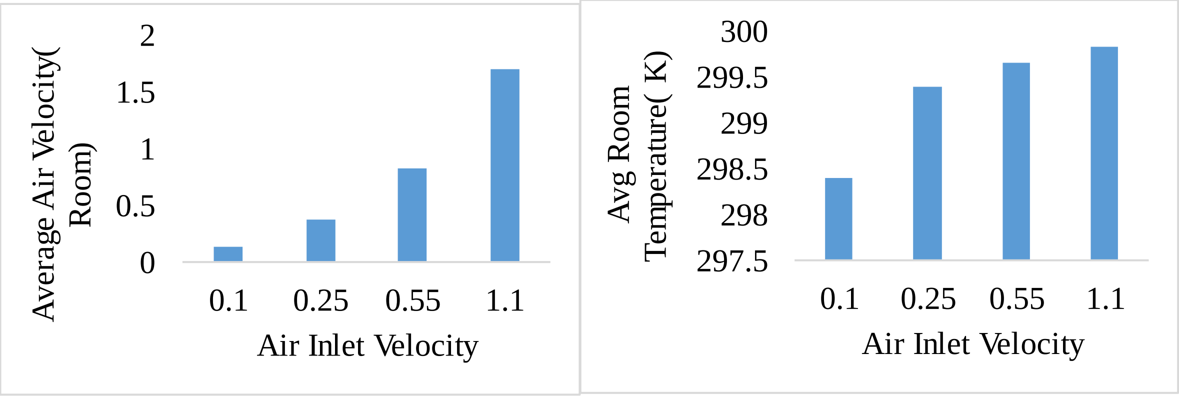
Figure 11 Average air velocity with both sides open design.

Ventilation Rate for single side opening is shown in table 2.

Table 2 Ventilation rate for single side opening

Inlet Velocity(m/s)	**V_{avg}(m/s)**	**Avg Temp(K)**	**Pressure Diff. (Pa)**	**Pressure Coefficient C_p**	**Q**
.1	.14	298.39	.22	18.33	1.29
.25	.37	299.39	1.42	17.10	3.30
.55	.83	299.66	6.67	15.88	7.14
1.1	1.70	299.82	24.87	14.05	13.76

Figure 12 shows the values with single side opening (window). From fig. 12 (a) the average room velocity increases with increase in air inlet velocity. The average room temperature is lowest for air inlet velocity of .1m/s with magnitude of 298.39K as can be seen from fig.12 (b).



(a) **(b)**

Figure 12 for single side opening room average velocity (a) and average temperature (b)

The average velocity and ventilation air flow rate is highest for 1.1 m/s air inlet speed as is evident from figure 13.

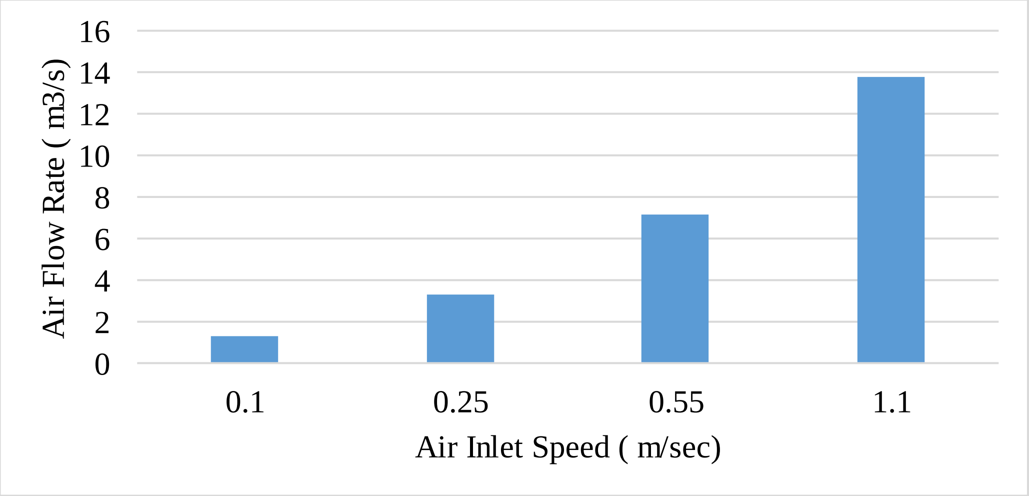


Figure 13 Air flow rate for single side open

CONCLUSION

CFD analysis is conducted on 2 different designs (2 sides opening and single side opening) of wind catcher using ANSYS CFX software. The investigation is performed at different air inlet velocities and different fluid flow variables are determined. The detailed conclusion are as follows: The ventilation rate of single side opening (window opening) is found to be better than both side opening design for all inlet velocities. The application of cooling pad has reduced temperature of room up to 2.86 degrees for both side opening design. The application of cooling pad has reduced temperature of room up to 1.65 degrees for single side opening design. In terms of room cooling both side opening design is better as compared to single side opening design. Ventilation rate increases with increase in air intake velocity for both designs. The highest cooling of room is obtained at lowest air inlet velocity and for higher velocities temperature difference is less than 1 degree Celsius. Ventilation rate increases with increase in air inlet velocities for both designs of both side opening and single side opening. Average room temperature is highest at .55m/s for both side opening and minimum at .1 m/sec air inlet velocity. Average room temperature is highest at 1.1m/s for single side opening and minimum at .1 m/sec air inlet velocity.

REFERENCES

Agarwal, A., O. B. Molwane, and I. Pitso. 2021. Analytical Investigation of the Influence of Natural Gas Leakage & Safety Zone in a Pipeline Flow. *Materials Today: Proceedings 39(1):547-552*.

Agarwal, A., and L. Mthembu. 2020. CFD Analysis of Conical Diffuser under Swirl Flow Inlet Conditions Using Turbulence Models. *Materials Today: Proceedings* 27(2):1350–55.

Ahmed Kabir, Ijaz Fazil Syed, Sivamoorthy Kanagalingam, and Ferozkhan Safiyullah. 2017. Performance Evaluation of Air Flow and Thermal Comfort in the Room with Wind-Catcher Using Different CFD Techniques under Neutral Atmospheric Boundary Layer. *Energy Procedia* 143:199–203.

Alshayji, Adel, and Shikha Ebrahim. 2020. Numerical Simulation of Heat Transfer Process in Inclined Roofs with Radiant Barrier System. *Journal of Engineering Research (Kuwait)* 8(2):305–23.

Asfour, Omar S., and Mohamed B. Gadi. 2006. Effect of Integrating Wind Catchers with Curved Roofs on Natural Ventilation Performance in Buildings. *Architectural Engineering and Design Management* 2(4):289–304.

Asfour, Omar S., and Mohamed B. Gadi. 2007. A Comparison between CFD and Network Models for Predicting Wind-Driven Ventilation in Buildings. *Building and Environment* 42(12):4079–85.

Bahadori, M. N., M. Mazidi, and A. R. Dehghani. 2008. Experimental Investigation of New Designs of Wind Towers. *Renewable Energy* 33(10):2273–81.

Bouchahm, Yasmina, Fatiha Bourbia, and Azeddine Belhamri. 2011. Performance Analysis and Improvement of the Use of Wind Tower in Hot Dry Climate. *Renewable Energy* 36(3):898–906.

Elmualim, Abbas Ali. 2006. Dynamic Modelling of a Wind Catcher/Tower Turret for Natural Ventilation. *Building Services Engineering Research and Technology* 27(3):165–82.

Elmualim, Abbas Ali, and Hazim B. Awbi. 2002. Wind Tunnel and CFD Investigation of the Performance of 'Windcatcher' Ventilation Systems. *International Journal of Ventilation* 1(1):53–64.

Hughes, Ben Richard, and S. A. A. Abdul Ghani. 2008. Investigation of a Windvent Passive Ventilation Device against Current Fresh Air Supply Recommendations. *Energy and Buildings* 40(9):1651–59.

Lakhdari, Abdelkader, Fateh Krim, and Abdelhalim Borni. 2019. Model Predictive Control for an SAPF Interfaced by a Wind Energy Conversion System Based on Permanent Magnet Synchronous Generator. *Journal of Engineering Research* 7(1):1–19.

Lee, Seung-Jun, Jae Hwa Lee, and Byoung Jae Kim. 2017. Improvement of the Two-Fluid Momentum Equation Using a Modified Reynolds Stress Model for Horizontal Turbulent Bubbly Flows. *Chemical Engineering Science* 173:208–17.

Li, Liu, and C. M. Mak. 2007. The Assessment of the Performance of a Windcatcher System Using Computational Fluid Dynamics. *Building and Environment* 42(3):1135–41.

Montazeri, H., and R. Azizian. 2009. Experimental Study on Natural Ventilation Performance of a Two-Sided Wind Catcher. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* 223(4):387–400.

Montazeri, Hamid. 2011. Experimental and Numerical Study on Natural Ventilation Performance of Various Multi-Opening Wind Catchers. *Building and Environment* 46(2):370–78.

Pawar, Sanjay B., A. S. Mujumdar, and B. N. Thorat. 2012. CFD Analysis of Flow Pattern in the Agitated Thin Film Evaporator. *Chemical Engineering Research and Design* 90(6):757–65.

Shwehdi, M. H., S. Rajamohamed, A. Ali Smadi, M. Bouzguenda, A. A. Alnaim, and S. Fortea. 2015. Energy Savings Approaches of Buildings in Hot-Arid Region, Saudi Arabia: Case Study. *Journal of Engineering Research* 3(1):127–36.

Zhang, Yin, Xiao Zhang, Shurui Guo, and Enshen Long. 2019. Thermal Plume Simulation of Vrf Air Conditioners for Cooling System in High-Rise Buildings: A Case Study in China. *Journal of Engineering Research (Kuwait)* 7(3):48–61.

Zile, Mehmet. 2019. Design and Implementation of Wind Speed/Solar Radiation Hybrid Energy Station Connected with the Network. *Journal of Engineering Research (Kuwait)* 7(4):203–14.