Journal of Building Engineering 6 (2016) 196–202

Contents lists available at ScienceDirect

Journal of Building Engineering

journal homepage: www.elsevier.com/locate/jobe

Experimental and CFD analysis of turbo ventilator

Ganesh K. Jadhav [a,*], P.M. Ghanegaonkar [b], Sharad Garg [c]

[a] Research Scholar, Sinhgad College of Engineering, Pune, India
[b] Professor in Mechanical Engg. Dept., Dr. D.Y. Patil Inst. of Engg. & Tech., Pimpri, Pune, India
[c] Continuum Technology, Pune, Flat No. C3/1, Dwarka Society, Mahalunge Ingale, Chakan, Pune, India

ARTICLE INFO

Article history:
Received 23 November 2015
Received in revised form
1 April 2016
Accepted 3 April 2016
Available online 4 April 2016

Keywords:
Turbo ventilator
Experimental setup
Testing
CFD

ABSTRACT

Proper ventilation is primarily necessary in buildings and industries to provide healthy atmosphere. The turbo ventilator is simple operating device and is available at affordable price. It removes hot gases, fumes and bad odor by enhancing ventilation. It is observed through the available catalogues of the manufacturers in India, that ldings and there is inadequate data and testing facilities for predicting performance of turbo ventilators. Present study is carried out to analyze the performance of turbo ventilator with respect to wind velocity and inlet throat diameter. This would give proper guideline to the user to select appropriate turbo ventilator and understand the extraction capacity of the ventilator at different wind velocities. In this work, we have made an attempt to make detailed literature survey of the previous research in this area. The effect of throat diameter and wind velocity on the performance of turbo ventilator is studied experimentally and using CFD software. Turbo ventilator models with throat diameter 600, 500 and 300 mm are tested for their performance. It is observed that turbo ventilator with lager throat diameter gives more mass flow rate and less rotational speed compared to small size ventilator. It is also observed that the mass flow rate of the turbo ventilator increases with the wind velocity, irrespective of the type of the ventilator. The simulation study has been conducted using CFD tools to compare the experimental performance. The Experimental results for 1 m/s, 2 m/s, 3.5 m/s, 5 m/s, 7.5 m/s and 9.5 m/s wind velocities are validated with CFD analysis. It is observed that CFD mass flow rate results are 12% to 15% higher than the experimental results. Unique feature of these ventilators is that it remains operational even in absence of wind flow due to stack effect.

1. Introduction

The turbo ventilator is a ventilation device which has simple operating mechanism. It is very light in weight and is available at affordable price. It is driven by natural wind force and is always a preferred choice to ventilate buildings without depending on power driven system such as air conditioning systems, electric exhaust fans. Commercial turbo ventilator was manufactured by Edmonds of Australia in the year 1934 [1]. These devices are commercially available in the market since many years; however still there is scope for further modification of ventilation device to improve its performance. It is defined by American Society of Heating, Refrigerating and Air conditioning Engineers (ASHRAE) (2011) as a heat escape port located high in the building and enclosed properly for weather tightness with the primary motive forces being stack effect and wind induction [2].

Wind driven turbo ventilators are used in many countries, because of its very low initial capital cost. They are more consistent than any other ventilation system. Turbo ventilator is intended to withstand high wind speeds in unusual weather conditions. It is made of stainless steel or aluminum, which is virtually maintenance free for more than ten to fifteen years.

A turbo ventilator having a number of vertical blades (curved or straight blades) in a circular array mounted on a frame as shown in Fig. 1.

A weatherproof dome is on top of the frame. The entire blade assembly is mounted on the central shaft which rotates under influence of impinging wind velocity. The shaft is mounted on the roller bearing. When wind impinges on the surface of the curved blades the resulting lift and drag forces cause the turbine to rotate. Due to this rotation inside air is thrown outward direction due to centrifugal force which produces a negative pressure inside the turbine which extracts stale air. However with this analysis it is seen that as air velocity increases beyond certain limits there appears slight blocking of the outgoing air. In the absence of wind, a

* Corresponding author.
E-mail addresses: ganeshjadhavcoep@gmail.com (G.K. Jadhav), pmghanegaonkar@yahoo.com (P.M. Ghanegaonkar), sharadnhn@gmail.com (S. Garg).

http://dx.doi.org/10.1016/j.jobe.2016.04.001
2352-7102/

Nomenclature

lps	Litre per second
TV	Turbo ventilator
Q	Mass flow rate

Fig. 1. Turbo ventilators under test.

turbo ventilator still remains effective due to stack effects [3] and [4]. Dale et al. [5] studied the effectiveness of the 305 mm turbo ventilator in improving ventilation of a room already fitted with two soffit vents of free vent areas (0.08 m^2). It is observed that there is reduction in temperature of the room by 0.56 °C. West S., [6] studied the effect of blade height of turbo ventilator on the performance. It is observed that mass flow rate increases by 13.5%, if the vanes height of turbo ventilator is increased by 50%. Lai C.M. et al., [7] have conducted flow visualization in and around the turbo ventilator and found that ventilators with larger diameter gives better ventilation rates, tested three different size ventilators of 152, 356 and 508 mm in diameters under wind speeds ranges between 10 and 30 m/s, but ventilation rate exhausted due to 356 and 508 mm ventilators are more or less the same. Shao Ting J. Lien et al., [8] studied the effect of roof angle on the performance of turbo ventilator and effect of forces on the ventilation device. Findings show that with increase in inclination of roof angle there is decrease in the rotational speed of ventilator which results in minimum extraction rate at low wind speed. This is due to, forces acting on the ventilator decrease with increase in inclination angle. Ismail M. et al., [9–11] investigated the performance of hybrid turbine ventilator for Malaysian climate conditions. The turbo ventilator is provided with the opening of 80 mm on the top and solar powered extracted fan at the bottom level. The inner duct of diameter 200 mm is fitted inside the turbo ventilator. This combination creates constant air flow and provides uniform distribution to extract heat effectively. It is observed that indoor air temperature drop down by 0.7 °C and humidity reduced by 1.7%. Al-Obaidi K.M. et al. [12] design and developed attic vent that provides a better solution to reduce solar heat gains trapped in a roof attic. This paper compares the performance of attic ventilation by a single hybrid turbine ventilator for a specific volume under both unvented and vented conditions. The study was done in an actual roof attic (10 m^3) located in the University Sains, Malaysia. The results showed that the reduction of the air temperature was 6.4 °C and humidity to about 40–50%. Lai C.M. et al., [13] has conducted flow visualization of turbo ventilator with the Gas-Tracing Technology and understand the direction of flow. The flow of air entering the turbo ventilator moves in the same direction as that of the rotation of the turbine and naturally becomes the force to push on the rotation of the turbine. The left-hand side flow then flows along the turbo ventilator to the wake region. The direction of the flow on the right is opposite to that of the rotating turbine, and thus becomes the defected field of airflow that blocks the turbine from rotating. The separation of outer flow takes place and air is forced outside which results in the up rise of air in the duct (connected to the ventilator). Karam M. Al-Obaidi et al. [14] review has shown that the attic ventilation strategy in Malaysia using the hybrid turbine ventilator (HTV) is a promising technique to ventilate the buildings. Author has designed and developed system which consists of an inlet vent from the gable and outlet vent as the turbo ventilator with curved blades of 450–500 mm diameter in size. Shieh T.H. et al., [15] in this study, carried out experiments by combining an air driven turbo ventilator and extractor fan operated by solar-power. The prototype of 500 mm diameter hybrid turbo ventilator with inner blades fitted at bottom of the shaft developed by Lai [7] which is replaced by the existing inner blades with 400 mm extractor fan run on solar panel. It is observed that with increase in the mass flow rate, rotational speed of fan increases up to 1500 rpm and wind speed of up to 5 m/s. If wind velocity goes beyond 5 m/s, the extractor fan do not contribute to the additional airflow extraction rate.

Some researchers have worked on the geometrical modifications in the ventilator to improve the mass flow rate. For small wind velocity the straight blades ventilator gives the better mass flow rate as compared to curved blades ventilator. The increase in mass flow rate is observed by increase in blade height of the ventilator. Mounting of extractor fan at the bottom of ventilator which was operated by external power and/or natural resulted in no change in the mass flow rate. It was found that very little work has been done on operating parameters like diameter at throat, diameter of turbo ventilator for performance of turbo ventilator. The present work aims to study the effect of throat diameter and wind velocity on the mass flow rate. These ventilators are tested on the experimental test rig. The CFD analysis is done to compare the experimental results.

2. Methodology

The test rig is designed and developed to test performance of different models of ventilators [16]. It is used to measure the mass flow rate of the turbo ventilator. Schematic diagram of experimental test rig is as shown in Fig. 2. Turbo Ventilator is placed on horizontal roof of a fully leak proof plenum chamber. The leakage test is performed on the test rig before experimentation. At first

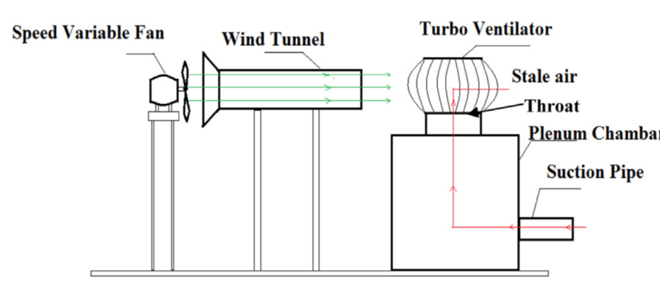

Fig. 2. Schematic diagram of test rig.

centrifugal blower is used to simulate the natural wind velocity ranging 1–10 m/s but distribution of wind velocity is not uniform on the ventilator. To overcome this problem wind tunnel arrangement is made to simulate the natural wind velocity in the range between 1 and 10 m/s. Wind tunnel gives the uniform distribution of air flow over turbo ventilator surface. First trial was made with 2.4 m tunnel length which showed some improvement in uniform pattern of wind velocity as compared to blower without wind tunnel. For further trial the length is increased to 4.8 m which gave satisfactory results. To reduce eddies formed and to get the uniform flow of air a honey comb structure was fabricated and installed at the outlet of the wind tunnel. Metrological data showed that the range of average wind velocity is 4–5 m/s (approximately) in urban parts of the India [23]. When wind is impinging on the turbo ventilator, it starts rotating and then it is interested that at what minimum wind velocity, turbo ventilator is extracting the stale air from the plenum chamber. Initially a nozzle meter is used to measure flow rate entering in to the plenum chamber. The air duct and nozzle meter offers resistance to air flow which is overcome by providing a booster fan on the upstream side of the nozzle meter. The booster fan is provided with variable speed drive and its flow is adjusted in such a way that the pressure at the inlet of the turbo ventilator is made to be atmospheric pressure. This process was requiring acute adjustment of the booster flow rate to make negative pressure at the throat of the turbo ventilator to atmospheric pressure. This adjustment was giving considerable error in the measured flow rate. Hence the nozzle meter arrangement is replaced by wind anemometer which was giving direct velocity at the suction pipe. We have ensured the correctness of these readings by calibrating with hot wire anemometer.

The mass flow rate of air is calculated by measuring velocity at the inlet of the suction pipe (150 mm diameter). This velocity of incoming air is measured inside the suction pipe with accuracy of 0.1 m/s using wind anemometer.

Mass flow rate is calculated in lps.

$Q = A \times V \times 1000.$

Where,

A = Cross sectional area of pipe in $m^2 = \pi/4 \times d^2$, d = diameter of pipe in m, V = Velocity of air at the suction pipe in m/s.

The measurement was carried out on three turbo ventilators with different throat sizes: one 600 mm curved blades ventilator, second 500 mm curved blades ventilator and third with 300 mm. (Refer Fig. 1).

3. Experimental results

The mass flow rates for the different wind velocities of different ventilator models are shown in Tables 1–3. The 300 mm curved vane ventilator is lighter in weight. It can be seen that the 300 mm

Table 1
Turbo ventilator (TV) with Throat Diameter 600 mm.

Sr. no.	Voltage of DC motor	Wind velocity (m/s)	Rotational speed (rpm)	Suction velocity of air (m/s)	Mass flow rate (lps)
1	65	1.01	14	0.2	3.52
2	90	2.00	28	0.5	8.83
3	115	2.75	39	0.8	14.13
4	120	3.44	44	1.2	21.12
5	160	4.55	69	1.9	33.44
6	180	5.25	76	2.2	37.10
7	215	7.60	95	2.8	49.17
8	280	8.87	109	3	51.24
9	340	9.5	120	3.1	53.01

Table 2
Turbo ventilator (TV) with Throat Diameter 500 mm.

Sr. no.	Voltage of DC motor	Wind velocity (m/s)	Rotational speed (rpm)	Suction velocity of air (m/s)	Mass flow rate (lps)
1	65	1.01	23	0.2	3.52
2	90	2.00	37	0.5	8.80
3	115	2.75	48	0.7	12.32
4	120	3.44	54	1.1	21.00
5	160	4.55	81	1.7	28.10
6	180	5.25	101	2.1	36.16
7	215	7.60	113	2.5	44.00
8	280	8.87	123	2.8	49.17
9	340	9.5	130	3	52.8

Table 3
Turbo ventilator (TV) with Throat Diameter 300 mm.

Sr. no.	Voltage of DC motor	Wind velocity (m/s)	Rotational speed (rpm)	Suction velocity of air (m/s)	Mass flow rate (lps)
1	65	1.01	30	0.2	3.52
2	90	2.00	45	0.4	7.04
3	115	2.75	56	0.6	10.56
4	120	3.44	62	0.7	12.32
5	160	4.55	107	1.6	28.16
6	180	5.25	120	1.9	33.44
7	215	7.60	132	2.4	42.24
8	280	8.87	151	2.8	49.17
9	340	9.5	195	3	52.8

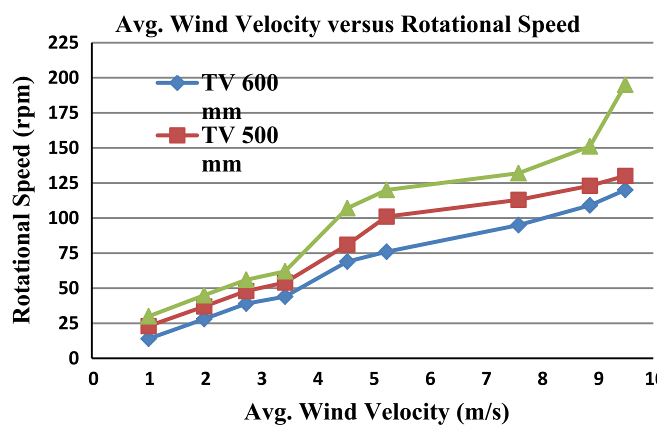

Fig. 3. Wind velocity versus rotational speed.

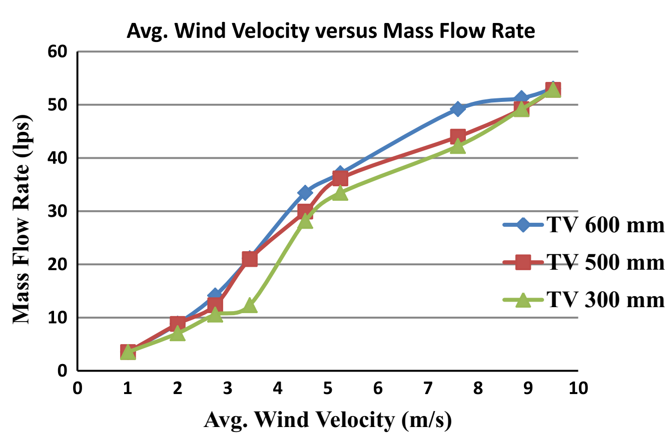

Fig. 4. Wind velocity versus mass flow rate.

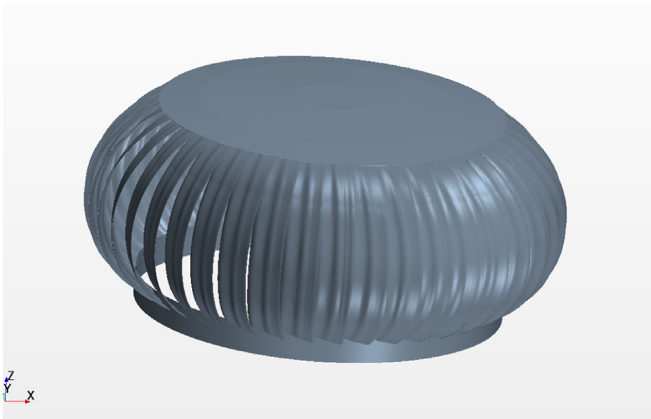

Fig. 5. Three dimensional model of turbo ventilator.

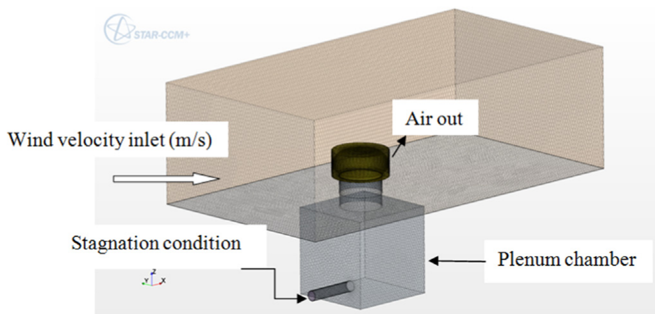

Fig. 6. Ventilator boundary condition.

curved vane ventilator has greater rotational speed at all wind speeds. This has caused a difference in their rotational speed for the same wind speed, as shown in Fig. 3. The measured mass flow rates of three turbo ventilators against average wind speed are shown in Fig. 4. It is found that the larger 600 mm size ventilator have significantly greater ventilation rates as they have larger throat diameter. However, there is a noticeable difference in mass flow rate between the 300 mm and 600 mm ventilators. The 600 mm turbo ventilator has greater ventilation capacity than the 500 and 300 mm ventilator. For example, at a wind speed of 5 m/s the 600 mm curved vane ventilator induces approximately 34 lps mass flow rate, the 500 mm vane ventilator induces about 30 lps and 300 mm vane ventilator induces about 28 lps. Suction velocity also recorded at the different wind velocities shown in tables. The highest rotational speeds recorded were around 195 rpm at 9.5 m/s wind speed of 300 mm size ventilator.

4. Validation of experimental results with CFD

An experimental result of turbo ventilator with 600 mm throat diameter shows the better mass flow rate. This is the reason why we have selected turbo ventilator with 600 mm throat diameter for the CFD analysis. To verify the experimental results, CFD analysis of 600 mm throat diameter has been carried out for wind velocities of 1 m/s, 2 m/s, 3.5 m/s, 5 m/s, 7.5 m/s and 9.5 m/s. The numerical simulation of the turbo ventilator is performed using STAR CCM+10.0. The equations for the conservation of mass momentum and turbulence are solved in STAR CCM+10.0. CFD studies are largely dependent on the meshing quality, turbulence models and boundary conditions. Several investigators [17–19] found that k-ε turbulence model provides reasonable estimate of the overall trend of airflow in terms of parameters such as pressure and velocity. Standard Realizable Two-Layer k-ε model is used in this study [20].

4.1. Geometry and boundary condition

The geometry of commercial available turbo ventilator with 600 mm throat diameter and blade height 300 mm is modeled in Computer Aided Three dimensional Interactive Application (CATIA) V5 R 20 [21]. It is then imported in to STAR CCM+10.0 [22] for meshing as shown in Fig. 5. A fine mesh of polyhedral scheme is used for meshing as this generates a fine mesh of 3 mm at the wall boundaries particularly at the blade surfaces of ventilator and a mesh size of 10 mm is used in the wind tunnel. The rectangular domain of size 6480 × 3600 × 2880 mm is used in the simulation. The total cell count in domain is 10,06,395. Simulation time using 32 GB RAM Dual core processor was approximately 750 h. Inlet velocities of 1 m/s, 2 m/s, 3.5 m/s, 5 m/s, 7.5 m/s and 9.5 m/s are used at the entry of turbo ventilator domain. Stagnation condition at inlet of suction pipe is assigned at the entry of the suction pipe as shown in Fig. 6. Biggest diameter of the turbo ventilator is d; the computational domain is taken for the simulation is three times d at the upstream and five times d for the downstream.

CFD simulation of turbo ventilator with 600 mm throat diameter is analyzed. The performance of a turbo ventilator is generally judged against its mass flow rate. Flow visualization and path lines were observed around and inside the turbo ventilator which as shown in Fig. 7.

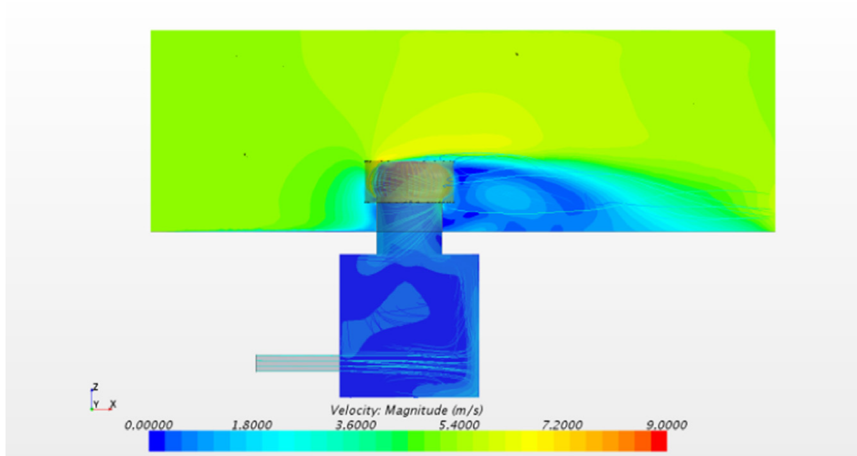

Fig. 7. Velocity plot of ventilator model.

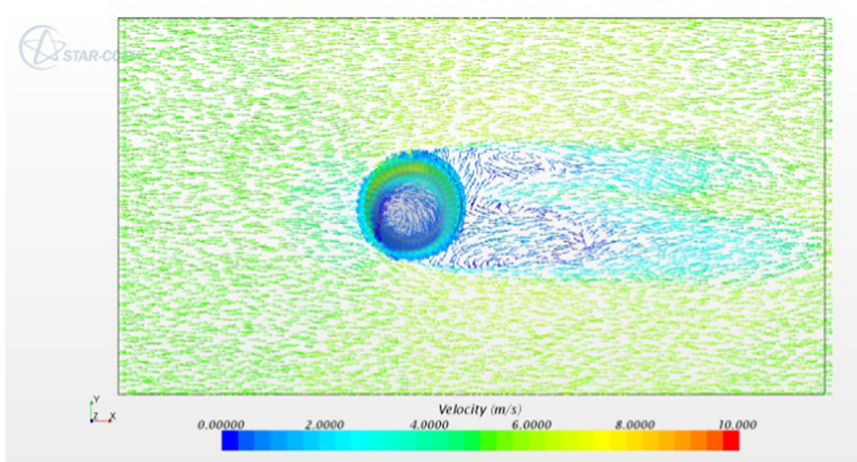

Fig. 8. Velocity vector contour (Plan view).

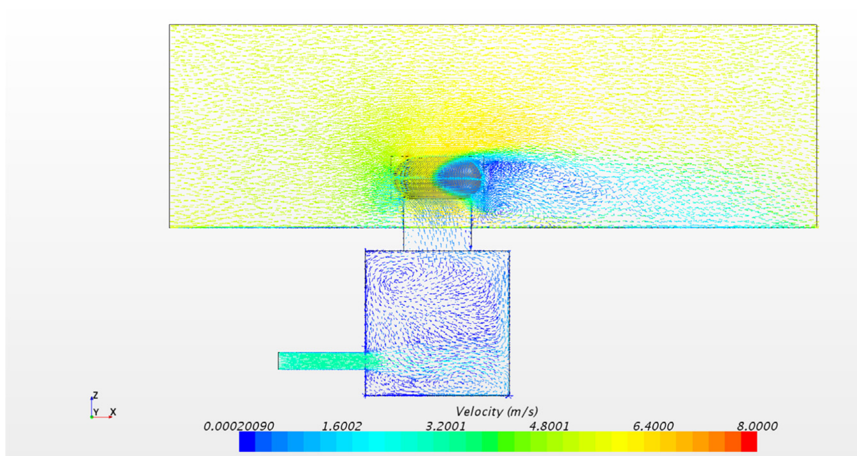

Fig. 9. Velocity vector plot.

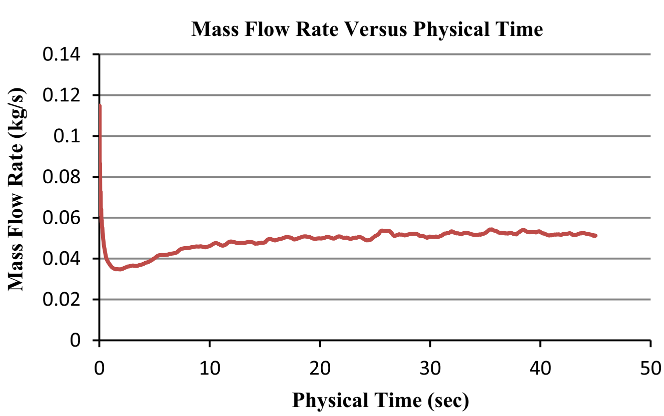

Fig. 10. Mass flow rate at 5 m/s wind velocity.

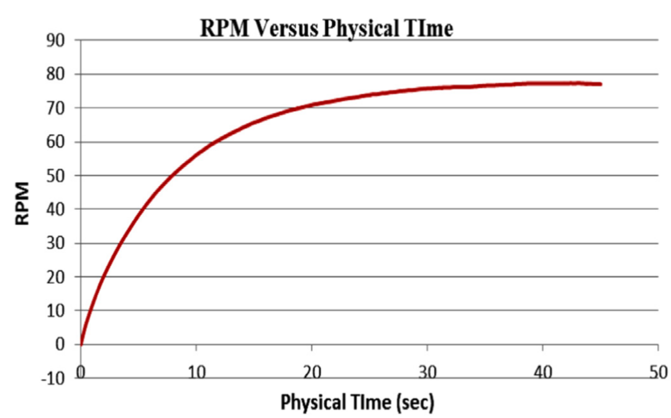

Fig. 11. Rotational speed verses physical time.

4.1.1. *Vortex formation within the ventilator*

A vortex is a region within a fluid where the flow is mostly a spinning motion about an imaginary axis. Ideally vortex formation should be at the center of turbo ventilator having vertical imaginary axis. However due to the entry of the air from one side the turbo ventilator the vortex shifts toward that side of the ventilator. Fig. 8 shows that there is low pressure at the center of vortex, which increases as it moves away from the center of ventilator.

4.1.2. *Vector Plot*

A velocity vector diagram is shown in Fig. 9 indicates direction of flow within and around the turbo ventilator. Vector quantity is displayed at discrete points (usually velocity with arrows) whose orientation indicates direction and whose size indicates magnitude. It is observed that upward arrows of velocity vector indicate that stale air is removed from the plenum chamber. It is clear from the Fig. 10 that, the simulation is converse and has reached to steady state condition after 40 s as physical time. Further concluded that trial simulation is completed and corresponding mass flow rate is 0.05 kg/sec (i.e. 40 lps). Fig. 11 indicates that rotational speed of turbo ventilator is 76 rpm achieved at 5 m/s. The results

Table 4
Experimental results validation with CFD.

Wind velocity (m/s)	Rotational speed of experimental results (rpm)	Rotational speed of CFD results (rpm)	Percentage difference (%)	Experimental results (Mass flow rate (lps))	CFD Result (Mass flow rate (lps))	Percentage difference (%)
1.01	14	16	12.5	3.52	4	12.5
2.00	28	31	10	8.83	10	11.70
3.5	45	50	10	21.12	24	12
5	70	76	8	34	40	15
7.5	98	110	10	49	56	12.5
9.5	120	136	11.7	53.01	62	14.51

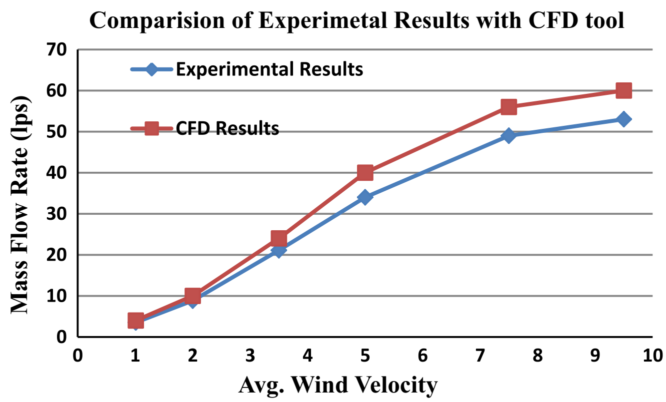

Fig. 12. Comparison of experimental results with CFD.

obtained from CFD and experimental reading for the same wind velocity were then compared and presented in Table 4. There is 12% to 15% higher prediction in CFD results. However both the analysis has shown the trend of increase in the mass flow rate with increase in wind velocity. Fig. 12 shows the comparison of experimental and CFD results for mass flow rate versus wind velocity characteristic.

5. Conclusion

Turbo ventilator models with throat diameter 600, 500, 300 mm are tested for their performance. The sufficient readings are taken to get the reliable data of the performance of the models. The mass flow rate of the ventilator is accurately measured with this test rig. The plot of mass flow rate versus wind velocity shows that the mass flow rate is increasing with the rotational speed of turbo ventilator and after a certain speed its rate decreases due to wind barrier. It is concluded that the ventilator has high extraction rate at higher wind speed. Turbo ventilator with larger throat diameter gives the more mass flow rate and less rotational speed compared to small size ventilator. The CFD results helped to study the internal as well as external flow around and in the turbo ventilator. The Experimental results for 1 m/s, 2 m/s, 3.5 m/s, 5 m/s, 7.5 m/s and 9.5 m/s wind velocities are compared with CFD analysis. CFD mass flow rate results are observed to be within 12% to 15% more than the experimental results. However both the analysis has shown the trend of increase in the mass flow rate with increase in wind velocity. The 12% to 15% more results obtained in CFD, this may be due to environmental variations during experimentation. However in CFD software, the boundary conditions remain same throughout the simulation.

Funding

The Authors acknowledge the research funds provided for this project by the Board of Colleges and University Development (BCUD), Savitribai Phule Pune University, Pune vide letter number: OSD/BCUD/360/135 Dated 27/11/2013.

Acknowledgment

Authors gratefully acknowledge the Dr. P.D. Patil, Chairman Dr. D.Y.P.I.E.T, Pimpri, Pune, Principal Dr. R. K. Jain and Mechanical Engineering department for providing the laboratory facilities and CFD software. The Authors acknowledge M/s Protech stampings Pvt. Ltd. Pune and Sinhgad college of Engineering, Pune for providing technical support and research facilities.

References

[1] V.H. Meadows, Rotary ventilator, 1929, US Patent 1857762.
[2] HVAC Applications, ASHRAE Handbook, 2011.
[3] Naghman Khan, et al., A review on wind driven ventilation techniques, Energy Build. 40 (2008) 1586–1604.
[4] Naghman Khan, et al., Performance of testing and comparison of turbine ventilators, J. Renew. Energy 33 (2008) 2441–2447.
[5] J.D. Dale, M.Y. Ackerman, Evaluation of the performance of attic turbo ventilators, ASHRAE Trans. 99 (1) (1993) 14–22.
[6] S. West, Improving the sustainable development of building stock by the implementation of energy efficient, climate control technologies, J. Build. Environ. 36 (2001) 281–289.
[7] C.M. Lai, Experiments on the ventilation efficiency of turbine ventilators used for building and factory ventilation, Energy Build. 35 (2003) 927–932.
[8] S.T.J. Lien, N.A. Ahmed, Effect of inclined roof on the airflow associated with a wind driven turbine ventilator, J. Energy Build. 43 (2) (2011) 358–365.
[9] M. Ismail, A.M. Abdul Rahman, Field study on the ventilation performance of the hybrid turbine ventilator (HTV) in the tropics, Int. J. Des. Manuf. Technol. 3 (2009).
[10] M. Ismail, A.M. Abdul Rahman, Comparison of Different Hybrid Turbine Ventilator (HTV) Application Strategies to Improve the Indoor Thermal Comfort, 2010.
[11] I. Mazran, The potential of hybrid turbine ventilator to improve indoor climatic conditions in hot-humid environment (Doctoral dissertation, Ph.D. thesis), University Sains Malaysia, Malaysia, 2010.
[12] K.M. Al-Obaidi, M. Ismail, A.M. Rahman, A comparative study between unvented and vented attics powered by the hybrid turbine ventilator in Malaysian houses, Int. J. Sustain. Energy (2014).
[13] I.S. Kuo, C.M. Lai, Assessment of the potential of roof turbine ventilators for bathroom ventilation, Build. Serv. Eng. Res. Technol. 26 (2) (2005) 173–179.
[14] K.M. Al-Obaidi, M. Ismail, A.M.A. Rahman, A review of the potential of attic ventilation by passive and active turbine ventilators in tropical Malaysia, Sustain. Cities Soc. 10 (2014) 232–240.
[15] T.H. Shieh, P.C. Chang, C.M. Chiang, C.M. Lai, Potential assessment of an innovative hybrid ventilator for building ventilation, J. Mech. Sci. Technol. 24 (11) (2010) 2341–2345.
[16] A. Revel, B.P. Huynh, Characterizing roof ventilators, Paper Presented at the 15th Australasian Fluid Mechanics Conference, The University of Sydney, Australia 13–17 December 2004.
[17] McGown and Gopalarathnam, Computational study of a circulation controls

airfoil using FLUENT, in: Progress in Astronautics and Aeronautics, AIAA, 2006, pp. 539–554.
[18] C.M. Mak, Francis W.H. Yik, A study of natural ventilation in a kitchen using computational fluid dynamics (CFD), Arch. Sci. Rev. 45 (2002) 1–8.
[19] N.A. Ahmed, A novel small scale efficient wind turbine for power generation, Renew. Energy 57 (2013) 79–85.
[20] Shao-Ting J. Lien, et al., Numerical simulation of rooftop ventilator flow, Build. Environ. 45 (2010) 1808–1815.
[21] CATIA V5 User's Guide, Available at: 〈https:/www.3ds.com/product/catia〉.
[22] CCM STRAR+User's Guide, Available at: 〈http://steve.cd-adapco.com〉.
[23] 〈http://www.synergyenviron.com/tools/wind_data.asp?loc=Pune%2CMaharashtra%2CIndia〉.

Ganesh K. Jadhav is M. Tech in Mechanical Engineering and working as Assistant Professor at Dr. D. Y. Patil Inst of Engg and Tech, Pimpri, Pune since last seven years. He is pursuing his Ph.D. degree at Sinhgad College of Engineering, Pune.