International Journal of Scientific and Technical Advancements

ISSN: 2454-1532

CFD based Modelling of a Ceiling Fan in a Room

Shubham Kumar Verma[1], Yatheshth Anand[1], Sanjeev Anand[2]
[1]Department of Mechanical Engineering, Shri Mata Vaishno Devi University, Katra, J&K, India-182320
[2] Department of Energy Management, Shri Mata Vaishno Devi University, Katra, J&K, India-182320
Email address: sverma4585@gmail.com

Abstract: Ceiling fans have been utilized from decades as a means for giving thermal comfort in tropical nations. Although, recent years have seen a huge increment in the utilization of air conditioning as a way to accomplish comfort, and also, the aggregate energy utilization and related CO_2 emissions. Ceiling fans are as yet practical choices to restrict utilization of air conditioners or in combination with air conditioners systems without negotiating on thermal comfort and still accomplishing energy saving. Ceiling fans create non-uniform velocity profiles, and hence generally non-uniform thermal environment, whose parameters might be hard to investigate with straightforward modelling methods. This issue can be examined by utilizing Computational Fluid Dynamics (CFD). Although, to date, there are only few works related to Ceiling fans, CFD and thermal comfort. More exact models are in this manner required to predict their performance. The paper presents a three-dimensional transient implicit CFD model of a typical Ceiling fan accessible in India and also studied the flow and velocity profile of air in three different planes in a confined volume of a single person hostel room located in Shri Mata Vaishno Devi University.

Keywords: Ceiling fan; Thermal Comfort; Computational Fluid Dynamics; Turbulence modelling; K-ω model

I INTRODUCTION

Ceiling fans have been used since decades as a means to improve indoor thermal comfort in buildings in tropical and subtropical climatic zones. The fans there offer a technically simple, inexpensive, individually operable and, above all, effective method to increase air movement and thus thermal comfort in a room [1]. It comprises of a get together of an electric motor with 3-4 blades suspended from the roof of a room. In spite of its straightforwardness and boundless use, the stream instigated by a roof fan in a shut room has not been explored, and problematic plans are in wide utilize. There is tremendous potential for energy saving and enhanced comfort by creating improved fan plans. This work builds up a principal comprehension of the stream qualities of a roof working inside a closed room [2]. In Indian residences, ceiling fans are as common as electric light bulbs, being present in almost every habitable space. They are part of most of Indian residences, and they are widely used in both old and more recent buildings [3,4]. In case of developing economy and rising level of populace which can bear the cost of procurement and operation of air-conditioning system for higher need of Thermal comfort, India has encountered ascend in sales of ventilation systems [5]. The power demand for space cooling involves up to 60% of summer crest load in huge urban communities, for example, Delhi [6]. Roof fans are likewise utilized as a part of workplaces, living arrangements as a contrasting option to broaden the late spring comfort envelope. These fans are of reasonable cost, basic in development, simple to introduce, and needn't bother with consistent or refined support. The flow pattern features instigated by roof fans are extremely useful for individuals of enthusiasm working in the field of HVAC. This flow is probable to be rotational, three-dimensional and turbulent. Along these lines, knowing the stream qualities, because of roof fan turn would enable enhancing the fan to outline notwithstanding choosing its ideal situation to spare energy. Unless the utilization of energy demanding air-conditioners are restricted just to times of greatly sweltering climate, at that point general Indian vitality utilization and related CO_2 emanations will fundamentally expand, prompting extreme ramifications for the worldwide atmosphere and furthermore difficult the dependability of the Indian power network. Ventilating more often than not gives uniform warm natural conditions; in this way fashioners can anticipate with certainty its execution utilizing conventional warm solace models [7].

To date, there are few existing reports on the utilization of CFD to show roof fans. Bassiouny et al. [8] executed straightforward 2D models. Along these lines, the really three-dimensional conduct of airstream produced by a roof fan isn't caught. Momoi et al. [9,10] grown excessively complex displaying approaches that are constrained in their application in light of the fact that numerous estimations are required due to the required information.

The objective of this study is to determine velocity and flow of air originating from the fan in the room without any inlet and outlet. So for that 3 planes were taken in the room i.e. first one at distance 0.15 m from the floor, second one is 0.68 m from floor and 3rd one is 1.2 m from floor namely Plane-1, Plane-2 and Plane 3 respectively. And have studied the various velocity contours variations in different Planes.

II SITE DESCRIPTION

The site which has been chosen is the room of the Trikuta Hostel of Shri Mata Vaishno Devi University (SMVDU) campus Kakryal close Katra town in Reasi District of Jammu and Kashmir. It is Located at a separation of just about 45 km from Jammu Airport and 14 km short of the holy town of Katra, the college is situated on a plateau surrounded by mountains on three sides in the foothills of the Trikuta Range where the sacred holy place of Mata Vaishno Devi is found. Katra is having regular hot and damp atmosphere and situated at 32.98°N, 74.95°E and has an elevation of approximately 2,474 feet [11]. The 2- dimensional sketch diagram of the room have been shown in three different views, that is, front, top, and side, are taken so that proper orientation of the site could be depicted and are shown in Figures 1(a), 1(b), and 1(c), respectively.

Shubham Kumar Verma, Yatheshth Anand, and Sanjeev Anand, "CFD based modelling of a ceiling fan in a room," *International Journal of Scientific and Technical Advancements*, Volume 4, Issue 2, pp. 219-224, 2018.

ISSN: 2454-1532

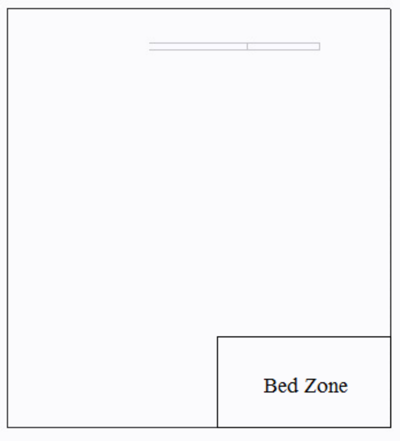


Figure 1(a) : Front view of room

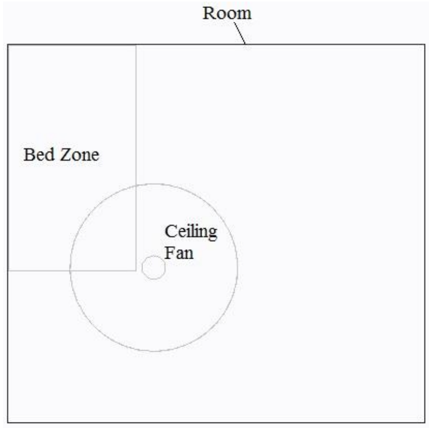
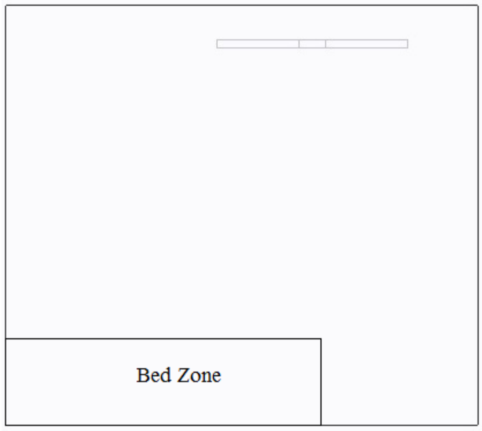


Figure 1(b) : Top view of room



1Figure 1(c) :Side view of room

III Modelling

Figure 2 represents the 3-d model of room created on Creo 2.0. Room created is having dimensions 3.48*3.16*3.1 high which was about the size of a single-person hostel room. Room is considered as closed volume and no inlet and outlet considered. As it is considered that room is closed and all windows are closed. The roof fan has been demonstrated as a momentum source. Displaying the actual blades would require extremely nitty gritty data about their geometry, and would prompt a significantly higher number of mesh elements and to the utilization of a moving mesh. Both these features would essentially increase the demand for computational power, and the conceivable sources of uncertainty, without ensuring better outcomes, yet constraining the ease of use and appropriateness of the model. Along these lines, the fan has been displayed as a ring with a diameter of 140 cm, and having separation from the roof of 254 mm, as that of actual fan, and with a central cylindrical solid element, since in a real roof fan no air radiates from the inside. The volume of the field for simulation was 33.72 m3. The rotational zone comprises of a tube shaped ring and a disc. Thus, the rotational zone involved around 1.1% of the recreation field.

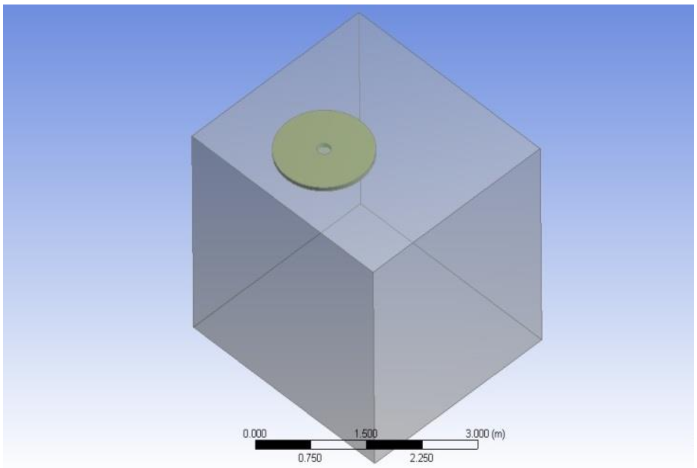


Figure 2: Three dimensional view of the room

IV Meshing

In this study, Ansys 18.2 student version is used to generate the mesh. Tetrahedron meshing is accomplished with the skewness of 0.9. The room as shown in Figure 2 is divided into a finite number of control volumes and grid points. Each grid point found in the computational domain is surrounded by one volume. All the variables chosen for the calculations are solved in these points. The calculation results by differential equations in these locations are replaced by discrete values. The current research used only the structured Tetrahedron mesh with number of elements reached up to 98002 as shown in Figure 3. The mesh independence test conducted on the model reveals that the solution of the problem is independent on the mesh size taken. And so the different mesh values need not to be taken. A general mesh file is shown in the below figure 3. As we can see that mesh is fine nearby fan. So room is meshed with element size of 164.4 mm and fan with 1.28 mm.

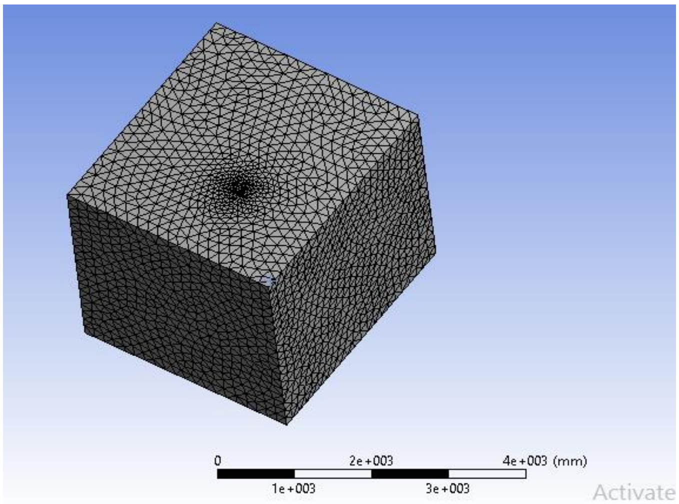


Figure 3: Mesh File of the Modeled Room

V Simulation

The computational fluid dynamics has been extensively and promisingly used in many complicated applications. In the present study, a finite-element Ansys, is used to simulate the flow pattern induced by a ceiling fan in a space. Fluent is used

Shubham Kumar Verma, Yatheshth Anand, and Sanjeev Anand, "CFD based modelling of a ceiling fan in a room," *International Journal of Scientific and Technical Advancements*, Volume 4, Issue 2, pp. 219-224, 2018.

to run the fluid based simulations. We have opted for Double precision based serial model in Absolute formulation to enhance accuracy and robustness of the results. Transient simulations have been performed to better model the real behavior of the ceiling fan. Moreover, considering this roof fan model as a part that can be utilized with advanced transient thermal comfort models, running transient simulations from the earliest starting point makes its relevance less demanding and more dependable. The air stream produced by a roof fan is exceptionally turbulent. Along these lines, picking the most fitting turbulence model is fundamental with a specific end goal to get exact outcomes. The SST k-ω turbulence model delivered the most exact and practical outcomes, being better than the other turbulence model considering both subjective and quantitative outcomes given by Babich *et.al*, 2017 [5]. Convergence criteria have been set equivalent to 1e-04 for the RMS residuals. Since there are no inlet and exhaust in this model, the conservation target does not have any impact. Besides, the sub-space used to display the fan essentially goes about as a momentum source, yet can't produce any mass imbalance as there are no physical obstructions between this sub-area and the rest of the piece of the room, and there is no mass generation or dispersal inside this sub-domain. The starting time step was 0.01s, with the most extreme and least esteems set equivalent to 0.01s separately. The energy source that simulates the real fan has been connected to the subdomain utilizing tube shaped parts: axial component 52 kg/m^{2}s^2, radial component 0 kg/m^{2}s^2 and theta component 7 kg/m^{2}s^2. The axial component pushes air downwards while the theta segment creates rotational movement. Calculation has been finished with 10 time steps of 10 s each and 2000 iterations.

VI Validation

To confirm the accuracy of the simulations the values of velocity, the model of Environmental chamber of CEPT University, Ahmedabad is developed in Creo-Parametric and applied same boundary conditions as applied by Babich[5]. Then comparison of our results with the results shown by Babich[5] has been done. For validation we have taken two Planes (i.e. 0.7 m and 1.3 m above the floor) during simulation as taken by Babich *et.al*,[5] in the Environmental chamber. In both planes Velocity is measured at 5 different Points (Centre, east corner, west corner, North and south corner as per Babich *et.al*, [5]) and compared with those which are measured by Babich *et.al.* [5] The comparison between both the measured and simulated values is shown below in Table 1 and Figure 4.

Table 1: Variation between Velocities measured in Babich model[5] and Simulated model

Distance from Centre of Fan	Measured at 1.3 m from floor(m/s)	Simulated at 1.3 m from floor(m/s)	% Error	Measured at 1.3 m from floor(m/s)	Simulated at 1.3 m from floor(m/s)	% Error
Centre	2.3	2.05	10.87	2.02	1.85	8.41
R200	2.5	2.41	3.6	1.98	1.8	9.09
R500	1.5	1.2	20	1.6	1.45	9.37
R600	0.99	0.86	13.13	0.8	0.81	1.25
R800	0.7	0.63	10	0.45	0.45	0
R1200	0.5	0.5	0	0.36	0.37	2.78
R1700	0.4	0.44	10	0.2	0.24	20
R2000	0.3	0.31	3.33	0.18	0.16	11.11

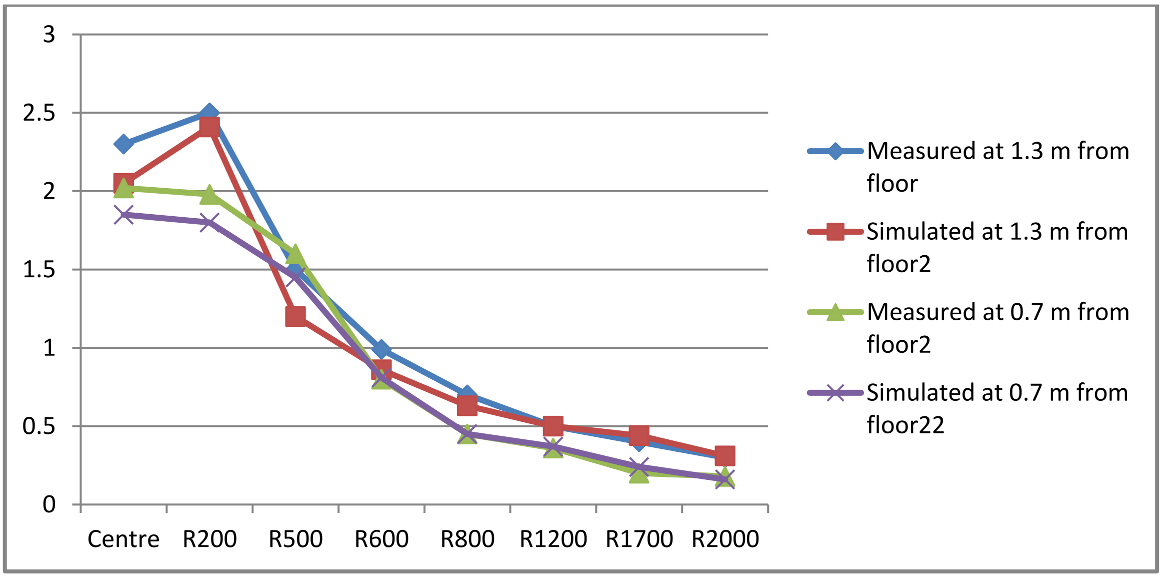


Figure 4: Graph shows Velocities at 1.3 m from floor of Babich[5] model

VII RESULTS

Shubham Kumar Verma, Yatheshth Anand, and Sanjeev Anand, "CFD based modelling of a ceiling fan in a room," *International Journal of Scientific and Technical Advancements*, Volume 4, Issue 2, pp. 219-224, 2018.

The results from the study have revealed the full complexity of Computational Fluid Dynamics simulations in buildings. The gathering of input data for modeling and generation of Boundary conditions for Computational Fluid Dynamics is very complex and not fully understood. The current work used many different methods to promote Computational Fluid Dynamics modeling. There are many different models utilized for the analysis based on the literature survey and each model is analyzed for different Boundary conditions and flow conditions. Based on detailed analysis of the various models, optimum conditions and model was adopted on the basis of Babich model and final analysis is performed on the problem.
Figure-6(a) shows the streamlines originated from the fan outer ring. There are 25 streamlines originated and showing how air is flown in a closed room. As we are able to see that the velocity of streamlines is varying from 0.0 m/s^2 to 2.2 m/s^2. As we can see that there are regions just below the fan are red in color which shows the maximum velocity and nearby the boundaries there are some regions where the velocity is zero which means that air is not circulated there. But if we see the location of bed so it is along the one of the boundary and there will be thermal stratified zones there.

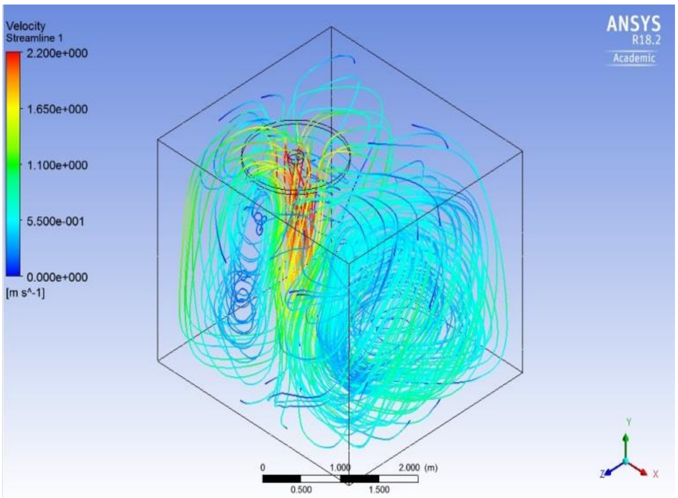


Figure 6(a): Streamlines showing air flow in room

Figure-6(b) shows the air speed distribution along the horizontal plane at distance 0.15 m above the floor (Plane-1). Reason for choosing the plane at distance 0.15 m above floor is that in hostel room when we sit on a chair then our foots are below the table at distance of 0.15 m from floor. So its important to study the contours in this plane. Also it can be seen that the velocity range in this plane is varying from 0.0 m/s^2 to 2.2 m/s^2. But here at distance 0.15 m above the floor velocity is zero only nearby boundaries. So we can say that here when we sit on chair then air is also circulated to our foot and there wasn't any thermally stratified zone. Figure 6(c) showing the velocity contours at distance 0.68 from the floor. The reason behind choosing this plane is to study the circulation of air when we are sleeping on the bed. As we can see that velocity is again varying from 0.0 to 2.2 m/s^2. If we see the bed zone then we will be able to say that when we were sleeping then there was some area where we observed almost 2 m/s^2 of velocity, but there are some zones in the bed zone where we were observing even 0 m/s^2 (approx.) so we call it thermally stratified zone. Figure 6(d) showing the velocity contours at distance of 1.2 m in room. The reason behind choosing the plane at distance 1.2 m from the floor is to investigate the flow nearby face when we were sitting on a chair or on bed.

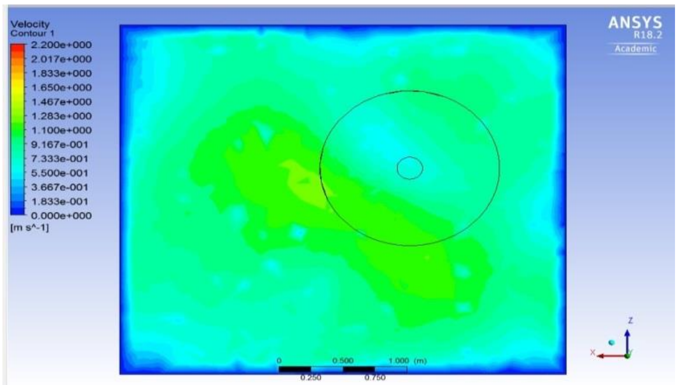


Figure 6(b): Velocity Profile on Plane-1 in room

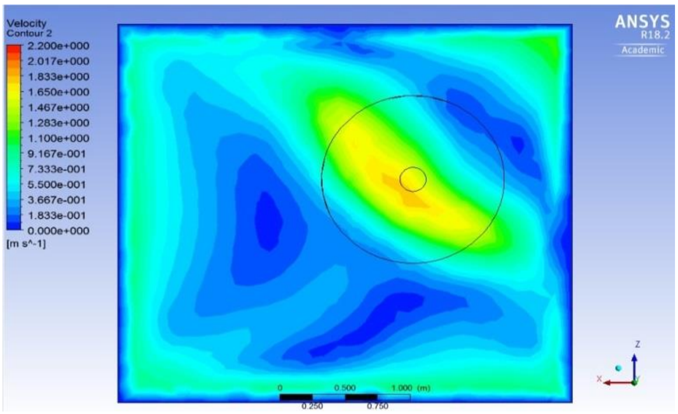


Figure 6(c): Velocity Profile on Plane-2 in room

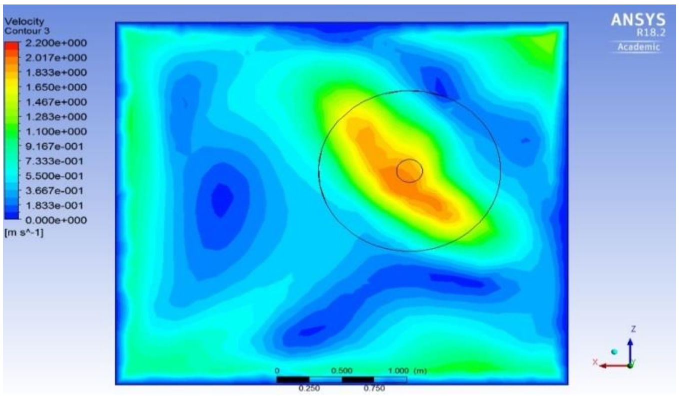


Figure 6(d): Velocity Profile on Plane-3 in room

VIII CONCLUSIONS

In this paper, study of air flow originated from the ceiling fan on three different Planes i.e.0.15 m above the floor (Plane-1); 0.68m above the floor (Plane-2) and 1.2m above the floor (Plane-3) has been done. The results showed that air flow on Plane-1 is low but almost uniform in the whole room and also sufficient as per the requirement. But in the Plane-2 and Plane-

3, there were some zones in the Planes where air speed is approximately zero and were called as thermally stratified zones. As position of the bed in our hostel is fixed (as bed is cemented type) so it is not possible to change its location. So it can be said that it needs some improvement either in fan location or bed for better circulation of air.

IX REFERENCES

1. S.C. Sekhar, Higher space temperatures and better thermal comfort—a tropical analysis. Energy and Buildings. 1995 Oct 1;23(1):63-70.
2. A. Jain, RR Upadhyay, S Chandra, M Saini, S Kale. Experimental investigation of the flow field of a ceiling fan. InASME 2004 Heat Transfer/Fluids Engineering Summer Conference 2004 Jan 1 (pp. 93-99). American Society of Mechanical Engineers.
3. S. Manu, Y. Shukla, R. Rawal, L.E. Thomas, R. de Dear, Field studies of thermal comfort across multiple climate zones for the subcontinent: India Model for Adaptive Comfort (IMAC), Build. Environ. 98 (2016)
4. M. Indraganti, Thermal comfort in apartments in India: adaptive use of environmental controls and hindrances, Renew. Energy 36 (2011)
5. F. Babich, M Cook, D Loveday, R Rawal, Y Shukla. Transient three-dimensional CFD modelling of ceiling fans. Building and Environment. 2017 Oct 1;123:37-49.
6. DSLDC, Hourly Demand Data from the State Load Dispatch Center, 2012.
7. P.O. Fanger, Thermal Comfort, McGraw-Hill, New York, USA, 1970.
8. R. Bassiouny, N.S. Korah, Studying the features of air flow induced by a room ceiling-fan, Energy Build. 43 (2011).
9. Momoi Y, Sagara K, Yamanaka T, Kotani H. Modeling of Ceiling Fan Based on Velocity Measurement for CFD Simulation of Airflow in Large Room. Proceedings of 9th international conference on air distribution in rooms, vol. 1, Coimbra, Portugal: 2004.
10. Momoi Y, Sagara K, Yamanaka T, Kotani H. Modeling of prescribed velocity generated by ceiling fan based on velocity measurement for CFD simulation. Proceedings of 10th international conference on air distribution in rooms, vol. 1, Helsinki, Finland: 2007.
11. Anand Y, Gupta A, Maini A, Gupta A, Sharma A, Khajuria A, Gupta S, Sharma S, Anand S, Tyagi SK. Comparative thermal analysis of different cool roof materials for minimizing building energy consumption. Journal of Engineering. 2014;2014.

Shubham Kumar Verma, Yatheshth Anand, and Sanjeev Anand, “CFD based modelling of a ceiling fan in a room,” *International Journal of Scientific and Technical Advancements*, Volume 4, Issue 2, pp. 219-224, 2018.

Shubham Kumar Verma, Yatheshth Anand, and Sanjeev Anand, "CFD based modelling of a ceiling fan in a room," *International Journal of Scientific and Technical Advancements*, Volume 4, Issue 2, pp. 219-224, 2018.