Simulation Of Air Flow Around Ceiling Fan In An Enclosed Space By Using Cfd

J.Bala Bhaskara Rao, D. Bhanuchandra Rao, J.Siddhartha Yadav, M.Sreerama.

Abstract: In this research the impact of room stature and fan speed on wind flow design and in like manner, room air movement nature of the 3-dimensional transient model of a ceiling fan utilizing Computational Fluid Dynamics (CFD). This is done by experimenting estimations of air speeds and violent motor vitality at different unequivocally found planes, in a structure envelope, for accomplishing human comfort, utilizing ceiling fan by ANSYS 15.0. Working the fan at higher rotational rates doesn't generally protect agreeable air circulation. Subsequently an ideal speed can be suggested, contingent upon the fan good ways from the floor. As, the progression of air originating from fan is having violent nature and there are such a large number of models accessible, so picking the right model is need. The fan speed, measurement, and number of sharp edges, cutting edge, and location all these assume a significant job in deciding the required stream design included in the space. In this investigation, the experimented work is accomplished for the generally utilized discontinuous model .This model is utilized to anticipate active and scattering energies to acquire the disturbance power generated.

Keywords: Roof fan, Turbulence, velocity, CFD, ANSYS, k-turbulence model.

1. INTRODUCTION

Ceiling fans have been utilized for quite a long time as methods for giving thermal comfort in tropical climates. In Indian residencies; ceiling fans are practically present in each living space which is utilized in both old and new structures. The fans there offer in truth straight forward, efficient, and freely operable and, in particular, successful procedure to build development of air and at last warm comfort in a room. This section examined about meaning of roof fan, history, utilizes, portions of roof fan and arrangements. Ceiling fan is a mechanical fan mounted on the roof which has cutting edges at the corners to circulate the air. Mechanically it has revolving vanes to produce currents of air, which are generated through low pressure and high volume of air. The actual working process of a ceiling fan includes evaporative cooling of sweat with increased heat convection into the surrounding air. Since, the ceiling fan is the most widely used source of cooling so an effective method to increase its effectiveness is quite demanding. Hence a brief literature survey stated the following. Ankur&Rochan et al [1] studied about the air flow of a ceiling fan. They concluded the maximum range to be around 2 to 3 m/s for the study. Vane anemometers were used to study the performance. Parker &Challahan et al [2] studied about the blade designs to develop a streamlined flow with uniform air development. Results from this study declared improved air flow and decrease in power consumption from around 10 to 28 %. Bhortake&Lachure et al [3] studied about the streamlined effectiveness of air by studying on the edges of the fan blades of a ceiling fan. Its results stated that the maximum air output was discovered with a fan of four sharp edges. Schmidt & Patterson [4] studied about the electrical consumption and its dependence on volume flow rate & mechanical force of a room ceiling fan. They concluded that the maximum flow rate is obtained at the tip of the edges of the fan and decreased as the air flow went down towards the floor. Tian and Gruyters et al.,[5] 2018 checked on the flow and potential applications utilizing co-reproduction and distinguish future research needs on coupling building vitality experimentation (BES) and Computational Fluid Dynamics (CFD)model. They have built up a transient CFD model to analyze temperature elements in a cool store and contemplated the Impacts of cooling control on quality change and vitality utilization. et al., 2018 surveyed the flow and potential applications using co-recreation and distinguish future research needs on coupling building vitality reenactment (BES) and Computational Liquid Elements (CFD) And built up a transient CFD model to analyze temperature elements in a cool store and considered the Impacts of cooling control on quality change and vitality utilization. In and Hsieh [6] anticipate and pinpoint the stream design, wind stream rate, productivity, and clamor for roof fans with various structure parameters utilizing numerical and test techniques. The examination reports that for a lacking lodging, 'breathe consequently' marvel can happen that influences the exhibition and force utilization of a roof fan since its gulf and outlet are nearly situated at a similar plane have done sharp edge parametric examinations and explored the effect of number of cutting edges on roof fan execution individually. Aftereffects of progress in air conveyance, mass stream rate and administration estimation of Business roof fans by changing cutting edge geometry are accounted for. Aziz &Shahat et al [7] studied about the impact of vortices on round and square diffusers of a roof ceiling fan. Experimental and numerical studies were done on the roof ceiling fan. Nawaz &Kanti et al [8] studied about various 14 CFD models of different diffusers and fan rotors.Utilizing these models different flow patterns were generated. In this study a novel way of installing a ceiling fan within the ceiling was developed. Ho &Rosario [9] used a novel method of combining air conditioner with ceiling fan to get few conclusions. They aimed at finding the best thermal comfort for consumers using this technique. A closer look at the experimental and numerical works reveals that air flow induced by ceiling fan in a room is influenced by various parameters like blade angle, blade span, number of wings, height from the ceiling, speed etc. Among these factors blade angle and speed plays key role in enhancing air flow. Hence , in this study a CFD experimentation is done using the k-E turbulence model & the results are compared with experimental data ,Hence an aim is done in the direction of studying air flow in an enclosed room to find the distribution of air by changing various factors and knowing their effects of uniform distribution of air inside the room.

2. PROBLEM DEFINITION AND BOUNDARY CONDITIONS

In present situation increment in comfort levels assumes key job while utilizing roof fan in obliged Room. The virtual models of fans are as shown in fig.1 &2. This can be accomplished by shifting rotational paces and by various sharp edge points and so on. By utilizing CFD reproduction programming, it can diminish the time and activity cost compared by numerical investigation so as to quantify. The element of the room model is 3 cubic meters. The pivoting velocities of fan are 250,300,350 and 400 rpm. The roof fan has three and four sharp edges. The reason for the reenactment is to break down the stream in room by fluctuating distinctive cutting edge points relating to their rotational rates. Wind current is defined as the movement of air in the room starting with one spot then onto the next. Wind stream rate is the most significant parameter to be considered as the exhibition acquired from ceiling fan. By utilizing CFD modeling programming, estimations can be done experimentally at a lower cost and the ideal parameters can be found out in a shorter time.

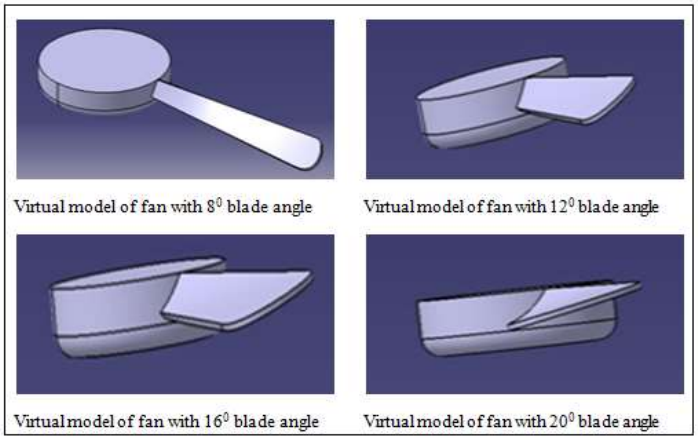

Figure.1 *Design of different blade angles*

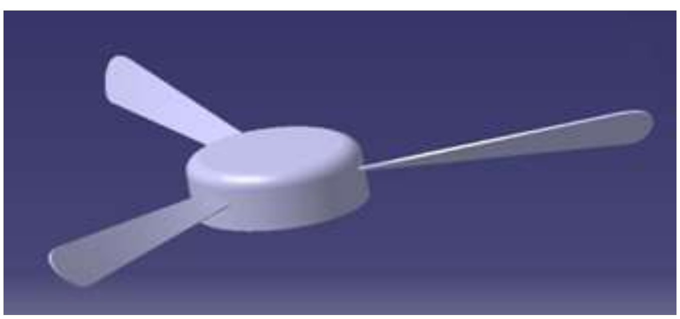

Figure.*2Modeling of the fan with different blades*

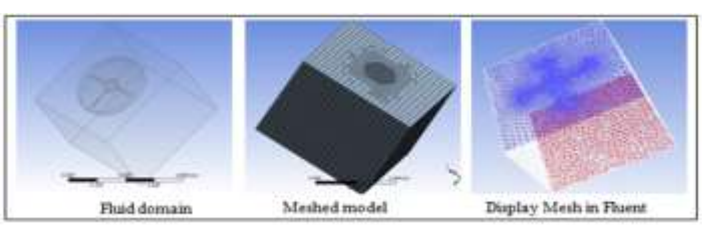

Fig. 3 *CFD models of Fluid, Meshed & Meshed display*

Table.1 *Boundary conditions for velocity & Kinetic energy*

	Boundary Condition	Velocity Magnitude	Turbulent Kinetic Energy
Inlet	Velocity inlet	1 m/s	0.00375 m^2/s^2
Outlet	Pressure Outlet	-	-

The differential conditions of a CFD program should be furnished with limitations and starting (for transient arrangements) conditions for complete arrangement over the model of interest. The limiting conditions for the most part include speeds, (or stream rate in lieu of them), weight and temperature over the bounding surfaces. The way towards understanding a fluid stream problem if frequently considered being the extrapolation of a lot of information characterized on the bouncing forms or surfaces into the space inside. It is, in this manner, significant for the client to gracefully genuinely reasonable and all around presented limit conditions to guarantee exact and stable The boundary conditions for the model are defined as per the model design as shown in figure.3. There are two inlets and two outlets for the design. The respective boundary conditions with inlet conditions are mentioned in the table.1 below as a sample. The desired fluid is added by using the create/edit option from the FLUENT database.

3. RESULTS AND DISCUSSION

The Experimental analysis started with the design of geometry and mesh for the structure. Once the model is prepared the boundary conditions and input conditions are instilled into the model. As meshing is defined as the discretization of created model into small volumes and solving it by using iterative methods. The analysis is followed by the results and discussions section followed by the conclusion. The designed model from CATIA is imported first and the analysis is carried out next. The impact of mounting a roof fan in rooms at specific statures and furthermore with explicit rotational velocities dependent on air development was considered to reach a few inferences that can help in vitality reserve funds. Figures beneath speak to the conveyance of air speed and turbulent kinetic energy. It describes that ceiling fan is turning at 400 RPM and edge of 12 degrees, which results the best from the roof enthusiast of 3 wings and 4 wings. The wind stream goes into the room through the flexibly diffuser on the highest point of the floor at uniform max throttle (1.0m/s). The effect of speeding up in the prompted stream is appeared in figure at the chose intermediate and most extreme fan speeds. It tends to be seen that speeding up brings about reenactment increment in neighborhood air speed in the room at various statures, especially near the fan. The figure.4 shows that speeding up from the middle of the road to most extreme speed prompts nearly multiplying the air normal speed esteems in the room. This is expected to builds the fan power utilization. The impact of mounting roof fan in rooms of various RPM's and Sharp edge edges was considered to make a few determinations that can help in energy reserve funds. The impact of speeding up on room walled in area, it tends to be seen that in-wrinkling the fan speed would expand the stream whirl and as needs be the air choppiness force. The outcomes from the investigation have uncovered the full intricacy of Computational Liquid Elements (CFD) reproduction in structures. The social occasion of info information for demonstrating and age of Limit conditions for Computational Liquid Elements (CFD) is perplexing and not completely comprehend.

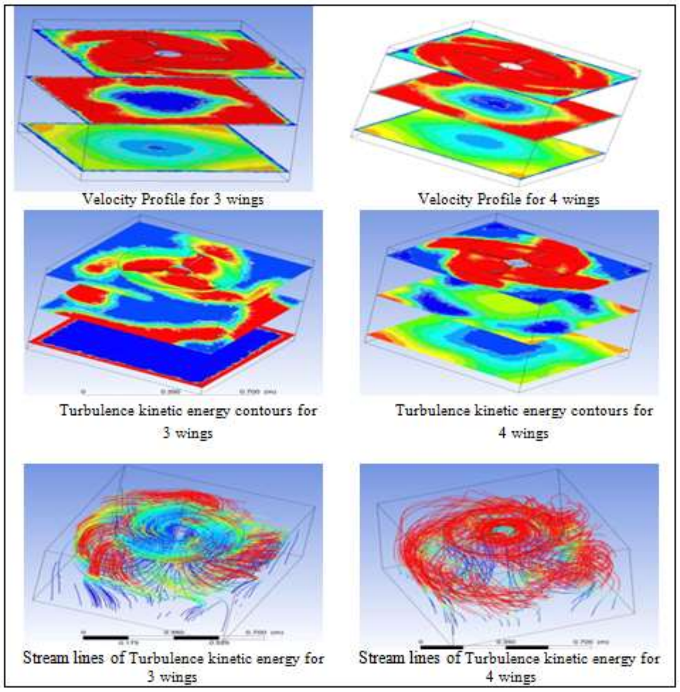

Figure.5 *Velocity distribution of 4 wing ceiling fan*

3.1 Velocity distribution representation

The following graph shows velocity distribution of 4 wing ceiling fan. Here the Velocity ranges from 1.08 m/s to 1.42m/s. Velocity of air increases with increasing the fan speed but, on other hand maximum velocity was founded at 12 degree Blade Angle. The present work utilized a wide range of techniques to advance Computational Liquid Dynamics displaying. In view of point by point investigation of the different models, ideal conditions and model was embraced based on K-Epsilon model and last examination is performed on the issue

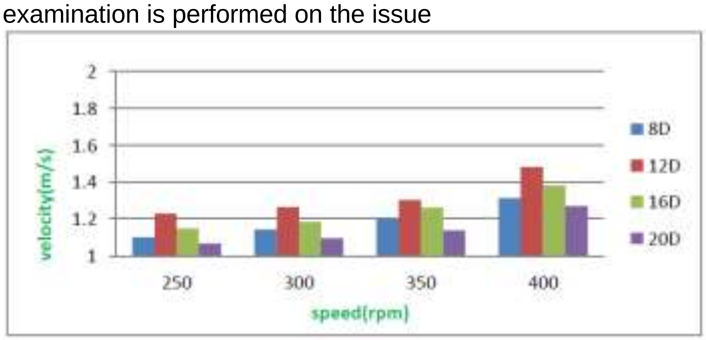

Figure.5 *Velocity distribution of 4 wing ceiling fan*

The following figures 5,6&7 shows velocity distribution of 3 wing ceiling fan. Here the Velocity ranges from 1 m/s to 1.4m/s. Velocity of air increases with increasing the fan speed but, on other hand maximum velocity was founded at 12 degree Blade Angle.

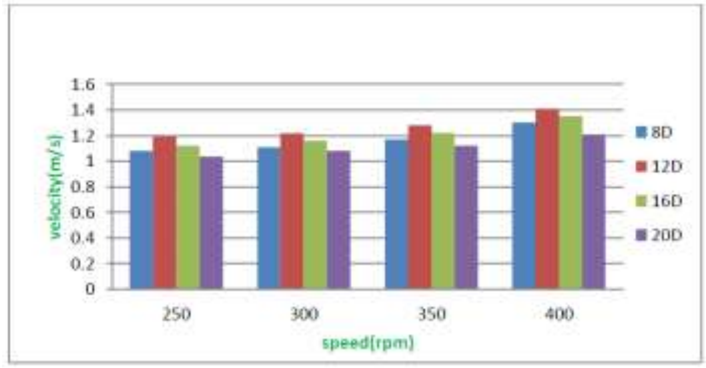

Figure.6 *Velocity distribution of 3 wing ceiling fan*

The following graph shows the comparison of velocity distribution of 3 wings and 4 wings. A fan with 4 wings produces comfort conditions with respect to their blade angles and rotational speeds. . It shows the streamlines of speed and turbulent kinetic energy of 3 wings and 4 wings began from the fan external ring. There are 250 streamlines showing how air is flowing in a shut room. As we can see that the speed of streamlines out is fluctuating from 0.0m/s to 2.2 m/s. As there are areas just underneath the fan red in color which shows the greatest speed and close by the limits there are a few locales where the speed is zero which implies that air isn't moving there.

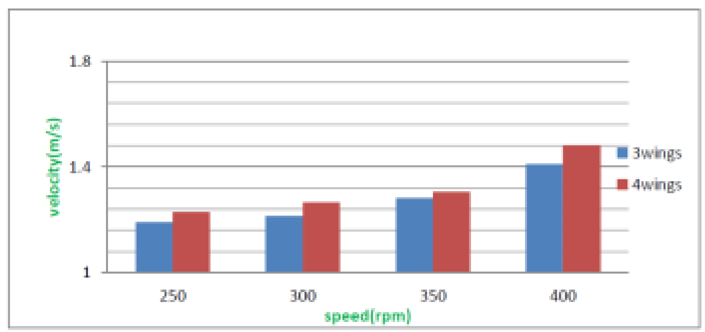

Figure.7 *comparison of velocity distribution of 3 wings and 4 wings*

3.2 Turbulence kinetic Energy representation

The following figures 8, 9 &10 shows the Turbulence Kinetic Energy Distribution of 3 wing fan. Here the Turbulence ranges from 0.0035m/s^2 to 0.0042m/s^2. Turbulence of air increases with increasing the fan speed but, on other hand optimum turbulence condition was founded at 12 degree Blade angle.

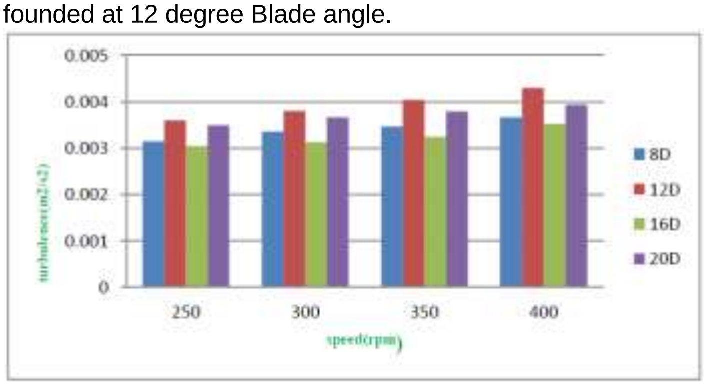

Figure.8 *Turbulence kinetic energy distribution of 3wing fan*

The following graph shows the Turbulence Kinetic Energy Distribution of 4 wing fan. Here the Turbulence ranges from 0.0037m2/s2 to 0.0044m2/s2. Turbulence of air increases with increasing the fan speed but, on other hand optimum turbulence condition was founded at 12-degree Blade angle.

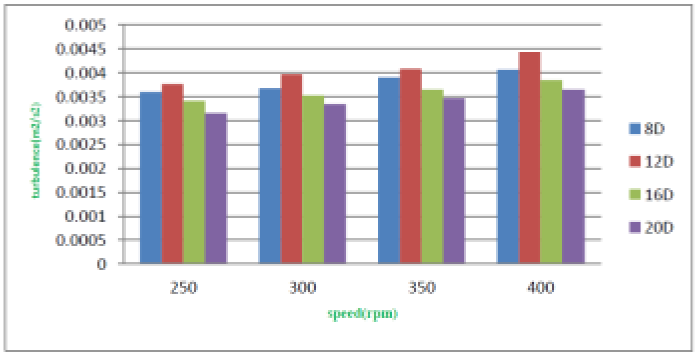

Figure.9*Turbulence kinetic distribution of 4 wing fans*

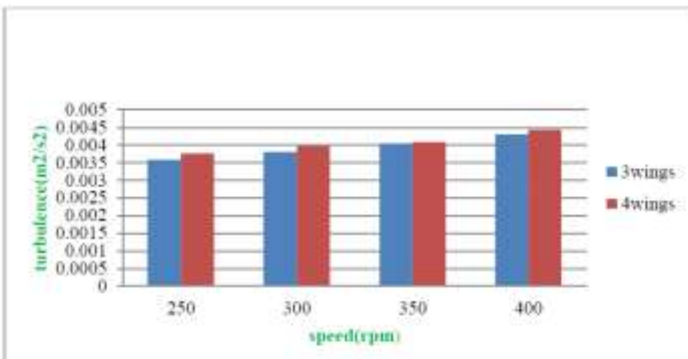

Figure. 10 *Comparison of turbulence distribution of 3 winged and 4 winged fans*

The following graph shows the comparison of turbulence distribution of 3 wings and 4 wings. A fan with 4 wings produces comfort conditions with respect to their blade angles and rotational speeds.

4 CONCLUSIONS

In this project, study of air flow originated from ceiling fan in an enclosed room was observed on different planes similarly stream lines of air flow also observed by using CFD software. We have taken different readingsof velocity and turbulent kinetic energy for different blade angles at varying rotational speeds and we can say that normal blade with 12 degrees has more average velocity. The simulated results were validated with experimental results and were obtained below 5% error. The inferences from the study are:

- The main aim of this study is to identify the air flow of a ceiling fan in a closed room. The floor affects the air flow pattern greatly in varying different rotational speeds and blade angles of ceilingfan.
- To facilitate this analysis, the ANSYS15.0 workbench finite volume analysis program was used to perform the modeling.
- The first stage of this project was to design a model, where velocity and turbulence pro- files were analyzed for air flow in constrained room and this basic model was analyzed. The basic model for analysis comprised of models of 3 wing and 4 wing fan in an en- closed space in which simulation of air flow has been has been carriedout.
- In the next stage variation of parameters under different blade angles and rotating speeds of the fan has been studied with respect to velocity and turbulence kinetic energy for the 4 and 3 wings respectively. The results so have been obtained were validated with references as mentioned in previouschapter.
- In the room with 3 wing fansgave optimum velocity and turbulence were obtained at 400 rpm with 12-degree blade angle. In the room with 4 wing fansgave the results of optimum velocity and turbulence to be 400 rpm with 12-degree bladeangle.
- A fan with 4 wings produced comfort conditions with respect to their blade angles and rotational speeds i.e:12 degree and 400 rpm.

REFERENCES

[1] Ankur, J., R. U. Rochan, C. Samarth, S. Manish and K. Sunil (2004). Experimental Investiga- tion of the Flow Field of a Ceiling Fan. ASME 2004 Heat Transfer/Fluid Engineering Summer Conference, Charlotte, North Carolina,USA.
[2] Parker, D. S., M. P. Challahan, J. K. Sonne, G. H. Su and B. D. Hibbs (2000). Development of a High Efficiency Ceiling Fan. Improving Building Systems in Hot and Humid Climates, Tex- as A&MUniversity.
[3] Bhortake, R. V., P. S. Lachure, S. R. Godase, V. V. More and K. S. Chopade (2014). "Exper- imental Analysis of Air Delivery in Ceiling Fan." International Journal of Emerging Technology and Advanced Engineering 4(6),247-251.
[4] Schmdit, K. and D. J. Patterson (2001). Performance result for a high efficiency tropical ceil- ing fan and comparisons with conventional fansdemand side management via small appliance efficiency. Renewable energy 22(3), 169-176.
[5] Lin, S. C. and M. Y. Hsieh (2014). An Integrated Numerical and Experimental Analysis for Enhancing the Performance of the Hidden Ceiling Fan.Hindawi.
[6] Aziz, M. A., I. A. M. G. Shahat, F. A. Mohammed and R. H. Mohammed (2012). Experi- mental and numerical study of influence of air ceiling diffusers on room air flow characteristics. Energy and Buildings 55,738–746.
[7] Ho, S. H., L. Rosario and M. M. Rahman (2005, August). Effect of using ceiling fan on hu- man thermal comfort in Air-conditioned space. In Procedings of 3rd International Energy Con- versation Engineering Conference, San Francisco,California
[8] Rohles, F. H., Konz, S. A., & Jones, B. W. (1983). eilin Fans Extenders of the SummerCom- fort Envelope. ASHRAE Trans, 89,245263.
[9] Morton-Gibson, M; Coutier,P.J.; Place,W.J. Effects of fan velocity on thermalcomfortinan office building, Proceedings of the 10th National Passive SolarConference, the Solar Energy & Utilities Conference, the DaylightingApplicationsConference.