2018 Building Performance Analysis Conference and
SimBuild co-organized by ASHRAE and IBPSA-USA
Chicago, IL
September 26-28, 2018

TRANSIENT 3-D MODELLING OF CEILING FAN FOR ACHIEVING THERMAL COMFORT

Y. Anand[1], S.K. Verma[1] and S. Anand[2*]
1. School of Mechanical Engineering, Shri Mata Vaishno Devi University, Katra, J&K, India
2. School of Energy Management, Shri Mata Vaishno Devi University, Katra, J&K, India

ABSTRACT

In this study, our aim is to check, compare and validate the 3-dimensional transient model of a ceiling fan using Computational Fluid Dynamics. This is done by comparing the experimentally measured values with simulated values of air velocities at various precisely located points, in a building envelope, for achieving thermal comfort, using ceiling fan by ANSYS 18.2. As, the flow of air originating from fan is having turbulent nature and there are so many models available, so choosing the correct model is necessity. In this study, the comparison of the experimental work is done for the four most widely used turbulence models i.e. the standard k- ω, the SST k-ω, the standard k-ε and the Realizable k-ε. After the analysis and comparison it was found that the result obtained from the SST k-ω are best suited to model for this type of problem, as the simulated values thus obtained shows agreement of 94% with the experimented values.

Keywords: Computational Fluid Dynamics, Thermal Comfort, Building Envelope, Turbulent, SST k-ω, Realizable k-ε, standard k-ε and standard k- ω.

INTRODUCTION

Fans have been utilized since past many years in order to improve indoor thermal comfort in buildings in tropical and subtropical climatic zones (Jain *et al.,* 2004). The fans there offer an in fact straight forward, economical, independently operable and, most importantly, effective technique to increase movement of air and ultimately thermal comfort in a room (Sekhar *et al.,* 1995). This is generally done by removing/ circulating the warm stratified air from/in the room. The downwash propelled by foil (rotating) drives the warm air downwards to blend with the cold air on ground level, countering the impacts of buoyancy. Fans are also widely used and accepted, due to their cost effectiveness and also good availability (Li *et al.,* 2016).

Energy preservation and rising fuel costs makes fans a natural decision with regards to saving our normal assets. Fans not just circulate cool air in the summer, but they additionally help with the development of warm air in the winter. Amid the summer months, a fan working in the typical mode can diminish how cool you feel in a room by as much as eight degrees, allowing you to raise the temperature of your air-conditioning system's and decrease the energycosts (Hawaii Energy Code, 2015). In the winter, fans ought to be worked in reverse mode to recycle the warm air caught at the roof. Something other than practical machines, the present fans have turned into an imperative design component in homes and are accessible in a wide cluster of sizes and styles (Ceiling Fans, 2013).

Introducing a roof fan in each living space will enable occupants to feel cooler while sparing energy. Furthermore, on the off chance that you have not updated your roof fans in the previous eight years or something like that, then you are missing an opportunity to save the energy requirement and minimizing your electricity bill also (Ceiling Fans, 2013). According to John moody, ceiling fan manufacturer "a roof fan can spare property holders as much as 40 % on their air conditioner bills by making a breeze that influences the ambient temperature to feel seven or eight degrees cooler than it really is.

Individuals feel inconvenience when someone sweats in a room/space with air present in room. Subsequently, individuals attempt to make air breeze around the bodies either normally or mechanically to upgrade body convective warmth exchange. Air movement helps sweat vanishing and, appropriately brings body comfort feeling. It is extremely troublesome for some individuals in developing nations to have an air-conditioning system to accomplish the conditions suitable for comfort in indoor. Rather, individuals depend for the most part on natural ventilation. Roof fans are utilized as a part of workplaces, living arrangements as a contrasting option to expand the comfort envelope during summer season. Generally, fans are of reasonable cost, basic in development, simple to fix, and needn't bother with maintenance. The features of the flow pattern prompted by roof-fans are extremely useful for those individuals of interest who were working in the field of Heating Ventilation and Air-Conditioning (HVAC) (Bassiouny *et al.,* 2011).

To date, there are few existing reports on the utilization of CFD to show roof fans.

Rohles *et al.,* 1982 & 1983 studied the usefulness of ceiling fans in order to enhance the comfort experimentally by investigating the 256 subjects under various air velocity and temperature in an environment chamber equipped with a ceiling fan. The results showed that an air plume originating from ceiling fan with velocity between 0.5 and 1.0 m/s compensates fora 2.8–3.3 °C temperature change; this represents an energy saving of 15–18%.

Morton-Gibson *et al.,* 1985 investigated the effects of ceiling fans or individual fans on thermal comfort in an office building and found that operating fans for about 1000 hour per year at 26.7 °C results in approximately the same comfort levels as 24.4 °C without fans and that the resulting savings are more than the cost and energy usage of the fans.

Tian *et al.,* 2018 reviewed the current and potential applications using co-simulation and identify future research needs on coupling building energy simulation (BES) and Computational Fluid Dynamics (CFD).

Gruyters *et al.,* 2018 developed a transient CFD model to analyze temperature dynamics in a cool store and studied the Effects of cooling control on quality change and energy consumption.

Krole *et al.,* 2018 presented the brief review over the commonly used models of ceiling fan and expresses some issues over the selection of model.

Chen *et al.,* 2018 investigated the influence of various parameters over air distribution and found that air velocity is only affected in the given cylindrical zone.

Verma *et al.,* 2018 modelled a ceiling fan for a hostel room of a University by using K-ω Turbulence model in order to study the air flow in room. They studied the velocity contours of a room at three different heights. As a result they observed that on the bottom plane (plane near to floor) velocity of air is low but it is uniform and there weren't stratified zones in the plane whereas in above planes there were stratified zones.

Balbich *et al.,* 2017 developed and validated a 3-dimensional transient implicit CFD model of a typical ceiling fan by comparing simulation results obtained using different URANS turbulence models with measured data in a controlled environment.They demonstrated ceiling fan as a source of momentum. In order to reduce the required computational power, fan has been utilized as a hollow ring with a central cylinder-shaped solid element. As a result, they highlighted that this ceiling fan model is able to replicate the predominant characteristics of the air flow generated by the fan. The best results are achieved when the SST k-ω turbulence model is used, with 83% of the simulated values being within the error bars of the respective measured value.

In present study, our aim is to make and validate a 3D fan model commonly used for air circulation in a particular space at a typical location and compare the measured data with the simulated results from 04 turbulent models.

METHODOLOGY

There are two approaches which are used during the study. First one is experimental and other one Computational fluid Dynamics based simulations.

Experimental Setup

The experiment was conducted in the two office spaces (Room-1 and Room-2) situated in Central workshop of Shri Mata Vaishno Devi University, Kakryal in District Reasi of Jammu and Kashmir. The location 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 (Anand *et al.,* 2014). Room 1, as shown in figure having a volume dimensioned as 3044×4076×3300 mm respectively whereas Room 2 is divided into two sections due to its structure and is having the dimension 3140×3040×3300 mm, and 3000×2780×3300 mm, respectively.

The ceiling fan used in this work has a sweep diameter of 1400 mm. There was a single fan (Fan 1) in Room 1 whereas 2 fans (Fan 2 and Fan 3) in Room 2 as shown in Figure 2(b). The speed of Fan 1, Fan 2 and Fan 3 are measured to be 275 rpm, 170 rpm and 255 rpm, respectively. The fan is installed at 0.82 m of distance from the ceiling. It is assumed that there is no air exchange between the room under investigation and the environment. This was done by taking into account the closed doors, windows and other vents in the respective rooms. The diameter of central hub of the fan is 200 mm and blades are approximately 600 mm in length. During experiment, all the room furniture and other objects were taken out of room to cause minimum obstruction to the air flow.

Measurements of velocities were taken in three different vertical planes i.e. 0.15 m from floor (Plane 1), 0.65 m from floor (Plane 2) and 1.2 m from floor (Plane 3) respectively. In each plane, velocities are measured at 8 points in Room 1 and 16 (eight for each fan) points in Room 2. The point where the measurements are taken are depicted in the Figure 1 shown below.

The readings were taken with the logging duration of around 5 min. It implies that when a reading was recorded then after recording it the anemometer has been placed stationary for around 5 min and next reading was taken after that time period, so as to let the anemometer come to rest thus not influencing the other values.

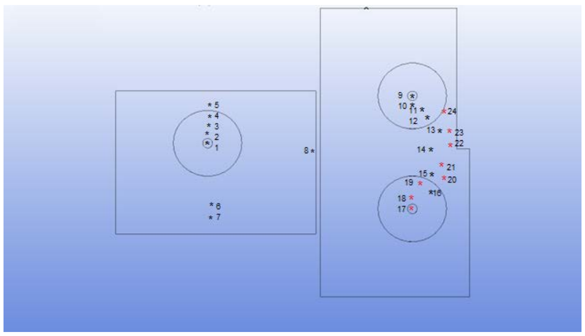
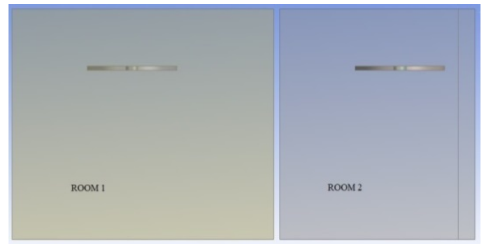


Figure 1 Top view of the rooms showing points where measurements were recorded

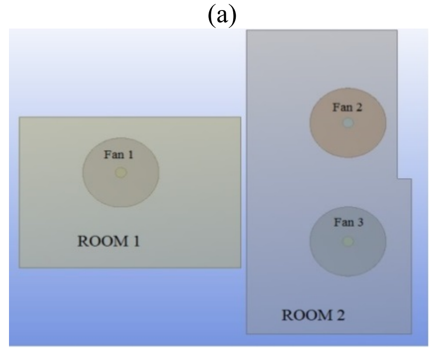
This process was repeated for all the readings. Digital anemometer AM- 4201 is used to measure the air velocities at different points shown in Figure 1. Digital non-contact type Tachometer is used to measure the fan speed (rpm). Anemometer AM-4201 which has been used for measurement of the air velocity can measure velocity only in one direction, which implies that we can measure only speed of the air, neither the components of velocity of air (axial, radial & tangential) separately. The measuring range of the anemometer is 0.4 m/s to 30 m/s with resolution of 0.1 m/s and accuracy of ± (2%).

CFD Based Approach

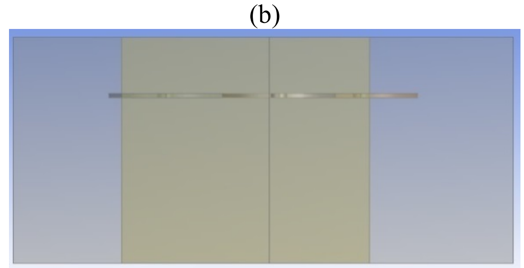
An identical setup, as utilised for the experiment, was then reproduced using CREO-Parametric and was investigated in the CFD program using ANSYS 18.2 Academic version. The 2D sketch of the room is shown in three different views, that is, front, top, and side, so that the proper orientation of the site could be depicted and are shown in Figures 2(a), 2(b), and 2(c), respectively.



(a)



(b)



(c)

Figure 2(a) Front view, (b) Top view, and (c) Side view, of Rooms

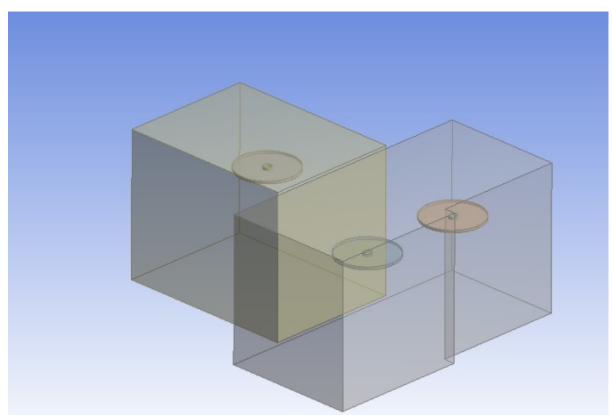


Figure 3 3D model of Room

Figure 3 represents 3D model of the rooms having the original dimensions to that in experimental setup. As doors and windows were kept closed during experiment so we have not included the doors and windows in the model considered room as close volume and conducted simply isothermal iterations in order to decrease the computational power required. The ceiling fan had been demonstrated as a source of momentum (Babich et al., 2017) as shown in Figure 4.

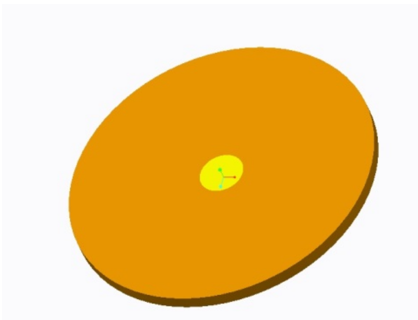


Figure 4 3D model of ceiling fan

Displaying the actual blades would require extremely nitty gritty data of the geometry, and also prompt significantly larger number of elements in mesh and high utilization of a moving mesh. Due to these features, the required computational power demand and uncertainty sources is going to increase, without ensuring superior results, yet constraining the convenience and fittingness of the model. Along these lines, the fan has been utilized as a hollow ring of diameter of 140 cm, and having separation from the roof of 820 mm, as that of the fan used during experiment, with a central cylinder-shaped solid element, since in a real roof fan no air radiates from the inside. The volume of the field for simulation was approximately 99.97 m3. The rotational zone comprises of a tube shaped ring and a disc. Thus, the rotational zone involved around 1.11% of the recreation field.

For this model, ANSYS 18.2 academic version is used to generate the mesh. The curvature type size functions of mesh used with the skewness of 0.9. Both the rooms are distributed into number of grids and elements. Every grid point found in the computational space is encompassed by one volume. Every one of the factors decided for the computations are solved in these points. The calculation results by differential equations in these locations are replaced by discrete values.

The mesh independence was also conducted with the help of four different meshes as tabulated in Table 1. As the result of mesh independence test, we got to know that results are independent of the type of mesh selected. Owing to the computation power requirement and to have a healthy compromise between the results and the limitation of the nodes, as in the ANSYS Academic version an appropriate mesh was used. The current study involves the meshing of room and fan with the maximum size of 146.47 mm and minimum size of 0.73 mm with growth rate of 1.2 (Mesh 3). It involves 32000 nodes and 150580 elements. Figure 5 shown below shows the mesh generated by Mesh 3.

Table 1 Different mesh used for mesh independence test

	Maximum size (mm)	**Minimum size (mm)**	**Nodes**	**Elem-ents**
Mesh 1	500	2.5	6116	19381
Mesh 2	250	1.25	11083	43489
Mesh 3	146.47	0.73	32000	150580
Mesh 4	95.25	0.48	80128	405655

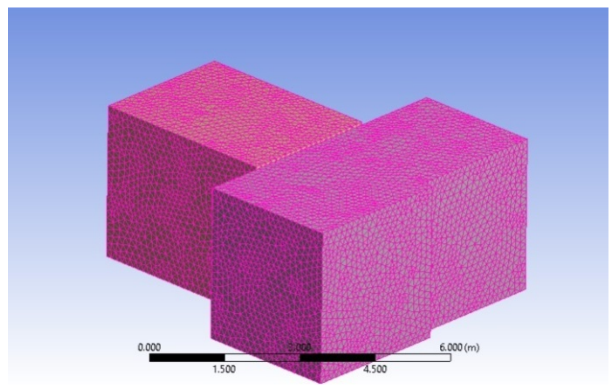


Figure 4 Meshing of the room model

The CFD has been broadly and promisingly utilized as a part of numerous muddled applications. In the present investigation, Fluent, a module of ANSYS, is utilized to simulate the pattern of flow induced by a roof fan in a space. Double Precision based serial simulations has been opted in Fluent set up. Transient pressure based solver has been opted to study the real behavior of fan. Absolute formulation has been adopted to ensure the robustness and accuracy in the Problem.

Also, considering the rooftop fan as a section which is going to be used along with the advanced transient models of comfort, so running the transient recreations from the most starting stage makes its pertinence not only much requesting but rather more reliable. The air stream created by a rooftop fan is extraordinarily turbulent. Thusly, picking the most fitting turbulence model is essential with a particular ultimate objective to get correct results. For ensuring better results we have performed simulations on four different models of Fluent i.e. Standard k-ω, SST k-ω, standard k-ε and Realizable k-ε models.

The criteria for convergence has been settle equivalent to 10-04 for the RMS residuals. As doors and windows are closed during the experiment so we can say that there isn't any inlet or exhaust in the model. Thus, conservation target is not going too impacted by this. Besides, the sub-space which is utilized to display the fan essentially goes about as a source of momentum. Yet it is not going to produce any im-balancing in mass so there weren't any physical obstructions between this sub-area and the rest of the part of the room. The time step in the start was settled to 0.1s, with the most extreme and least esteems set equivalent to 0.1s separately. The source of energy/momentum which is going simulates the real fan has been connected to the subdomain utilizing tube shaped parts. Fan 1 has been applied with axial component 60 kg/m2s2, radial component 0 kg/m2s2 and theta component 8 kg/m2s2 whereas Fan 2 has been applied with axial component 45 kg/m2s2, radial component 0 kg/m2s2 and theta component 8 kg/m2s2 and Fan 3 with axial component 52 kg/m2s2, radial component 0 kg/m2s2 and theta component 8 kg/m2s2. The theta segment given to cylindrical disc creates the rotating movement whereas axial component work is to push the air downwards.

VALIDATION

Validation has been done by comparing the measured and simulated values at 24 points in each of the three Planes i.e. 0.15 m above the floor, 0.65 m above the floor and 1.2 m above the floor. Comparison of values at total of 52 points has been done. Results show that there is some variations at some points and also there were some points in the planes where measured and simulated value are same. It has been largely observed that Variation in the values at some points is very much. We can say that it is due to the range of measuring instrument. As the range of Anemometer is between 0.4 m/s and 25 m/s, so it may be possible that values given below 0.4 m/s by Anemometer is not accurate enough for the velocities below 0.4 m/s.

RESULTS

In the present study we have studied our rooms using four different turbulence model i.e. SST k-ω, Standard k-ω, Realizable k-ε and standard k-ε models. While considering the different models for simulation, it has been observed that SST k-ω shows the excellent agreement with the values which are measured experimentally. Actually when all the 72 points are considered, it has been observed that, there was an average error of 6% when SST k-ω model has been used. Table 2 shows the variation while we are using different models at different distance from the floor. It has been observed that SST K-ω is the promising model which shows the better result as that of realistic values found during the experiment.

Here we can see the variation of simulated and measured values at 1.2 m height from the floor as shown in Figure 5. It can be seen in case of SST K-ω that the deviation of measured and simulated value is within the acceptable range at all the points. Even it has been clearly seen that at point 5 both values coincides.

Table 2 Average error of respective models in different Planes

Height from floor (m)	SST k- ω	standard k-ε	Realizable k-ε	Standard k-ω
1.2	4.5	25.23	11.31	13.5
0.65	5.47	30	15	17
0.15	8	20	48.5	50
% Error	6	25	25	27

However in case of Realizable K- ε it has been observed that deviations are within permissible limits for point 1-6,8,10,11,12,14-24. But much greater deviation has been observed for the remaining points. Similarly in case of Standard K-ω, the points in which deviation is more than permissible are at points 8-10, 13, 17 and18. But in case of Standard k-ε there are only 5 points which are in permissible range i.e. Points 5, 9, 10, 16-20 and 24.

Now, if we will see the variation of simulated and measured values at 0.65 m height from the floor. It can be seen in Figure 6 that in case of SST K-ω, the deviation of measured and simulated value is within the acceptable range at all the points except point 1.

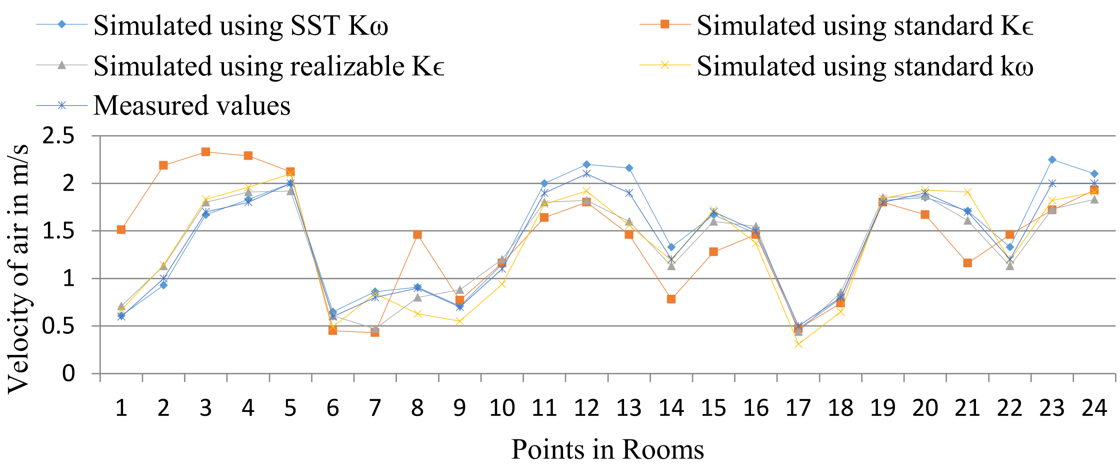


Figure 5 Comparison of the measured values of velocity with simulated values in different models at 1.2 m of height from floor

This is due to the fact that the value measured at point 1 is below 0.4 m/s but the range of our anemometer is between 0.4 and 25.0 m/s. However in case of Realizable K- ε it has been observed that deviations are within permissible limits for all points except 1, 2, 6,13 and 23. Also there is very high deviation observed for the point 6. Similarly in case of Standard K-ω, the points in which deviation is more than permissible are 2, 3, 6, 10, 16, 17 and 18. Again in this model also deviation at Point 6 is very high. But in case of Standard k-ε there are only 5 points which are in permissible range i.e. Point 5, 8, 9, 10, 12, 14 and 24.

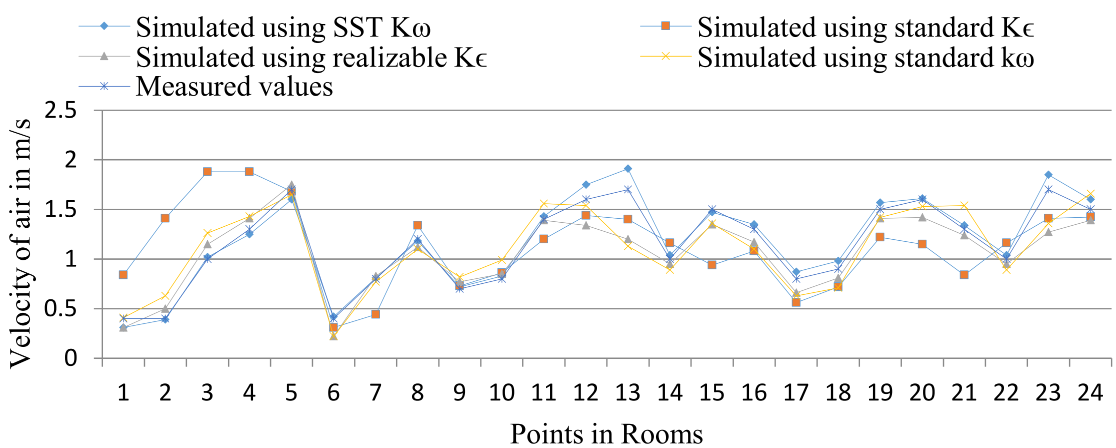


Figure 6 Comparison of the measured values of velocity with simulated values in different models at 0.65 m of height from floor

However, when we see the Figure 7 it has been observed that air has been circulated uniformly in whole room and the there wasn't any stratified zone formation at 0.15 m above the floor. If we talk about SST K-ω model, then there were three points on which deviation from measured value is zero and remaining are in permissible range. But in this plane remaining all other model shows very high deviations.

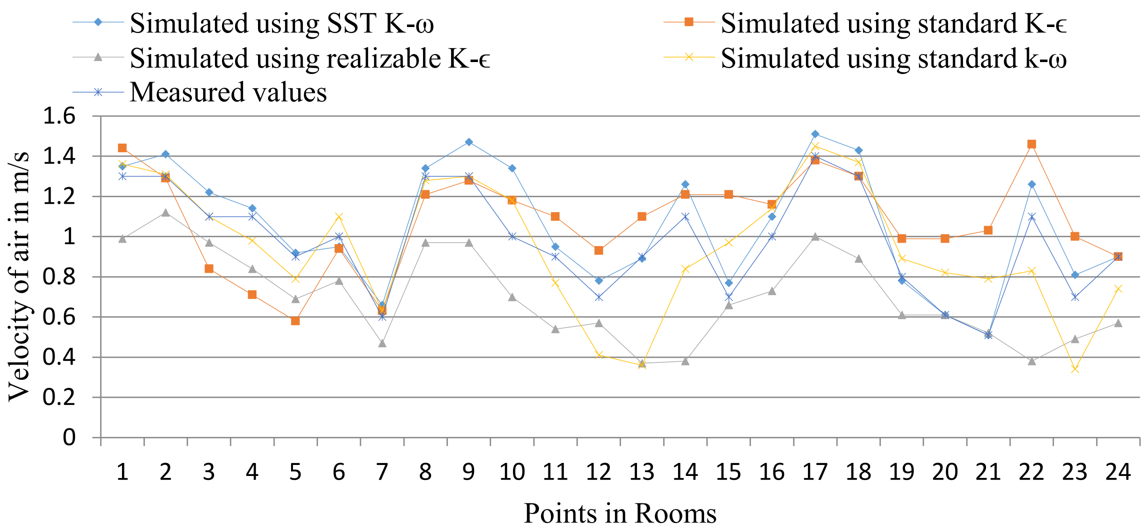


Figure 7 Comparison of the measured values of velocity with simulated values in different models at 0.15 m of height from floor

CONCLUSION

This paper presents a transient 3D Computational Fluid Dynamics based model of a roof fan. The exploration question was whether a straight forward model can be utilized for studies related to thermal comfort. Keeping in mind the end goal, to validate the model, a comparison has been made between the simulated and the measured values. The results clarified that the model which has been developed predicts the accurate behavior of the air velocity. When SST K-ω model has been used it has been observed that there was average error of only 6 % in the simulated model from measured one which is quite impressive.

So by the study we got to know that SST k-ω gives the most reliable and accurate results than the other models. By performing this simulation, it is easy for us to predict the location of fan in order to minimize the stratified zones and enhance the thermal comfort in the room or any closed area.

This study also leads us to one more finding that if we used the optimized speed then it will lead to reduction in power consumption. Further these results can also be utilized for attaining thermally comfortable condition inside building envelope such as in the case of rented offices or apartment where the occupant has not much to change rather than the furniture.

REFERENCES

[1] Anand, Y., Gupta, A., Maini, A., Gupta, A., Sharma, A., Khajuria, A., Gupta, S., Sharma, S., Anand, S. and Tyagi, S.K., 2014. Comparative thermal analysis of different cool roof materials for minimizing building energy consumption. Journal of Engineering, 2014.

[2] Babich, F., Cook, M., Loveday, D., Rawal, R. and Shukla, Y., 2017. Transient three-dimensional CFD modelling of ceiling fans. Building and Environment, 123, pp.37-49.

[3] Bassiouny, R. and Korah, N.S., 2011. Studying the features of air flow induced by a room ceiling-fan. Energy and Buildings, 43(8), pp.1913-1918.

[4] Ceiling Fans, 2013, American Lighting Association, http://www.americanlightingassoc.com/About-ALA/Press-Releases/Featured-Article---Ceiling-Fans-(1).aspx (Accessed on 7th February, 2018)

[5] Chen, W., Liu, S., Gao, Y., Zhang, H., Arens, E., Zhao, L. and Liu, J., 2018. Experimental and numerical investigations of indoor air movement distribution with an office ceiling fan. Building and Environment, 130, pp.14-26.

[6] Gruyters, W., Verboven, P., Delele, M., Gwanpua, S.G., Schenk, A. and Nicolaï, B., 2018. A numerical evaluation of adaptive on-off cooling strategies for energy savings during

long-term storage of apples. International Journal of Refrigeration, 85, pp.431-440.

[7] Jain, A., Upadhyay, R.R., Chandra, S., Saini, M. and Kale, S., 2004, January. Experimental investigation of the flow field of a ceiling fan. In ASME 2004 Heat Transfer/Fluids Engineering Summer Conference (pp. 93-99). American Society of Mechanical Engineers.

[8] Król, A. and Król, M., 2018. Study on numerical modeling of jet fans. Tunnelling and Underground Space Technology, 73, pp.222-235.

[9] Li, W., 2016. Numerical and Experimental Study of Thermal Stratification in Large Warehouses (Doctoral dissertation, Concordia University).

[10] S.C. Sekhar, Higher space temperatures and better thermal comfort—a tropical analysis. Energy and Buildings. 1995 Oct 1;23(1):63-70.

[11] The Hawaii Energy Code, 2015 https://energy.hawaii.gov/wpcontent/uploads/2016/07/Ceiling-FansFactsheet_final.pdf, (Accessed on 7th February, 2018).

[12] Tian, W., Han, X., Zuo, W. and Sohn, M.D., 2018. Building Energy Simulation Coupled with CFD for Indoor Environment: A Critical Review and Recent Applications. Energy and Buildings.

[13] Verma, S.K., Anand, Y and Anand, S. "CFD Based Modelling of a Ceiling Fan in a Room." International Journal of Scientific and Technical Advancements , vol. 4, no. 1,2, 19 Feb. 2018, pp. 219–224., www.ijsta.com/papers/NCEEITET-2018/IJSTA-V4N2R58Y18.pdf.

[14] Rohles Jr, F. H., Konz, S. A., & Jones, B. W. (1982, October). Enhancing thermal comfort with ceiling fans. In Proceedings of the Human Factors Society Annual Meeting (Vol. 26, No. 2, pp. 118-120). Sage CA: Los Angeles, CA: SAGE Publications.

[15] Rohles, F. H., Konz, S. A., & Jones, B. W. (1983). eiIin Fans Extenders of the Summer Comfort Envelope. ASHRAE Trans, 89, 245-263.

[16] Morton-Gibson, M; Coutier,P.J.; Place,W.J. Effects of fan velocity on thermalcomfort in an office building, Proceedings of the 10th National Passive SolarConference, the Solar Energy & Utilities Conference, the DaylightingApplications Conference, the Building with the Sun Conference (1985)406–409.