Thermal comfort analysis of a naturally ventilated regular size room equipped with ceiling fan

A Maheshwar¹, L Ramesh Krishnan¹ and L Gangadhara Kiran Kumar¹,*

¹Mechanical Engineering Department, National Institute of Technology Calicut, Kerala 673601, India

*Email: ganga@nitc.ac.in

Abstract. Residential and commercial buildings in India being the bigger contributors to energy consumption possess large energy-saving potential. There is a significant increase in the use of air conditioning to achieve thermal comfort, thereby increasing the total energy consumption and related CO₂ emissions. Ceiling fans are common in almost every habitable space of Indian residences, can be used as a hybrid ventilation system. A ceiling fan having exceptionally low energy consumption can be used to achieve thermal comfort thereby limiting the use of air conditioners. In this study, thermal comfort analysis was carried out on a naturally ventilated regular size room equipped with ceiling fan. The room is 3.96m x 3.96m x 3.05m in size. Ceiling fan inside the room generates the air flow and thermal fields. 166 rpm, 215 rpm and 250 rpm are the three fan speeds selected for this study. Both Computational Fluid Dynamics (CFD) simulation and experimental analysis was conducted. By analysing both numerical and experimental results, 215 rpm fan speed is selected as an optimum speed to achieve thermal comfort in regular size room. This proves fan can be used as a stand-alone device to achieve thermal comfort, where the indoor temperature is maintained at 28°C.

1. Introduction
Residential and commercial buildings contribute 34% of energy use globally, leading to 15% of overall energy-related CO₂ emissions. In India, thermal comfort accounts for 40% to 60 percent of annual electricity consumption. [1]. Thermal comfort is an important factor that affects one’s wellbeing, health, and productivity. The use of air conditioning to achieve thermal comfort has increased significantly, resulting in higher overall energy consumption and CO₂ emissions. Intent on thermal comfort, they possess large energy saving potential. The ceiling fans are common in almost every habitable space of both earlier and more recent buildings in India. Ceiling fans, in combination with natural ventilation, may be used to create a hybrid ventilation system that circulates better air movement from the floor to the ceiling or vice versa, offering adequate thermal comfort at high air speeds without the use of air conditioning. Ceiling fan having exceptionally low energy consumption, limits the usage of air conditioners [2]. Bala et al. [3] studied the direction of airflow in a room to find the distribution of air by changing various factors like blade angle, number of blades and fan speed and their effects on the uniform distribution of air inside the room. Alizadeh et al. [4] carried out a CFD simulation to investigate the thermal comfort level inside the room having a ceiling fan installed. Francesco et al. [5] validated CFD simulation results with the measured experimental data. The model developed by him is able to accurately predict the airflow generated by ceiling fan in a room. Wenhua et al. [6] validated the CFD simulation with the experimental results. CFD simulation is done using k-ε turbulence model. They concluded that the ceiling height and rotational speed have a greater impact.
on mean air velocity and air circulation of occupant zone than the fan's distance from the ceiling and blade geometry. Hsin et al. [7] investigated the impact of air distribution on thermal comfort in a room with a ceiling fan using a detailed human body model. From the literature review, it is clear that ceiling fans can provide thermal comfort in some conditions. In addition, before deciding on the need for other types of cooling equipment, a detailed study of thermal comfort analysis of spaces with ceiling fans is essential.

Current research aims to perform thermal comfort analyses in a regular size room having a ceiling fan. Both CFD simulation and experimentation are conducted for the proposed room. Three fan speeds, 166 rpm, 215 rpm and 250 rpm are selected for the study to carry out both the CFD simulation and experimental study. Thermal comfort conditions are analysed based on the results obtained from the current study.

2. Methodology
Air movement caused by a ceiling fan is measured at three different fan speeds 166 rpm, 215 rpm and 250 rpm in the experimental facility are collected in detail. The same experimental facility has been modelled using a 3D modelling software, creo parametric 4.0. This model is imported into CFD software, ANSYS Fluent for further analysis.

2.1. Experimental facility
For this study, an experimental facility of size 3.96 m (length) × 3.96 m (width) × 3.05 m (height) is used. This facility is present inside the fluid mechanics laboratory that belongs to the National Institute of Technology Calicut, Kerala. This room has two windows and one door, located on the ground floor. Only one wall is exposed to the outside atmosphere. The wall exposed to the outside environment have good shading that protects it from direct solar radiation. Inside this room a ceiling fan, desk, monitor, CPU, tube light and an occupant are present.

2.2. Fan design
It is a typical fan having three blades with a sweep size of 1.3 m controlled with a 4-step regulator. The ceiling fan is located in the room's centre. The fan is 0.30 m away from the ceiling, defined by the ANSI/IES Standard 55 (2013). The length of the blade is 0.5 m. Leading edge, that encounters the air first is angled by 2° and the other edge, trailing edge is angled by 8° [8].

2.3. Experimental conditions
The experiment takes place in Kerala during the winter season. At the time of the experimentation the outdoor temperature range was 26°C to 27°C and relative humidity varied between 61% and 66%. The average indoor temperature maintained in the experimental facility is 28°C. Because of the internal heat generation, there is an increase in indoor temperature. Two windows are kept open for this study. Open windows and a ceiling fan can provide ventilation to the space, allowing for better indoor conditions.
thermal environment. The experiment is repeated for three different fan speeds 166 rpm, 215 rpm and 250 rpm with a similar outdoor condition.

2.4. Measurement points
Velocity and temperature readings are taken at 4 positions. Out of 4 positions, two positions were taken along the perimeter of the fan, one position along the mid of the fan and one position at 0.5 m away from the perimeter of the fan. In each position, readings are taken at 9 different heights (0.1m, 0.35m, 0.6m, 0.85m, 1.1m, 1.4m, 1.7m, 2m and 2.3m). Readings are recorded at 9 heights for 4 different positions, this results in 36 locations. In each location, readings are taken for 2 minutes and the data is logged for every 2 seconds. Thermal anemometers, testo 405i and testo 480, with 0.02 m/s and 0.5 °C accuracy, were used for measuring instruments.

2.5. Numerical simulation
3D model of the experimental facility is created using CREO parametric 4.0 software, as depicted in figure 1-4. This model is imported into ANSYS fluent 2019 R3 for further CFD analysis. The three-dimensional model is divided into two parts: a revolving reference frame around the ceiling fan and the rest of the model. This rotating reference frame has the same diameter as the experiment facility’s fan. The rotating reference frame is cylindrical with 1.308 m diameter and 0.08 m height which is positioned 0.3 m from the ceiling. The size of the rotating reference frame should be minimum, to accommodate the fan model to reduce the simulation error [9].

![3D model of the experimental room](image1.png)

**Figure 3.** 3D model of the experimental room

![Rotating body](image2.png)

**Figure 4.** Rotating body

3D model of experimental facility includes two windows, monitor, CPU, desk, door, tube light and human occupant. The human occupant structure is modelled to have a surface area of 1.8 m².
(ASHRAE standard 2013). The 3D model is modelled as a solid part to represent the fluid region and all the objects including occupant inside the model have been extruded out of the solid body. In figure 4, the fan model has been extruded out of the rotating reference frame. The rotating frame motion option is selected for the rotating body and respective rpm is given for simulation. Heat flux of 65 W/m² is given for the occupant as per the ASHRAE 62.1 standards. The modelling and boundary conditions applied for CFD simulation are specified below in table 1 and 2.

| Parameter                        | Model/Significance                        |
|----------------------------------|-------------------------------------------|
| Type of solver                   | Steady, Pressure based solver             |
| Turbulent Model                  | Realizable K-ε Model                      |
| Pressure-velocity Coupling       | Simple algorithm                          |

| Object                          | Boundary type                             |
|---------------------------------|-------------------------------------------|
| Wall – rotating body            | Frame motion, Rotation                    |
| Wall – stationary envelope      | Frame motion, Rotation – 0 rpm            |
| Occupant (human model)          | Constant heat flux - 65 W/m²              |
| light                           | Heat flux - 10 W/m²                       |
| CPU                             | Heat flux - 30 W/m²                       |
| Monitor                         | Heat flux - 5 W/m²                        |
| Walls, Door                     | Constant wall temperature - 28°C          |
| Ceiling                         | Constant wall temperature - 28°C          |
| Windows                         | Pressure – outlet                         |
| Floor                           | Constant temperature - 28°C               |
| Table                           | Wall, Zero heat flux                      |

2.5.1 Turbulence model
For viscous computation, a realisable k-model was used in this study. The turbulent viscosity formulation is new, and the transport equation for the dissipation rate is new, so this model varies from the standard k-ε in two ways. The realisable k-ε model has the advantage of providing better results for flows involving boundary layers, rotation, recirculation, and separation when there are high adverse pressure gradients. For rotating regions, k-ε realisable is a significant member of the turbulence models.
The realizable k-ε transport equations for k and ε are given as:
\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k
\]
(1)
and
\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon \rho}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_3 G_b + S_\varepsilon
\]
(2)

Where
\[
C_1 = \max \left[ 0.43, \frac{\eta}{(\eta + 5)} \right], \quad \eta = S \frac{k}{\varepsilon} \text{ and } S = \sqrt{2 S_{ij} S_{ij}}
\]

\(G_k\) and \(G_b\) denote the production of turbulence kinetic energy as a result of mean velocity gradients and buoyancy in the equations above. The fluctuating dilatation in compressible turbulence contributes to the overall dissipation rate, and \(Y_M\) reflects this contribution. \(C_2\) and \(C_{1\varepsilon}\) are constants. \(\sigma_k\) and \(\sigma_\varepsilon\) are the turbulent Prandtl numbers for \(k\) and \(\varepsilon\), respectively. \(S_k\) and \(S_\varepsilon\) are user-defined source terms.

Eddy viscosity \(\mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon}\)

\(C_{\mu}\) is constant for the standard \(k - \varepsilon\) model, for the realizable \(k - \varepsilon\) model it is calculated from.
\[
C_{\mu} = \frac{1}{A_0 + A_5 \frac{k^2}{\varepsilon}}
\]
where
\[
U^* = \sqrt{S_{ij} S_{ij} + \tilde{\Omega}_{ij} \tilde{\Omega}_{ij}}
\]
(4)
and
\[
\tilde{\Omega}_{ij} = \Omega_{ij} - 2 \varepsilon_{ijk} \omega_k
\]
\[
\Omega_{ij} = \tilde{\Omega}_{ij} - \varepsilon_{ijk} \omega_k
\]
(5)

The mean rate-of-rotation tensor \(\tilde{\Omega}_{ij}\) is interpreted in a rotating reference frame with the angular velocity \(\omega_k\). The model constants are \(A_0\) and \(A_5\).

2.5.2 Grid independent study

The 3D model consists of two bodies, a rotating body (rotating reference frame) and a stationary body (room). These two bodies are merged into a single part model. Figure 5 shows how hex-dominant cutcell meshing was used to discretize the computational domain. This cutcell meshing produces a better result than the regular tetrahedral meshing and saves computational time. Grid independence test was conducted for four mesh sizes (1.9 million, 2.3 million, 2.6 million and 2.9 million). The mesh with 2.6 million grids did not have any significant difference in the test results when compared with 2.9 million grids as shown in figure 6. As a result, the medium grid number of 2.6 million is used for...
all simulations in this paper to minimise computational time without conceding result quality. Grid skewness is 0.01 with a cell size growth rate of 1.1 and an aspect ratio of 1.07.

![Hex-dominant meshing in both stationary body and rotating body](image)

**Figure 5.** Hex-dominant meshing in both stationary body and rotating body

![Grid independent study](image)

**Figure 6.** Grid independent study

2.6. *Thermal comfort – ASHRAE 55*

Thermal comfort at elevated airspeed - ASHRAE standard 55 (2013) is divided into two categories, with and without occupant control. In the case of occupant control, there is no higher bound on airspeed, while in the case of without occupant control; the upper bound is set at 0.8 m/s. [9]. These conditions are only applicable when the operative temperature is above 25.50°C, implying that if the operative temperature is below 25.50°C, there is a risk of draught discomfort. For instance, ASHRAE 55 predict the comfort condition for the elevated air velocity is 0.8 m/s and operative temperature is between 26.9°C to 30.2°C. Figure 7 depicts the ASHRAE comfort chart based on the clothing value of 0.5 clo (trousers, short-sleeve shirt) and metabolic rate of 1.1 Met (office activity and typing). By
observing this temperature range, there is a potential in achieving thermal comfort for our test conditions.

Operative temperature, \( t_o = \left( t_{mr} + (t_a \times \sqrt{10v}) \right) \left( 1 + \sqrt{10v} \right)^{-1} \) (6)

Where,

\( t_a = \) air temperature (°C), \( t_{mr} = \) mean radiant temperature (°C) and \( v = \) airspeed (m/s)

![Figure 7. Thermal comfort at elevated airstreams for warmer air temperatures][9]

3. Result and Discussion
The experiment was conducted for three different speeds. Temperature and velocity values were recorded from 36 different locations. In the numerical analysis, the velocity and temperature values were exported from 120 different points in the 3D computational model using the post-processing tool of ANSYS Fluent. Four vertical lines of a height of 2.3 m were drawn under the fan at the 4 selected positions. These four positions are mentioned above in the experimental investigation. On each line 30 heights were considered. 30 heights on the vertical lines in four position results in 120 locations. Temperature and air velocity values were taken from 120 locations. The operational temperature was calculated using the above-mentioned formulae. From the obtained operating temperature and air velocity values, the points were plotted on the elevated airstream thermal comfort chart. Based on the position of points on the thermal comfort chart, thermal comfort levels were identified. The results obtained are discussed in detail below.

3.1. Experimental results
The below figures shows the experimental results that were plotted on the graphs respective to the three different fan speed 166 rpm, 215 rpm and 250 rpm cases. Each point on the graphs denotes a location where the readings are recorded in the experimental setup. There are 36 points on each graph. The LEED compliance test was also conducted to cross check the obtained results. In the LEED compliance test PMV and PPD values are calculated using CBE thermal comfort tool based on ASHRAE-55. On counting the number of points on the shaded region of the graph and points that are accepted by the LEED compliance test, an optimum fan speed was selected.
Figure 8. Thermal comfort at elevated airspeeds, experimental investigation (166 rpm)

Figure 9. Thermal comfort at elevated airspeeds, experimental investigation (215 rpm)

Figure 10. Thermal comfort at elevated airspeeds, experimental investigation (250 rpm)
In figure 8, the majority of the points are outside of the shaded area. Just a few points fall inside the accepted shaded zone, indicating a 36 percent acceptability score. Reasonable number of points fall within the shaded region, as shown in figure 10; it indicates 61% acceptability score. In figure 9, the shaded region has the highest number of points, indicating a higher acceptability rating of 83%. The LEED compliance results came up with similar findings. With the highest acceptability rate, 215 rpm fan speed provides thermal comfort at elevated air velocity. Most of the points in the 215-rpm case fall on the light-shaded region. Light shaped region indicates that there is no need for local control. The velocity ranges between 0.2 m/s to 0.8 m/s and the operative temperature ranges between 27°C to 28°C.

3.2. Numerical analysis result

In the below figures, results extracted from the ANSYS post-processing tool are plotted. Each point in the graph represents a selected location in the 3D computational domain, from where the temperature and velocity readings were extracted. The readings extracted from the numerical analysis results for 166 rpm fan speed are plotted in figure 11. Similarly, 215 rpm fan speed results are plotted in figure 12 and the 250 rpm fan speed results are shown in figure 13. The temperature and velocity readings extracted from 120 locations are depicted on each figure. Since the numerical analysis had results of every location, it was easy extract large amount of data using the ANSYS post processing tool. It's difficult to keep track of temperature and velocity values in a variety of locations during an experiment. Hence, the data obtained from the experimentation is less when compared to numerical work. Since the results are taken from 30 heights in 4 positions, location of the points are closer to each other making them to overlap in case of numerical analysis.

Figure 11. Thermal comfort at elevated airs speeds, numerical analysis (166 rpm)
A suitable fan speed for achieving optimum thermal comfort at elevated airspeeds was determined by counting the number of points on the shaded region and the number of points approved by the LEED compliance test. By studying all three graphs, 215 rpm have many points falling into the shaded region with a LEED compliance of 91%. The LEED compliance is 67% for 166 rpm and 78% for 250-rpm conditions. 215 rpm was found to be the most appropriate for maintaining thermal comfort in all three-test conditions with the highest compliance rate among the three. Most of the points fall in the lightly shaded region in the case of 215 rpm, denoting there was no need for local control.

4. Conclusion
It was evident from the findings of an experimental and numerical study that ceiling fans can be used as a system to achieve thermal comfort by maintaining high airspeed. For this analysis, fan speeds of 166 rpm, 215 rpm, and 250 rpm were considered and the fan speed of 215 rpm was chosen as the optimum speed for achieving thermal comfort in the current study, after analysing both numerical and experimental data. For this study, it is clear that a regular size room having an average indoor temperature of 28°C and outdoor temperatures varying from 26°C to 27°C can be kept thermally comfortable by using a ceiling fan in natural ventilation with windows open. Since airflow is a major factor influencing thermal comfort for the above stated conditions, maintaining air velocity between 0.2 and 0.8 m/s helped in achieving thermal comfort even at 28°C without the use of an air conditioner.
References

[1] Bureau of energy efficiency, the government of India, ministry of power, https://beeindia.gov.in, 2021.

[2] He Y, Chen W, Wang Z and Zhang H 2019 Review of fan-use rates in field studies and their effects on thermal comfort, energy conservation, and human productivity Energy Build. 194, 140–62

[3] Rao J B B, Rao D B, Yadav J S and Sreerama M 2020 Simulation of Air Flow Around Ceiling Fan In An Enclosed Space By Using Cfd 9 1061–4

[4] Alizadeh M and Sadrameli S M 2018 Numerical modeling and optimization of thermal comfort in building: Central composite design and CFD simulation Energy Build. 164 187–202

[5] Babich F, Cook M, Loveday D, Rawal R and Shukla Y 2017 Transient three-dimensional CFD modelling of ceiling fans Build. Environ. 123, 37–49

[6] 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 Build. Environ. 130, 14–26

[7] Lin H H 2019 Improvement of human thermal comfort by optimizing the airflow induced by a ceiling fan. Sustain. 11

[8] Zadravec M, Basic S and Hribersek M 2007 The influence of rotating domain size in a rotating frame of reference approach for simulation of rotating impeller in a mixing vessel J Eng Sci Technol 2, 126–38

[9] ANSI/ASHRAE, Standard 55-2013: Thermal Environmental Conditions for Human Occupancy, American Society of Heating, Refrigerating and Air-Conditioning Engineering, Atlanta, GA, 2013.