International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 5 Issue 4, May-June 2021 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

A Review of Case Study of Room Comfort Analysis if Fan & Exhaust Fan Being Used

Jitendra Kumar, Sunil Kumar Chaturvedi, Abhishek Bhandari

Department of Mechanical Engineering, NRI Institute of Research and Technical, Bhopal, Madhya Pradesh, India

ABSTRACT

Now a days with the changing of climatic condition and rise of temperature globally becoming a challenging phase to human survival and achieving an human comfort at lesser cost is also challenging where world are facing energy crisis also. Comfort related to making up of room air which includes treatment of air temperature by mixing it, humidity and air velocity. Installation of air conditioning or use of such devices are costlier and not possible to have everyone while other devices like fans and exhaust fan are sufficient to make up room air by analyzing thermal study and flow pattern of inlet and outgoing air at cheaper cost. In this work already exiting designed room space is analyzing by considering variables like room size, air flow pattern, wind velocity, room heat load of electric equipment, Almeria and stationary item are included. In this work analysis is done to evaluate the heat transfer co-efficient of room air and target is to achieve human comfort by proper selection of ventilated air location and air flow direction. With the development of Numerical simulations using a Computational Fluid Dynamics (CFD) technique it is possible to investigate thermal comfort and indoor air quality for room space using Ansys for Fluid Fluent. Since the CFD used approximations, it was necessary to validate the CFD program before it was used as a tool of study. Actually this is a case study of HOD cabin during electric load (Energy Audit) evaluation of entire academic infrastructure at NRI college and intention was to make the entire mechanical engineering department floor comfort space at cheaper cost. It is also a part to validate the urgent requirement of A/C to our institutional management to provide given quotation based on evaluated data.

***KEYWORDS**: Room Comfort, Ceiling Fan, Exhaust Fan, CFD, Turbulent flow, Enthalpy*

***How to cite this paper**:* Jitendra Kumar | Sunil Kumar Chaturvedi | Abhishek Bhandari "A Review of Case Study of Room Comfort Analysis if Fan & Exhaust Fan Being Used" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-5 | Issue-4, June 2021, pp.1573-1576, URL: www.ijtsrd.com/papers/ijtsrd43649.pdf

IJTSRD43649

I. INTRODUCTION

In India the most common appliances used to achieve cooling comfort during summer is fan, because of it's cost and cooling capacity. In this paper a CFD analysis based case study of mechanical engineering department, NIRT Bhopal room cooling comfort in summer, applying ceiling fan and exhaust fan assuming room temperature at 303 K. In this paper a exhaust fan used to develop a draught to generate a comfort cooling condition in summer. The benefit of this analysis is to generate valuable data to make it more efficient and reduce electricity using this analysis. This report will help to define orientation of ceiling fan and exhaust fan. The goal of this paper is to make the room more comfortable using less electricity or less power consumption to achieve it. Software for computational analysis such as Ansys fluent is a useful tool that can be used to create a virtual model of the building interior and simulate air flow, temperature profile and humidity which are directly related to thermal comfort, before the actual construction can be done. Modifications to an existing building can also be simulated using the CFD met basics to any physical renovations.

CEILING FAN: - Ceiling fans is a mechanical device, which is mostly driven with electrical supply. It create sufficient thermal cooling comfort to cool people effectively by introducing slow movement. Basic principle of Fans are that, it never actually cool air, unlike air-conditioning equipment, they in fact heat the air due and create low pressure volume of air, so that cold air can fill that place to maintain it normal. Similarly, a ceiling fan can also be used to reduce the stratification of warm air in a room by forcing it down to affect both occupants' sensations and thermostat readings, thereby improving climate control energy efficiency.

EXHAUST FAN:- Exhaust Fan are a mechanical device which is basically driven with electrical supply. Its RPM is slightly higher than ceiling fan, the basic use of ceiling fan is, it drags out unpleasant, smoky and bad odors, moisture in air space and the warm air that's drawn out using an exhaust fan is then pulled through a either with ducting system or just in open are and expelled outside.

OBJECTIVE: Comfort is a requirement of human body feeling good while livelihood is doing whatever (i.e. during working and living etc.), but here comfort related to making up of room air including temperature, humidity and air velocity. The main objective of this project helps in providing an alternative feasible solution in the real existing problem in the design of ventilated air conditioning system with the general available facility in home that is by using ceiling fan and exhaust fan and their location in room. The transfer of heat to and from process fluids is an essential part of making comfort air. It is also intending to study and analyze the heat transfer coefficient and pressure drops for different mass flow rates and inlet and outlet temperatures.

Main Objectives are:
1. To design and analyze the room ventilation system.
2. To make the room comfort using fan and exhaust fan.
3. To analyze the heat transfer coefficient of comfort room air.
4. To analyze the existing room comfort zone by varying mass flow rate of air, velocity of flow and air circulation pattern of room air.

II. LITERATURE REVIEW

From various open sources search of the literature published from 1970 to 2017 was performed. The following search terms were used: “ventilation”, “ventilation efficiency”, “ventilation effectiveness”“air flow distribution”, “air change rate”, “mixing ventilation”, “displacement ventilation”, “personalized ventilation”, “exhaust ventilation”, “piston flow,“ protective ventilation”, “stratum ventilation” and “hybrid ventilation”. One or more of the following search terms were used in every combination: “effectiveness”, “efficiency”and “air flow distribution”. Following this met, interesting articles were discovered in the reference lists, based on their titles and abstracts with emphasis on ventilation effectiveness and ventilation efficiency and the most interesting papers were selected for more detailed study. Excluding co-authors, a few frequently-cited authors were added as search terms (Melikov, A.; Nielsen, P.; Seppänen, O.; Karimipanah, T.; Niu, J.L. and Olesen, B. et al.). In addition, a manual search was conducted in Google for guidelines (given by ISO, ANSI/ASHRAE, CDC and WHO) and relevant sections of books, bibliographies and monographs. To find more relevant studies and sections of books, excluding co-authors, other well-known experts in this field were also personally contacted in the study (including: Clement D. J.; Melikov, A.; Nielsen, P.; Seppänen, O.; Olesen, B.; Mundt E.; Li, Y.G.; Heiselberg, P.; Müller, D.; and Koskela, H.). Finally, the proceedings of a series of international conferences in this field were searched, including Roomvent 1998, Clima 2000, Indoor Air 2002.

Bradley S. Hurak had research for a computational fluid dynamic analysis was performed to investigate the effect of the physical configuration of inlet and outlet vents on the temperature and flow patterns inside a room modeled for simplicity as a two dimensional enclosure. It was determined that for use in both heating and cooling of a room, a low or floor located inlet vent coupled with an outlet that is positioned on the upper half of a wall yields the most desirable results in reaching, or nearly reaching, comfort conditions in the shortest amount of time. However, if either heating or cooling is expected to be the primary energy consumption, it may be advantageous to deviate from this configuration.

Peter V. Nielsen analyzes room air distribution and describes a design chart that makes it possible to evaluate variables such as air quality, air speed and draft rate (DR), temperature gradient (PD_{grad}), mean radiant temperature, and radiant temperature asymmetry (PD_{rad}) as a function of flow rate and temperature difference in the supply system. Five different air distribution systems are analyzed: mixing ventilation from a wall-mounted diffuser, mixing ventilation from a ceiling-mounted diffuser, mixing ventilation from a ceiling-mounted diffuser with swirling flow, displacement ventilation from a wall-mounted low-velocity diffuser, and a low-impulse system based on a textile terminal in the ceiling. The procedure to some extent can be based on flow elements, but especially full-scale experiments and computational fluid dynamics predictions have been used for the development of the design charts described herein.

E. Adeeb, A. Maqsood, A. Musthaq and C. H. Sohn studied in paper includes parametric study and optimization of non-linear ceiling fan blades by combining the techniques of Design of Experiments (DOE), Response Surface Mets (RSM) and Computational Fluid Dynamics (CFD). Specifically, the nonlinear (elliptical) planform shape of ceiling fan blade is investigated in conjunction with blade tip width, root and tip angle of attack. Sixteen cases are designed for three blade ceiling fan using two level full factorial model. The flow field is modeled using Reynolds-Averaged-Navier-Stokes approach. The performance variables used to formulate a multi-objective optimization problem are volumetric flow rate, torque and energy efficiency. Response Surface Met is used to generate the optimized design for non-linear ceiling fan blade profile. The results reveal that the interactions between the design variables play a significant role in determining the performance. It is concluded that the nonlinear forward sweep has a moderate effect on response parameters.

CFD has gained importance and emerged as a useful tool for the prediction of air movement in ventilated spaces (Nielsen, 2015). CFD numerically solves the conservation of mass and momentum (Navier-Stokes equations) for the isothermal flow conditions. It solves the energy transport equation when a heat source is involved, which demands calculation of the temperature distribution in the flow domain. Most often CFD studies involving indoor air also solve the species transport equation to determine contaminant distribution. Indoor airflows are often turbulent, and thus require a turbulence model to be solved simultaneously. The solutions of these equations give the distributions of air velocity, temperature, pressure and contaminants present in the computed space.

Posner et al. (2003) studied the indoor airflow in a model room and showed that obstacles in the room can have a significant effect on the airflow motion in the room. Khan et al. (2006) did CFD simulations to study the effects of inlet and exhaust locations and gas densities on contaminant concentrations.

Bulińska et al. (2014) performed CFD simulations in a room with one sleeping person to find the measuring area of mean CO2 concentration. Rojas et al. (2015) studied the mixing of living room air and bedroom air while using supply air nozzles only in bedrooms. Gilani et al. (2016) did sensitivity analysis for CFD simulations of a test room using different grid sizes, turbulence models, discretization schemes and convergence criteria. Ning et al. (2016) used CFD to simulate airflow field, mean age of air and CO2 distributions inside a bedroom using different heights of conditioned air supply outlet. Thus, several papers are available in the literature showing different applications of CFD modeling in indoor environments.

The search terms for this literature review are relatively broad, so articles focusing on very specific and detailed areas of contaminant transmission and airflow patterns may not have been included. All references identified in this study were initially selected on the basis of their title and/or abstracts, with the emphasis on the ceiling fan, ventilation effectiveness and efficiency. This study classify and summarizes known ventilations systems and ceiling fan that have been used for various building types into different mets based on the concentration distribution, the location of the air supply/exhaust device and the use of natural and mechanical forces.

III. PROBLEM FINDING

The following are the analysis points being observed and recorded during study of literature.

1. The validation was done by comparing the CFD results with experimental data obtained in room with air flow and ventilation inside the room. The experimental measurements were carried out to obtain the flow characteristics in the room with energy equation along with viscous laminar equations.
2. Measurement procedure of Room consists of some electrical equipment which considered as wall in the process of analysis.
3. Measurement procedure of Room consists of some electrical equipment which considered as wall in the process of analysis.
4. The dimensions of the room are 4.30 m long, 3.50m high and 3.96 m wide.
5. The thermal condition of room was used to simulate summer conditions at 30 degree C.
6. The heat rate per body surface area is 115 W/ m2. The room contains four occupants with a total heat rate of 200 W,
7. Four equipment boxes each with 100 W and six ceiling lamps of 64 W. The total cooling load is very high (90W/m2).
8. The measurements for air velocity, air temperature, and SF6 concentrations were conducted in nine removable poles installed in the rooms.
9. Five thermocouples were used to measure the temperatures of the walls, ceiling, window, and floor, as well as the supply and exhaust air to define the initial room environment, to improve current condition of room.

IV. METHODOLOGY

1. The model has been generating over the workbench of CATIA and Ansys student version. With the realizable k-ε model, which was considered more functional than the standard k-ε model for simulating rotational shear flow of air, was combined with a multi layer approach using the room model. This combination was adopted together with the implicit simple algorithm. The finite volume met was applied with the second-order upwind scheme for discrediting the governing equations of velocity components and pressure, and the first-order upwind scheme for temperature and passive scalar. For the near-wall region, the all-wall treatment was applied to emulate the high y^+ wall treatment with standard wall laws for y+>30, the low y+ wall treatment for fine meshes with y+ to be approximately 1 or less to give reasonable results for intermediate meshes where the cell centric fell in the buffer layer.
2. The rotating reference frame was applied only in the rotational region by assuming that the region was in a quasi-steady state. This met did not explicitly model rotation; instead, it generated a constant grid flux in the appropriate conservation equations by automatically adding the source terms with respect to the Coriolis force and Centrifugal force, which were calculated with an eq. based on the properties of the reference frame. Although this met underestimates the weak effect, it is appropriate for the flow, which is most likely to be influenced by time-averaged properties. A significant amount of simulation time can be saved with this met, when compared to simulating the ceiling fan's rotation in a transient state.

V. RESULT AND DISCUSSION

Result and Analysis:- for the completion of this work as thought developed to make the room space comfort on already existed constructed part of building and preparing report as suggestions to change ventilation system and effective utilization of already using electrical appliances to make up room space considering the turbulence flow behavior of air. To analyze the room comfort on CFD the following procedures required and have to be define the boundary condition as a variables as-

Step 1: Development of 3D cad model

Step 2: Generation of mesh importing cad model to Ansys workbench

Step 3: Method

- Selection of fluid as material
- Selection of viscous laminar equation
- For standard wall condition
- Define k-e model 2 equation
- Assuming temperature at 303 k
- Velocity of fan input assumed to 075-1.8 m/s^3

Energy equation on

Step 4: Define boundary:

During the model development, define boundary

Stationary:

- Walls
- Equipments
- All stationary as wall

Rotating:

- Ceiling fan blade
- Exhaust fan blade

VI. CONCLUSION

This study reveals to make space comfort it required somehow to use the things which are artificial consume powers, encountering energy loss, degradation of energy, ultimately operating cost are high and even at this cost results are global warming desires are high, it's a human tendency, we are trying to achieve one room space comfort whereas impacting to whole planet. In this work using CFD and experimental analysis will be conducted to exactly know the basis variable parameter and improvement variables parameter.

Based on study it has been observed that it is necessary to install an exhaust fan toward the open area as well to the workshop end so that a natural fluid flow can be easy to the room and cooling capacity will slightly increase automatically and hence the human comfort level can be boosted a little bit.

And as second choice, it is better to make an open window, just in front of the main door and its opposite wall, will help to reduce heating effect, because there will be some natural draught.

As a suggestion to the specific room space, it is better to install a split ac, by replacing both ceiling, because ceiling fan just, let human body feels like cool but in actually it warms up the space and likewise ceiling fan, exhaust fan helps to improve air quality to the space, it reduces humidity factor as well fresh air intake to the space environment

REFRENCE

[1] E. Adeeb, A. Maqsood, A. Musthaq and C. H. Sohn studied in paper includes parametric study and optimization of non-linear ceiling fan blades by combining the techniques of Design of Experiments (DOE), Response Surface Mets (RSM) and Computational Fluid Dynamics (CFD)

[2] Posner et al. (2003) studied the indoor airflow in a model room and showed that obstacles in the room can have a significant effect on the airflow motion in the room.

[3] Khan et al. (2006) did CFD simulations to study the effects of inlet and exhaust locations and gas densities on contaminant concentrations.

[4] Bulińska et al. (2014) performed CFD simulations in a room with one sleeping person to find the measuring area of mean CO2 concentration.

[5] Rojas et al. (2015) studied the mixing of living room air and bedroom air while using supply air nozzles only in bedrooms.

[6] Gilani et al. (2016) did sensitivity analysis for CFD simulations of a test room using different grid sizes, turbulence models, discretization schemes and convergence criteria.

[7] Bradley S. Hurak (2016) Computational Fluid Dynamics Analysis of Air Flow and Temperature Distribution in Buildings

[8] Ning et al. (2016) used CFD to simulate airflow field, mean age of air and CO2 distributions inside a bedroom using different heights of conditioned air supply outlet.

[9] Bhortake, R. V., P. S. Lachure, S. R. Godase, V. V. More and K. S. Chopade (2014). "Experimental Analysis of Air Delivery in Ceiling Fan. "

[10] Aziz, M. A., I. A. M. G. Shahat, F. A. Mohammed and R. H. Mohammed (2012). Experimental and numerical study of influence of air ceiling diffusers on room air flow characteristics. Energy and Buildings 55, 738–746.

[11] A. T. Balasbaneh, “Hot climate air flow study and effect of stack ventilation in residential building”, MSc Thesis, Faculty of Civil Engineering, UniversitiTeknologi Malaysia, Johor Bahru, Malaysia, 2010.

[12] Afaq, M. A., A. Maqsood, K. Parvez and A. Mushtaq (2014). Study on the design improvement of an indoor ceiling fan. 11th International Bhurban Conference on Applied Sciences and Technology, IEEE.

[13] Amano, R. S., E. K. Lee, C. Xu and J. Xie (2005). Investigation of the Unsteady Flow Generated by an Axial Fan: Experimental Testing and Simulations. International Journal of Rotating Machinery 3, 256-263

[14] Ankur, J., R. U. Rochan, C. Samarth, S. Manish and K. Sunil (2004). Experimental Investigation of the Flow Field of a Ceiling Fan. ASME 2004 Heat Transfer/Fluid Engineering Summer Conference, Charlotte, North Carolinia, USA.

[15] Azim, M. A. (2014). Velocity field analysis of a ceiling fan. Turkish Journal of Engineering, Science and Technology 01, 1-7.

[16] Aziz, M. A., I. A. M. G. Shahat, F. A. Mohammed and R. H. Mohammed (2012). Experimental and numerical study of influence of air ceiling diffusers on room air flow characteristics. Energy and Buildings 55, 738–746.

[17] Bhortake, R. V., P. S. Lachure, S. R. Godase, V. V. More and K. S. Chopade (2014). "Experimental Analysis of Air Delivery in Ceiling Fan."

[18] J Lau 1, * and Q Chen1 Ray W. Herrick Laboratories, School of Mechanical Engineering, Purdue University, 585 Purdue Mall, West Lafayette, IN 47907-2088, USA

[19] McDonell G. 2003. Underfloor and displacement: Why they're not the same, ASHRAE Journal, v 45, n 7, July, pp 18-24

[20] *Bardina, J. E., Huang, P. G., Coakley, T. J. (1997), “Turbulence Modeling Validation, Testing, and Development”, NASA Technical Memorandum 110446.*

[21] Peter V. Nielsen E. Adeeb, A. Maqsood, A. Musthaq and C. H. Sohn studied in paper includes parametric study and optimization of non-linear ceiling fan blades by combining the techniques of Design of Experiments (DOE),

[22] A. T. Balasbaneh, “Hot climate air flow study and effect of stack ventilation in residential building”, MSc Thesis, Faculty of Civil Engineering, UniversitiTeknologi Malaysia, Johor Bahru, Malaysia, 2010

[23] Robert Bruegmann. "Central Heating and Ventilation: Origins and Effects on Architectural Design"

[24] https://www. ansys. com/en-in/academic/free-student-products

[25] AbdAlnasserAlmate A. Ali, Ahmed CherifMegri use of computational fluid dynamics to predict airflow and temperature distribution in a residential building with an under flow air distribution.

[26] Chen, Q., 1995. Comparison of different k-ε models for indoor air flow computations. Numerical Heat Transfer, Part B, 28: 353-369. DOI: 10. 1080/10407799508928838

[27] Wang, X. and D. W. Pepper, 2007. Numerical simulation for under-floor air distribution system with swirl diffusers. J. Heat Transfer, 129: 589-594. DOI: 10. 1115/1. 2709974

[28] Janssen, John (September 1999). "The History of Ventilation and Temperature Control".