RESEARCHERID
THOMSON REUTERS
[Desai et al., 7(11), Nov 2022]

ISSN : 2455-9679
Impact Factor : 4.256

ijrtsm

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"CFD ANALYSIS OF A HOSPITAL PATIENT ISOLATION ROOM FOR EFFECTIVE UTILIZATION OF A POSITION OF AIR CONDITIONING"

***Desai Kausal Hemant* [1], *Rajneesh Kumar Gedam* [2]**
[1]*M. Tech Scholar, Department of Mechanical Engineering , RKDF College Bhopal, M.P, India*
[2]*Assistant Professor, Department of Mechanical Engineering , BERI Bhopal, M.P, India.*

ABSTRACT

In this paper, computational fluid dynamics (CFD) modeling of contaminant spread in isolation room with different ventilation rates and negative pressure differentials. The numerical results from CFD software Solid work. Here it also analyzed the temperature stratification in order to maintain better thermal conditioning inside the control system. Then after it also calculate the effect of the position of the diffuser inlet on the temperature stratification and also find the optimal position of the diffuser inlet in order to provide better mixing of air inside the system, so that better comfort conditioning can be achieved. Here we consider the three different cases that is diffuser inlet neat to the heat source, diffuser inlet at the center of the room and near to the free space area. It these three cases we find the center position is the optimum position. After obtaining the optimum position, here it also find the effect of different diffuser inlet design, here it also consider the three different design of diffuser inlet that is rectangular, circular and triangular diffuser inlet geometry. After analyzing the different geometry it is found that the circular diffuser inlet geometry is the optimum one because during this case the temperature difference in between the plane is less.

Keyword: *Solid work, temperature stratification, CFD, horizontal planes, diffuser inlet, HVAC.*

I. INTRODUCTION

Since 1990s, humans have already facing a serious shortage of resources. Saving energy has already become the focus in countries around the world. In energy consumption, buildings section is the main energy consumer, among which air conditioning system is up to 60%～70%. There-fore, to save energy, the prospect of energy efficiency of air conditioning is very broad. In addition, as people's indoor retention time increase gradually, indoor air quality is directly related to the personnel health and work efficiency. Sick building syndrome caused by the bad indoor ventilation has aroused wide attention of all social sectors, so the improvement of the indoor air quality (IAQ) has become one of the problems which should be solved in air conditioning system.

For the most buildings, the design standards of air conditioning system are still provide a single, homogenous thermal environment and ventilated environment for all areas of the building, which have little opportunity to meet each occupant on thermal environment needs and preferences. Although the traditional air conditioning system with upper air supply can satisfy the requirements of in-door temperature and humidity, it cannot provide better indoor air quality and consume more energy.

Under-floor air distribution system has been used more and more widely in office buildings because of its superiority of flexibility, energy saving, lower investment, improvement of comfort and health, and satisfaction with individual requirement of local thermal environment control, first, a review on the application and development of under-floor air condition system has been carried in this paper. Then an office room with under- floor air condition has been simulated to learn the indoor temperature field, velocity field and thermal comfort in the circumstances of under floor air distribution. Meanwhile, according to the thermal equilibrium and human physiological temperature regulation model, the heat dissipation of human body to environment in under floor air distribution room has been calculated using the thermal equilibrium equation of human body. And the analysis on human thermal comfort has been analyzed at the view point of human physiology. And the results have been compared with some thermal comfort standards, such as PMV. Through the simulation and analysis, when human feel the most comfortable in under floor air distribution room, parameters such as air temperature, air velocity and air supply volume have been obtained. This thesis aims to demonstrate how CFD technology can be used to under floor air distribution and overhead air distribution parameter like velocity, temperature, and different mole fraction relative etc, and compare them a practically.

Room air Distribution

Air is introduced to, flows through, and is removed from spaces is called **room air distribution.** HVAC airflow in spaces generally can be classified by two different types: mixing (or dilution) and displacement

Mixing Systems

Mixing systems generally supply air such that the supply air mixes with the room air so that the mixed air is at the room design temperature and humidity. In cooling mode, the cool supply air, typically around 55 °F (13 °C) (saturated) at design conditions, exits an outlet at high velocity. The high velocity supply air stream causes turbulence causing the room air to mix with the supply air. Because the entire room is near-fully mixed, temperature variations are small while the contaminant concentration is fairly uniform throughout the entire room. Diffusers are normally used as the air outlets to create the high velocity supply air stream. Most often, the air outlets and inlets are placed in the ceiling. Supply diffusers in the ceiling are fed by fan coil units in the ceiling void or by air handling units in a remote plant room. The fan coil or handling unit take in return air from the ceiling void and mix this with fresh air and cool, or heat it, as required to achieve the room design conditions. This arrangement is known as 'conventional room air distribution'

Displacement Ventilation

Displacement ventilation systems deliver air at floor level into the space at very low velocity, typically less than 50 feet per minute (fpm). At this velocity, the air coming out of the diffuser can barely be felt, and the fresh air “pools” onto the floor. The system produces two distinct zones of air, one characterized by stratified layers of relatively cool and fresh air, the other by fairly uniform hot and stale air. The vertical flow profile in the lower zone can be generally described as upward laminar flow, or “plug flow.” The effect of the plug flow is to displace the hot stale air into an area well above the breathing level of the occupants, giving occupants the benefit of breathing significantly higher-quality air. The displacement effect is augmented by the presence of heat sources within the occupied space.. The thermal plume created by a heat source has the effect of enhancing the airflow around the source, thereby improving overall heat removal. The plume is inherently advantageous in applications where air contaminants and heat are linked to the same source. Draft is usually not an issue in spaces served by displacement ventilation systems, but the temperature difference between the floor and head levels (the temperature gradient) is an important design issue.

Displacement ventilation systems are generally applied to spaces that require cooling during occupied hours. They do not function well as heating systems; when heating is required, it is generally supplied by a separate system. Conventional European design practice limits the use of displacement ventilation systems to spaces with peak cooling loads of 12 Btu/hr-ft2 or less. In spaces with higher cooling loads, radiant cooling systems are used in combination with displacement ventilation. Recent U.S. research suggests that displacement ventilation systems can be applied to spaces with cooling loads up to 38 Btu/hr-ft2.

Under Floor Air Distribution

Under floor air distribution systems are a general class of air distribution systems that deliver air through diffusers in the floor, with the intent of maintaining comfort and indoor air quality levels only in the occupied lower portion of space. These systems are increasingly popular alternatives to the traditional overhead, or “fully-mixed” systems, which attempt to condition the air in the whole volume of space. Underfloor systems provide unique opportunities for energy savings, enhanced comfort control, and improved indoor air quality and introduce air at the floor level, with return grilles located near the ceiling. The space is divided into two variable zones, an occupied zone extending from the floor to top level, and an unoccupied zone extending from the top of the occupied zone to the ceiling.

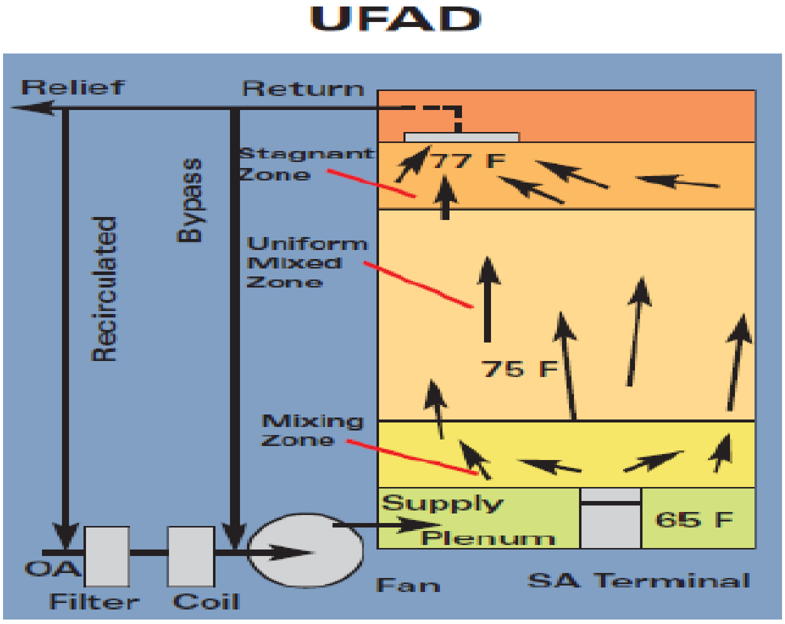

Figure 1: Underfloor air distribution system [Source: Testing Adjusting and Balancing Bureau talk for ASHRAE by James E. Woods]

The systems are designed for the lower occupied zone only. The temperature conditions in the upper zone are allowed to float above normal comfort ranges. To avoid occupant discomfort, air is introduced into the space between 65°F and 68°F.

Underfloor air distribution systems fall into two general categories distinguishable from one another by the temperature and velocity profiles they create in the occupied space. The first type is a displacement ventilation system; the second type is a hybrid underfloor system.

II. PROBLEM FORMULATION

The survey of different previous works we predict the temperature stratification in the most important parameters in order to maintain better comfort conditioning. The purposes of this study reduce the temperature difference in between the two horizontal layer and increase the momentum of the velocity at the free space area in order to reduce the temperature difference in between the horizontal plane.

This chapter is a review of general theory of the governing equations for fluid flow. The governing equations of fluid flow are called the Navier-Stokes equations. In this section, concisely we will discuss the principles of the CFD with its components. Moreover, fundamental description of turbulence will be described.

Governing Equations

- Conservation of mass
- Conservation of momentum
- Conservation of energy
- Conservation of species
- **Mass Conservation Equation**:

$$\frac{\partial \rho}{\partial t} + \nabla . \rho U = 0 \qquad (3.1)$$

Momentum Equation:

$$\frac{\partial \rho (u,v,w)^T}{\partial t} + (\nabla \rho U u, \nabla . \rho U v, \nabla . \rho U w)^T + \left(-\frac{\rho w^2}{r}, \rho g, \frac{\rho u w}{r}\right)^T = -\nabla \pi + \nabla . \tau \qquad (3.2)$$

Energy Equation:

$$\frac{\partial \rho h}{\partial t} + \nabla.\rho U h = \nabla.\lambda_e \nabla T - \nabla.q_{rad} + \nabla.\textstyle\sum_i \rho h_i(T) D_e \nabla Y_i \qquad \textbf{(3.3)}$$

Species Equation:

$$\frac{\partial \rho Y_i}{\partial t} + \nabla.\rho U Y_i = \nabla.D_e \rho \nabla Y_i - R_i \qquad (3.4)$$

The enthalpy h is defined by:

$$h = \textstyle\sum_i Y_i h_i(T) \qquad (3.5)$$

Where $h_i(T)$, is the specific enthalpy of species *i*, includes the heat of formation of *i*. When needed, T is computed using h, Yi. Where *Y*, *h*, $\lambda_{e,}$ ρ and R_i are the enthalpy mass fraction, effective diffusivity and the flow density and mass rate of consumption of species respectively.

Turbulence Model

Turbulence is simulated using the Standard k-ε model. In this model, the model transport equations are solved for two turbulence quantities i.e., k and ε. The Standard k-ε model turbulence model solves the flow based on the assumption that the rate of production and dissipation of turbulent flows are in near- balance in energy transfer. The k and ε equations are given by:

$$\frac{\partial \rho k}{\partial t} + \nabla.\rho U k = \nabla.(\alpha\mu\nabla.k) - R_{ij}\frac{\partial U_i}{\partial x_j} - \rho\epsilon \qquad (3.6)$$

$$\frac{\partial \rho \epsilon}{\partial t} + \nabla.\rho U \epsilon = \nabla.\left(\frac{\mu_t}{\sigma_t} + \mu\right)\nabla\epsilon - C_1.\frac{\epsilon}{k} R_{ij}\frac{\partial U_i}{\partial x_j} - C_2 \rho \frac{\epsilon^2}{k} \qquad (3.7)$$

The standard value of k-ϵ is used.

The flowchart of the CFD solver used to model air distribution system

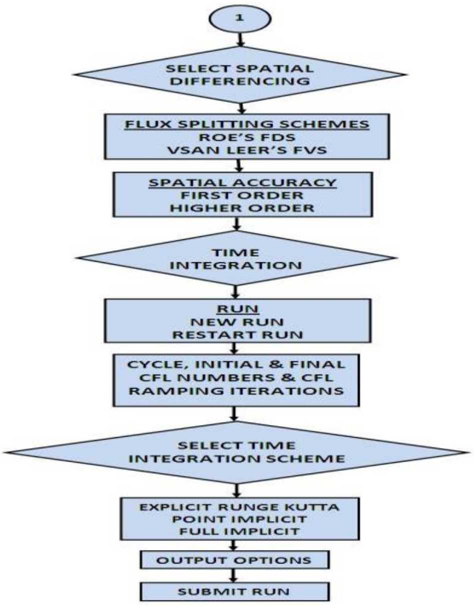

Figure 2: The flowchart of the CFD solver

III. CONCLUSION

In this work, numerical analysis is conducted in order to investigate the room air temperature stratification with UFAD system.

Quantitative analysis was presented. The results show that the different parameters effect the room air temperature stratification. Here it shows the effect of diffuser inlet temperature, heat load, volume flow rate and also shows the effect of diffuser inlet position on the velocity flow pattern and on the temperature stratification.

In order to provide the better mixing of air inside the system, it is necessary to find the optimum inlet velocity of air and also finding the optimum position of diffuser inlet so that it can provide the momentum to air molecules in order to provide the better comfort conditioning inside the system.

The above results of room air temperature stratification can provide the guidance on calculating the amount of cooling airflow needed to remove heat loads from a building space and establishing the model of heat transfer in a room with UFAD system.

Here we are finding the temperature difference in between the horizontal plane, the minimum difference temperature in between the horizontal plane shows the better mixing and also shows the better comfort conditioning inside the system. Here we have also calculated the effect of diffuser inlet and it shows that, type having diffuser at the middle of the system shows the least temperature difference in between the horizontal planes and shows the best case for comfort conditioning.

REFERENCES

[1] Holiday, A., Andreou, S., & Hayder(2015). M. Air Flow Visualization in an Office Room, European International journal of science and technology.

[2] Li, Y. (2012). Numerical simulation and analysis for indoor air quality in different ventilation. Health, 4(12), 1352.

[3] Sarkar, J., & Mandal, S. (2008). CFD Modeling and Validation of Temperature and Flow Distribution in Air-Conditioned Space.

[4] K Vaseemul Rahaman, Prof. Shankar Kumar, (2015), CFD Analysis of Air Conditioning equipment of CAD Lab for Enhancing its Performance,International Journal of Recent Development in Engineering and Technology, 1(5) , pp. 68-72.

[5] Sun, Z., & Wang, S. (2010). A CFD-based test method for control of indoor environment and space ventilation. Building and Environment, 45(6), 1441- 1447.

[6] Aryal, P., & Leephakpreeda, T. (2015). CFD Analysis on thermal comfort and energy consumption effected by partitions in air-conditioned building. Energy Procedia, 79, 183-188.

[7] Bamodu, O., Xia, L., & Tang, L. (2017). A numerical simulation of air distribution in an office room ventilated by 4-way cassette air-conditioner. Energy Procedia, 105, 2506-2511.

[8] Tian, W., Sevilla, T. A., Li, D., Zuo, W., & Wetter, M. (2018). Fast and self-learning indoor airflow simulation based on in situ adaptive tabulation. Journal of Building Performance Simulation, 11(1), 99-112.

[9] Yang, W., Zhu, X., & Liu, J. (2017). Annual experimental research on convective heat transfer coefficient of exterior surface of building external wall. Energy and Buildings, 155, 207-214.

[10] Wang, H., & Zhai, Z. (2012). Application of coarse-grid computational fluid dynamics on indoor environment modeling: Optimizing the trade-off between grid resolution and simulation accuracy. HVAC&R Research, 18(5), 915-933.

[11] Romano, F., Colombo, L. P., Gaudenzi, M., Joppolo, C. M., & Romano, L. P. (2015). Passive control of microclimate in museum display cases: A lumped parameter model and experimental tests. Journal of Cultural Heritage, 16(4), 413- 418.

[12] Popovici, C. G. (2017). HVAC system functionality simulation using ANSYS- Fluent. Energy Procedia, 112, 360-365.