A REVIEW PAPER ON EVALUTION OF CIRCULAR SWIRL FLOOR DIFFUSER UNDERFLOOR AIR DISTRIBUTION SYSTEM USING CFD

Shishir kumar saini[1], Mr.Sachin Baraskar[2]

[1] Research Scholar, Mechanical Department, SSSUTMS, M.P.,INDIA
[2] Assistant Prof., Mechanical Department, SSSUTMS, M.P.,INDIA

ABSTRACT

A 3D-CFD investigation of airflow, temperature distribution and thermal comfort in laboratory room of a university conditioned with underfloor air distribution (UFAD) system is presented for different operating and geometric conditions. The geometric configuration is a room (4.65m x 3.63m x1.98m) that includes four windows with the same dimensions (0.735 x 0.92 m), two circular diffusers having the diameter of 0.3568m and an exhaust grille (0.74 x 0.3m). While in the previous work, the geometric configuration of the room was same as in our work except two rectangular diffusers having the same dimensions (0.5 x 0.2m). Simulations are implemented, using a commercial CFD package (Fluent 14.5), to understand the effects of supply air temperature, supply air velocity, space height and number of supply air diffusers on the performance of the air conditioning system and thermal comfort. The airflow and temperature distribution in the conditioned space, UFAD system is capable of creating smaller vertical variations of air temperature and a more comfortable environment and energy saving the supply air velocity and temperature, number of diffusers and height of the space have a significant impact on thermal comfort.

Keyword : -*CFD Investigation,UFAD System,Temprature,Velocity.*

1. INTRODUCTION

Under floor air distribution (UFAD) is a method of delivering space conditioning in offices and other commercial buildings that is increasingly being considered as a serious alternative to conventional overhead air distribution systems because of the significant benefits that it can provide. This technology uses the space (under floor plenum) between the structural concrete slab and the underside of a raised access floor system to deliver conditioned air directly into the occupied zone of the building. Air can be delivered through various supply outlets located at floor level (most common), or as part of the furniture and partitions. UFAD systems have several potential advantages over traditional overhead systems, including better thermal comfort, improved indoor air quality, and reduce energy use. By combining a building's heating, ventilating, and air-conditioning (HVAC) system with all major power, voice, and data cabling into one easily accessible service plenum under the raised floor, significant improvements can be realized in terms of increased flexibility and reduce costs associated with reconfiguring building services.

1.1 Technology Description

UFAD systems are uniquely characterized by their function to allow individuals to have some degree of control over their local environment, without adversely affecting that of other nearby occupants. The kinds of diffusers supported, active or passive, further distinguish UFAD systems. Active diffusers (for purposes of this report) are explained as those with local means of volume adjustment (such as an integral variable speed fan or that is amenable to automatic zone control (in addition to means for occupant control). Passive diffusers, although they may have means for occupant adjustment, are combined with terminal or system elements to achieve zone comfort. Systems designed with all fan-assisted active diffusers typically utilize zero-pressure plenums. Passive diffusers need pressurized plenums. The majority of UFAD systems presently being deployed have pressurized plenums with either active or passive diffusers

1.2 System Overview

UFAD is a more energy efficient system than the ‘mixing flow’ ceiling based distribution system, due to the following reasons:

i. Air is provided at low flow rates, as buoyancy is utilized to spread air through the space. Low flow rates result in the reduction of energy usage of the fans used to distribute air by the under floor ducts.
ii. The supply air is at a relatively high air temperature (i.e. 19°C), as the air is supplied closer to the source of heat gains. Relatively high air temperature means lower chilled water temperatures, resulting in an effective reduction of the heat load / tonnage of the chillers installed.
iii. Since new air is kept separate from stale air, the breathed air quality is substantially uncontaminated.

2. LITERATURE REVIEW

Ahmed CherifMegriet. al. (2017) demonstrate not only the benefit of UFAD system in a residential house in terms of energy saving but also the significance of thermostat location in the forecast of building energy consumption and results also indicatethat the conventional single/multi-one models are not suitable to use for UFAD systems in the building energy demand predictions.

AbdAlnasserAlmate A. Ali et.al.(2014)demonstrate the use of Computational Fluid Dynamic (CFD) in building applications and show flow and thermal patterns, using temperature and heat flux boundary conditions obtained experimentally.The steady-state simulation of an Under Floor Air Distribution (UFAD) system was investigated using the program FLUENT. Numerical analysis have been performed to forecast the temperature and the velocity distributions within a conditioned single room using cooling UFAD system. The study of the vertical momentum of the supply airflow, the distance from diffuser, the orientation of the boundaries on the airflow and the temperature gradient were analyzed for a three dimensional enclosure, with two supply diffusers located at the center of the room, two windows, on south and east walls and an exhaust grille.

Ahmed CherifMegri et al. (2014)demonstrate the application of Computational Fluid Dynamic (CFD) in building execution. In specific, CFD has been used for temperature and airflow prediction of building spaces conditioned using UFAD System. The space used is an instrumented laboratory room (old daycare center) located at the University of Wyoming.

Juan et al. (2013) showed a CFD model on the real atmosphere of computer room. In their study, the geometric model was created using the parametric qualities of the pre-processor Gambit, in combination with elements created with the Rhinoceros NURBS modeling tool.

Joseph et al. (2012) numerically demonstrated the air movement and temperature distribution within an air-conditioned gymnasium with four different, but commonly found, exhaust positions in Hong Kong.

Hirnikel et al. (2010)performed investigations on three air distribution systems to check these contaminant removal effectiveness investigated contaminant removal effectiveness for a bar/restaurant by using CFD modeling. They found that directional airflow systems could reduce people’s exposure to contaminants.

Nakahara et al. (2009)this study is the second part of previous research (see reference above), dealing with simulation analysis on seasonal loss. Mixing energy Loss (ML) was estimated using regressive models which have been developed based on the experimental results of Part 1. Dynamic heat-loss calculation method, which includes case studies to estimate the seasonal properties of heat that are supplied to a space by regressive model of mixing energy loss. Factors on the seasonal heat consumption were also discussed.

Sandberg et al.(2008)This paper concludes the measurement results and shows the existence of limitations of displacement ventilationperformed in an office room. There are specific design requirements for supply air terminals and restrictions on the maximum heat load due to the temperature gradient. The breathing zone concentrationis the same as with traditional mixing systems when the normal flow rates are supplied to office rooms.

Seem et al. (2007)This paper compares the energy use characteristics ofconventional HVAC systems andpersonal environmental control (PEC) systems, concluding that the advantages of increased staff productivity outweigh costs in other areas (from UFAD Design Guide by Bauman).

Son et al. (2005) gives a detailed numerical simulation of contaminant transportand thermal comfort in air conditioned rooms. However, their consequences are limited to two-dimensional geometry, and the real environment is simplified into regular geometry. Recently, under floor air distribution (UFAD) systems have become popular design alternatives to overhead air distribution systems for thermal and ventilation control. This system was first introduced to cool a computer room in 1950s and is emerging as a leading ventilation system design in modern commercial buildings.

Woods (2004) did a literature review, field investigationsandsearches, to assess the actual performance of UFAD system in the real world. He explained that there are gaps in available data: valid and reliable field data are not from a sufficient population of existing facilities to conclude that an under floor system's performance is superior to an overhead system; and that designers must be made aware that under floor as well as overhead systems require more care in design, installation, and operations. He also suggested that objective analysis should be made before choosing an HVAC system.

Webster et al. (2002)explained a series of full-scale laboratory experiments to determine room air stratification for a variety of design and operating parameters.

Fukao et al. (2002) carried out comparative field measurements for both UFAD and OHAD systems in an actual large-scale office building.

3. METHODOLOGY

Computational fluid dynamics is the computer based analysis by which we can analyze the various things like fluid flow, pressure distribution, heat transfer, and related to the phenomenon in the chemical reactions. And the CFD simulation software is predicted to the impact of the fluid throughout the designing as well as during the end of the use. The CFD gives the qualitative prediction of the fluid flow with the assistance of the following tools:-

- Mathematical model (partial differential equations).
- Numerical method (discretization and solution of the problem).
- Software tools (pre and post processing, solver).

The CFD simulations are used in the various field of the fluid flow for example:-

- Design the vehicle to improve the fluid characteristic.
- In chemical engineering to maximize the yield of their equipment's.
- Architects engineers used to design the home for safe living.
- Military organizations to advance weapons and estimate the damage.
- Aerodynamic lift and drag i.e. airplanes and wind mills.
- Power plant combustion
- Biochemical engineering like simulating the blood flow in the veins.

There are the many advantages to use the CFD analysis compare to the traditional experiment analysis. In the experimental process the cost is increases are decreases with simultaneous increase or decrease in number of the configurations required for the testing. But in the CFD analysis large amount results can be obtained by adding the expanses once. In the CFD analysis we can optimized the equipment's parameters in a very inexpensive way. This section is briefly describing the concept of the CFD analysis. This project is related to the temperature distribution and the fluid flow in the evaporator equipment.

3.1 Computational Tool

This section describes about the CFD tools which are required for the CFD analysis of the problem. There are the three key elements for the processing of the CFD simulations: the pre-processor, solver, and post-processor are defined.

1.Pre-processor: A pre-processor is defined to the geometry of the problem. And it is fixed to the domain for the computational analysis and then generates the mesh of the geometry. Here also set the nomenclature like inlet, outlet, and wall etc. Generally, the finer the mesh of the geometry in the CFD analysis gives more accurate solution. Fineness of grid also determines the computer hardware and more time needed for the calculations.

2.Solver: - In the solver processor the calculations is done by using the numerical solution methods. There are the many numerical methods which are used for the calculations for example:-the finite element method, thefinite difference method, thefinite volume methodand spectral method. Most of them in CFD codes use finite volume method. In this project the finite volume method is used. The solver perform the following steps:-

• Firstly the fluid flow equations are integrated over the control volumes (resulting in exact conservation of applicable properties for individual finite volume),

• Discretization of these integral equations (producing algebraic equations through converting of the integral fluid flow equations),

• And then finally an iterative method is used to solve the algebraic equations.

• Pressure based coupled solution method CFD code is used for solving the simulations in this project.

3. Post-Processor: The post-processor is provided to the visualisation of the results of the solutions. It contains the ability to demonstrate the geometry and mesh also. And in this processor we can create the vectors, contours, and 2D and 3D surface plots of the problem solutions. Here the model also can be manipulated. In this process one can also get the animation of the problem.

4. CONCLUSIONS

The major objectives of this project were to a) develop CFD models of CAD and UFAD systems, particularly of the type of diffusers used in the Buildings Technology b) verify the developed models using the previous model from Task; and c) provide detailed information regarding velocity and temperature concentrations throughout the domain of interest. It was hoped that by developing a verified CFD model of a UFAD system, numerical simulations could supplement previous results and eventually a model could be develop to design UFAD system for real world applications.

5. REFERENCES

1. ASHRAE Inc. 1997. ASHRAE Handbook of Fundamentals, American Society of Heating, Refrigerating and Air Conditioning Engineers, Inc., Atlanta, Georgia.

2. ASHRAE Inc. 2004. Thermal Environmental Conditions for Human Occupancy. ANSI/ASHRAE Standard 55-2004.

3. Bauman F. 1999. Giving occupants what they want: guidelines for implementing personal environmental control in your building. Proceedings at World Workplace 99, Los Angeles, CA.

4. Bauman F and Webster T W. 2001. Outlook for underfloor air distribution. ASHRAE Journal, 43(6):18– 27.

5. Bjerg B, Svidt K, Zhang G, Morsing S, and Johnsen J. 2002. Modeling of air inlets in CFD prediction of airflow in ventilated animal houses, Computers and Electronics in Agriculture, 34: 223-235.

6. Chen Q, and Xu W. 1998. A zero-equation turbulence model for indoor airflow simulation, Energy and Building. 28:137-144.

7. Chow W.F., and Fung W.F. 2016. Numerical studies on indoor air flow in the occupied zone of ventilated and air-conditioned space, Building and Environment, 31: 319-344.

8. Dorer V, Weber A. 1999. Air, contaminant and heat transport models: integration and application, Energy and Buildings, 30: 97-104.

9. Emmerich S. 2017. Use of computational fluid dynamics to analyze indoor air quality issues. NISTIR 5997, Building and Fire Research Laboratory, National Institute of Standards and Technology, Gaithersburg, MD.

10. Fan Y. 1995. CFD modeling of the air and contaminant distribution in rooms, Energy and Buildings, 23:33-39.