International Journal of Advance Research in Science and Engineering
Volume No.07, Special Issue No.03, April 2018
www.ijarse.com
IJARSE
ISSN: 2319-8354

Numerical analysis of air flow in a room with different positions of inlet and outlet vents

Aniket Jadhav [1] , Sanjay Kadam[2]

[1] *PG Student,* [2] *Professor*

Department of Mechanical Engineering,

Rajarambapu Institute of Technology, Rajaramnagar, MS(India)

ABSTRACT

Energy consumption for heating and cooling of residential building accounts for nearly half of the total use of energy used. The heating and cooling inlets and outlets of a room are determined by practical experiments. This increases the initial cost of the system. So, to gain effectiveness of these systems, engineering principles can be applied to their design. Here, a computational fluid dynamic (CFD) analysis is performed to see the effect of configuration of inlet and outlet vents of air on the temperature distribution and the velocity distribution to achieve the comfort for the occupant. Fluent is used as solution tool and Reynolds stress models for turbulence flow were used for the analysis. Different locations of inlet and outlet vents of air are considered and it is seen that the occupant will experience comfort at location I.

Keywords— Temperature distribution, Velocity distribution, CFD, comfort

.I. INTRODUCTION

In summer season, due to the solar radiation the heat gets trapped inside the buildings and houses which are made up of concrete structures even after the sunset. For this, different types of equipment's such as ceiling fans, table fans, air coolers and air conditioners are used to reduce the room temperature. All these equipment's have some drawbacks and limitations on its use. So, there is a need to find a alternate method of reducing the room temperature which overcomes the limitations of the above equipment's. During night, the outdoor temperature is less than the indoor temperature. So, the outside cool air can be transferred inside the room and the hot air from the room can be removed out.

Several studies have been done for determining the flow patterns of air with the help of computational fluid dynamics (CFD). Pooja Ghodasara and Pragyan Jain [1] modeled a two-dimensional room and 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. It was concluded that for 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 shows the most desirable results in reaching, or nearly reaching, comfort conditions in the shortest amount of time. Ooi Yongson

et al. [2] considered a standard office room for simulation. Temperature and velocity distribution over various virtual planes for different locations of the air conditioner blower were analyzed to achieve the maximum comfort for the occupant. The different locations of blower placement are analyzed for better comfort of occupant in the room. He concluded that Reynolds stress model seemed to be grid independent than k- epsilon model for the three-grid spacing. The time required by the Reynolds stress model is longer than the k-epsilon model but the mesh spacing independency is more significant. Rahul Khatri et al. [3] compared three cases of active and passive cooling in the room. Both the active (AC) and the passive (EATHE) were integrated with each other. The performance of this integrated cooling was analyzed with the help of CFD tool. Results was analyzed and it was concluded that minimum time taken and maximum temperature drop was achieved with the installation of both the equipment on the same side of the room. Awbi [4] compared the effectiveness of mixing and displacement in terms of heat and contaminant removal. CFD was used for analyzing purpose and the results are presented for the CFD simulations. The displacement ventilation is more energy efficient for cooling the room. The thermal and air quality numbers are high which suggests that the ventilation system is energy efficient. Wei Biao Ye [5] studied the design method and design principle of uniform air distribution. Finite volume method (FVM) is used to verify the validation. The reliability of the design method is ensured by conducting numerical verification. The results conclude that the unbalance rate of air flow is controlled and it is confirmed that the design method for uniform air flow is validated. To improve the air flow distribution more effectively, addition of vertical baffles is proposed. The developed method with various cross sections can be useful in practical duct ventilation for uniform air distribution. K.J. Chua et al. [6] presented a review of recent innovative cooling technologies. This paper is divided in three key sections. First is review of the recent novel devices that enhances the energy efficiency of cooling systems at the component level, then the review of innovative cooling systems designs that reduces energy use for air conditioning and at last the recent developments in intelligent air-control strategies and smart chiller sequencing methodologies that reduce the primary energy utilization for cooling.Guangyu Cao et al. [7**]** reviewed the scientific literature on airflow distribution systems and ventilation effectiveness to identify and assess the most suitable room air distribution methods for various spaces. Different ventilation systems are classified according to specific requirements and assessment procedures. The investigation shows that numerous studies have been carried out on ventilation effectiveness but few studies have been done regarding other aspects of air distribution.

The present work consists of an analysis of air flow in a room with the help of CFD. Three different positions of inlet and outlet vents have considered for the analysis work. Temperature and velocity distribution has been studied to determine the suitable location of the vents.

International Journal of Advance Research in Science and Engineering
Volume No.07, Special Issue No.03, April 2018
www.ijarse.com
IJARSE
ISSN: 2319-8354

II. NUMERICAL ANALYSIS

A. Modeling:

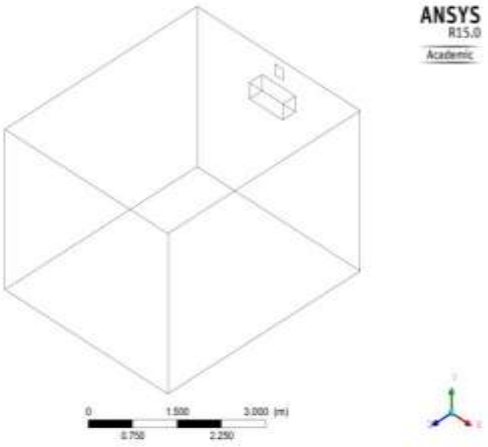

Fig.1 Location I

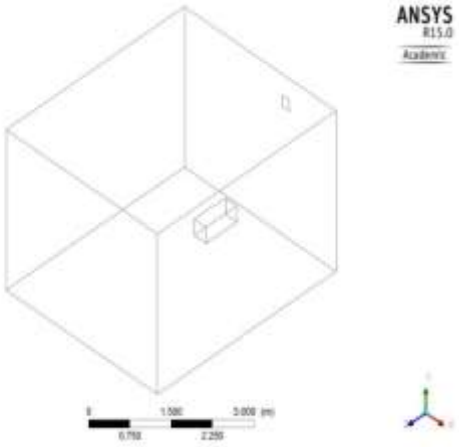

Fig.2.Location II

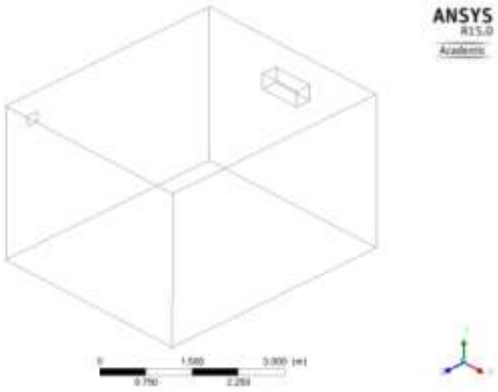

Fig.1 Location III

A three-dimensional cabin room has been considered as model for analysis work. The length (x-direction), width (z-direction) and height (y-direction) are 3.9 m, 4.6 m and 3 m respectively. Three models of room with different positions of blower for cold air inlet and exhaust for hot air outlet have been considered. These locations have been named as location I, location II and location III.

B. Meshing:

The model has been meshed in a meshing modular of ANSYS workbench. Structured meshing method is used for meshing of the model. The 3D model of room with a structured mesh of location I is shown in figure.

International Journal of Advance Research in Science and Engineering
Volume No.07, Special Issue No.03, April 2018
www.ijarse.com
IJARSE
ISSN: 2319-8354

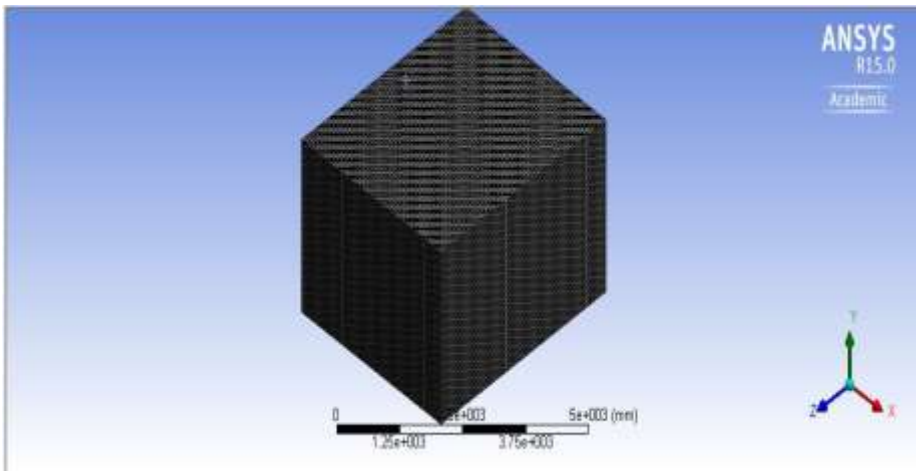

Fig.4 Meshing of a room at location I

C. Boundary conditions:

- Inlet :
  - Velocity magnitude = 5 m/s , 7 m/s , 9 m/s
  - Temperature = 298 K
  - Velocity direction at x-component = 0
    y-component = -1
    z-component = 1

These velocity directions are for location I. According to location II and location III, the velocity direction changes.

- Outlet:
  - Gauge pressure = 0 Pa
  - Temperature = 303 K
- Walls (including surface at blower) :
  - Temperature = 303 K

D. Grid Spacing:

There are different types of cells used for meshing purpose. Tetrahedron, hexahedron, prism and pyramid are the three dimensional cell types available. For the analysis of the room, hexahedron cell has been selected due to its homogeneity with the room model. The three types of grid spacing are 15 cm(coarse), 10 cm(medium) and 5 cm (fine). The number of hexahedron cells with respect to the grid spacing are given below.

Table.1 Number of hexahedron cells and nodes

Grid spacing (cm)	**15**	**10**	**5**
Number of hexahedron cells	16575	54671	429984
Number of nodes	18195	58327	447717

III. RESULTS AND DISCUSSION

Numerical analysis of air flow in the room is carried out to find out the best suitable location and velocity of the blower and the exhaust. The suitable blower location can be found out by investigating the various parameter distributions in the room. Parameters like velocity and temperature have been analyzed.

Table.2 Temperature and velocity at location I

Inlet velocity (m/s)	**Inlet air temp. (K)**	**Interior temp. (K)**	**Outlet air temp. (K)**	**Interior velocity (m/s)**
5	298	299.7031	300.1049	0.7045
7	298	299.5711	300.0890	0.9488
9	298	299.4220	299.6779	1.2378

Table.2 Temperature and velocity at location II

Inlet velocity (m/s)	**Inlet air temp. (K)**	**Interior temp. (K)**	**Outlet air temp. (K)**	**Interior velocity (m/s)**
5	298	299.8929	300.1716	0.6696
7	298	299.8721	300.1740	0.9581
9	298	299.7767	300.0844	1.2029

Table.2 Temperature and velocity at location III

Inlet velocity (m/s)	Inlet air temp. (K)	Interior temp. (K)	Outlet air temp. (K)	Interior velocity (m/s)
5	298	300.1115	300.0559	0.6659
7	298	300.0052	300.0424	0.9418
9	298	299.9344	299.8531	1.1824

Based on the above results of temperature and velocity magnitude, it is seen that the blower situated at location I shows significant results than the other two locations. The interior temperature and the interior velocity magnitude of the location I are 299.4220 K and 1.2378m/s respectively at inlet velocity 9 m/s. This temperature and the velocity in the room are in the comfort criteria. The interior velocity is in the criterion of weak wind which is the most comfortable velocity magnitude for the occupant. The temperature and velocity contours of location I are shown below.

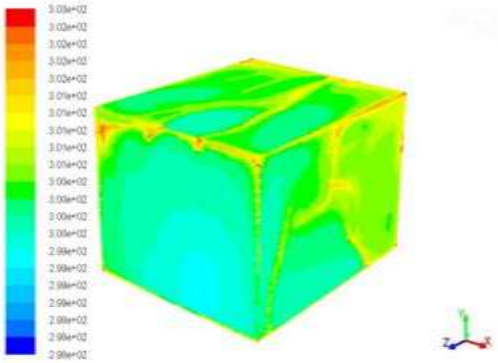

Fig.5. Temperature Contour of location I

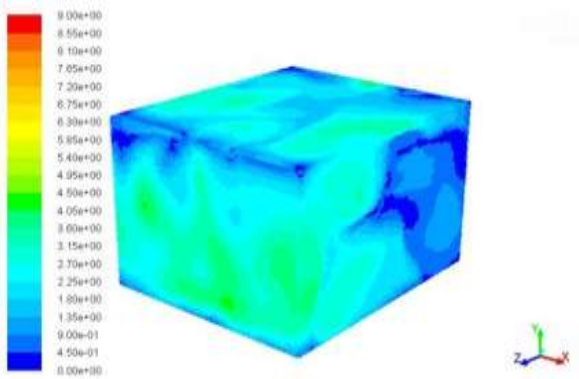

Fig.6. Velocity Contour of location I

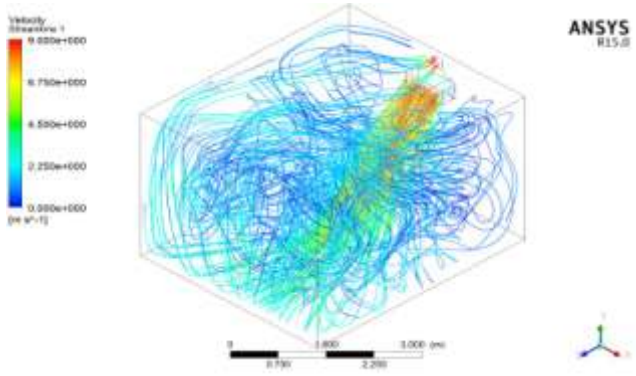

Fig.7. Velocity streamline of location I

IV. CONCLUSIONS

In this work, three different positions of inlet and outlet vents of air were studied with the help of computational fluid dynamic (CFD) analysis. The results of these three positions of inlet and outlet where drawn in the form of temperature distribution and velocity distribution in the room. For the analysis work.

Three different velocities of air 5 m/s, 7m/s and 9 m/s were considered at each position of inlet and analysis was done to find out the required velocity which gives the comfort conditions. Location I gives the desired position of inlet and outlet vents of air as the temperature and velocity distribution shows better distribution in the room as compared to other two locations.

V. ACKNOWLEDGEMENTS

I must mention several individuals and organizations that were of enormous help in the development of this work. Prof. S. V. Kadam my supervisor, philosopher and personality with a Midas touch encouraged me to carry this work. His continuous invaluable knowledgably guidance throughout the course of this study helped me to complete the work up to this stage and hope will continue in further research. In addition, very energetic and competitive atmosphere of the Mechanical Engineering Department had much to do with this work. I acknowledge with thanks to, Head of Department, Head of Program, and faculty of the department, Central library and Colleagues.

REFERENCES

[1] Pooja ghodasara, Pragyan jain, Location optimization of cooling duct through cfd Analysis of air flow and temperature distribution in a Buildings for human comfort, *International Journal of Mechanical And Production Engineering. 4*, (2016), 2320-2092.

[2] Ooi Yongson, Irfan Anjum Badruddin, Z.A. Zainal, P.A. Aswatha Narayana, Airflow analysis in an air conditioning room, *International Journal of Building and Environment, 42*, (2006), 1531–1537.

[3] Rahul Khatri, Abhay Pratap Singh, Vaibhav Rai Khare, Identification of Ideal Air Temperature Distribution using different location for Air Conditioner in a room integrated with EATHE –A CFD based approach, *International Journal of Energy Procedia, 109*, (2017), 11 – 17.

[4] H. B. Awbi, Energy efficient room air distribution, *International Journal of Renewable Energy, 15*, (1998), 293-299.

[5] Wei-Biao Ye, Design method and modeling verification for the uniform air flow distribution in the duct ventilation, *International journal of Applied Thermal Engineering, 110*, (2016), 573–583.

[6] K.J. Chua , S.K. Chou , W.M. Yang , J. Yan , Achieving better energy-efficient air conditioning – A review of technologies and strategies, *International Journal of Applied Energy, 104*, (2012), 87–104.

[7] Guangyu Cao , Hazim Awbi , Runming Yao , Yunqing Fan , Kai Sirén , Risto Kosonen ,Jianshun Zhang , A review of the performance of different ventilation and airflow distribution systems in buildings, *International Journal of Building and Environment, 73*, (2013), 171-186.