

Computational Fluid Dynamics Analysis of Effective Utilization of Air Conditioning in Class Room.

Ashish Mogra^{*a}, Nishant Dhengre,^a Dr. K.K. Gupta^b, Vijayendra Singh Sankhla^c, Ravindra Parmar^d

^aAssistant Professor, NMIMS, MPSTME, Shirpur

^bProfessor, NMIMS, MPSTME, Shirpur

^cAssistant Professor, GITS, Udaipur

^dM Tech RET , GITS, Udaipur

* Corresponding author.

E-mail address: Ashish.mogra@nmims.edu, Tel.: +91-9604066126

Abstract-Thermal comfort and indoor air quality affects the productivity of an individual to a great extent. Students as well as professors spend almost half of their day in classrooms, hence the air flow distribution from the air-conditioning systems plays an important role in determining whether the students will receive the air velocity and air temperature to the comfortable accepted range. In this paper a Computational Fluid Dynamic (CFD) simulation is performed on the overhead air-conditioning system of a classroom. A CFD model has been developed for position of air conditioner in class room. Originally the position of air conditioner is centrally located. A comparison has been made in between two model placed centrally and proposed diagonally position. It is found the better air flow distribution when air conditioner is placed diagonally.

Keyword: Air conditioning, CFD, boundary condition, thermal comfort, velocity distribution.

Introduction

Indoor comfort is now considered one of the most essential parameter as it has been scientifically proved that it affects the productivity and efficiency of an individual. Thermal comfort is related to temperature and airflow distributions in air conditioning space, which plays an important role in optimum design of air conditioning system or ventilation system. This study was performed in a classroom of SVKM's NMIMS having an overhead air conditioning system with an aim to understand air flow and temperature distribution. The position of the existing air conditioners were also changed to compare the effects with the existing one. It was discovered that the position of the air conditioning unit will affect the air flow distribution which will further cause changes in thermal sensation among occupants. For this study Computational Fluid Dynamics (CFD) is used to investigate the air flow pattern and air flow distribution, which will help to identify the locations where air distribution is inadequate.

Holiday et al. (2015), in their study the office room was virtually created to make it ready for flow simulation. To do that, the dimensions of the room and all furniture sets were measured accurately and then everything created was Part files which were later assembled to prepare the office room model. Yang Li (2012), performed numerical simulation and analysis for indoor air quality in different ventilation concluded that the displacement ventilation mode can create an ideal velocity field, temperature field for personnel activity area, improve indoor thermal comfort efficiently and can improve the indoor air quality as well. Sarkar and Mandal (2008), performed study on CFD Modelling and Validation of Temperature and Flow Distribution in Air-conditioned Space by provided experimental study on flow pattern and velocity distribution in an enclosed space with a single inlet and outlet had been conducted. The flow pattern, velocity and temperature distributions in the above enclosed space had been investigated using

three dimensional CFD simulation and the results obtained from the simulation were validated with experimental data. Rahaman and Kumar (2015), did a study on CFD analysis of air conditioning of CAD lab by about CFD model and its calibration methodology bridges the gap between understanding and application of CFD simulation and field measurement for natural ventilation systems. Sun et al. (2010) also provides a model of a CFD study of an indoor environment. A dynamic simulation was used to evaluate HVAC control systems based on a CFD model of a room, a mathematically modelled PID controller, and an actuation model. While the control model was beyond the scope of this current research, the CFD model was a helpful guide in gauging model inputs. Aryala and Leephakpreedaa (2015), performed CFD Analysis on Thermal Comfort and Energy Consumption Effected by Partitions in Air-Conditioned Building concluded that the CFD experiments show that the thermal comfort level deteriorates in some specific regions with the installation of partition and the energy consumption is increased by 24%. Thermal comfort and energy consumption can subsequently be improved by adjusting air parameters at the supply diffuser and locations of extract grilles.

Methodology

The following assumptions are made for simulation:-

1. The air conditioning system in selected lecture rooms is fully functioning and running well.
2. The rooms are fully sealed and enclosed without any holes or gaps (excluding doors, windows and exhaust vent).
3. The outside temperature on the surface of the room is constant
4. Internal heat source emitted from the digital devices and lights will be neglected due to minimal effect on the temperature.
5. The furniture (chairs and tables) are included in the simulation.
6. The temperature over the top of the desk is assumed to be human body temperature (i.e. 303K).
7. Air from the air conditioners flow vertically downwards.
8. Velocity of air is usually between the range (0.08-0.3) m/s and for the present study, air velocity is assumed to be 0.2 m/s.

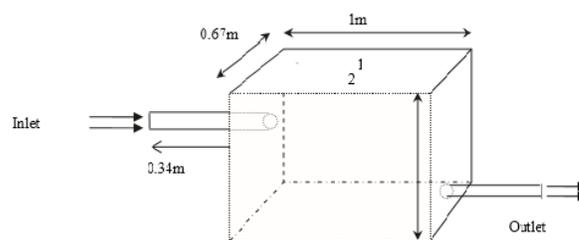
The following boundary conditions have been used in CFD simulation:

At inlet: $V_x = V_i$, $V_y = 0$, $V_z = 0$; $T = T_i$

At outlet: $P = P_{atm}$

At side wall: Isothermal, $T = T_w$

At top and bottom wall: Constant heat flux, $\frac{\partial T}{\partial y} = 0$



Condition 1: Centrally Air-Conditioner arrangement in the classroom:

The class room was virtually created to make it ready for flow simulation. To do that, the dimensions of the room and all furniture sets were measured accurately and each object was created as Part files, which were later assembled to prepare the class room model. The dimensions of the room and furniture sets are listed in Table 1. The location of all furniture sets, the air conditioning unit, and the door was kept the same as that of the original room to maintain the resemblance of the model with the class.

Table 1

ITEM	DIMENSION (cm)	THERMAL CONDUCTIVITY ($\frac{W}{m/k}$)	MATERIAL
DOOR	248x114x7	0.17-0.25	Wood
BENCH	140x78x84	0.17-0.25	Wood
WINDOW	213x233	0.90-1.5	Glass
WHITEBOARD	122x245	1.52-1.65	Ceramic (Silicon Carbide)
AC	95x95		Daikin 2 tons cassette
WALLS	303x141x35	2.07-2.94	Brick Wall
Beam	28x45	0.72-0.82	Plaster Material

The exploded and the assembly views of the office model are shown in Figures 1 and 2, respectively. Analysis under CFD Fluent will be performed on the designed model. After input of desired data, CFD models will be generated and results for different operating conditions were obtained and compared.

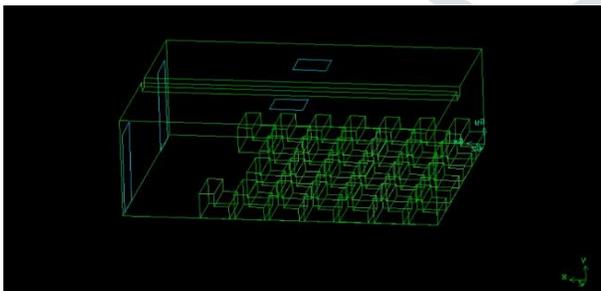


Figure 1 Line diagram of centrally located AC in class room.

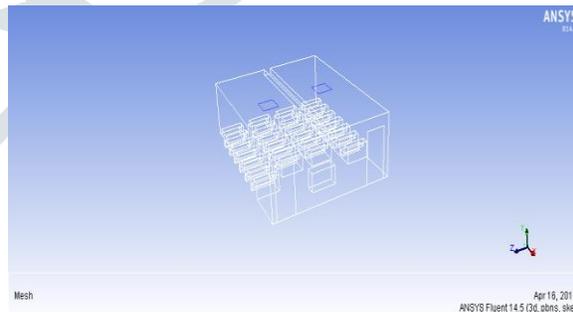


Figure 2 ANSYS model of centrally located AC in class room.

Condition 2: Diagonal arrangement of air-conditioners:

All the other parameters are exactly the same as that of condition-1. Assumptions and boundary conditions are also kept similar to condition-1. Considering the center of the classroom as an origin point, the coordinates of the air-conditioners is as follows:

- Unit-1: (x, y, z) = (2, 3.03, 2) m
- Unit-2: (x, y, z) = (7.07, 3.03, 5.53) m

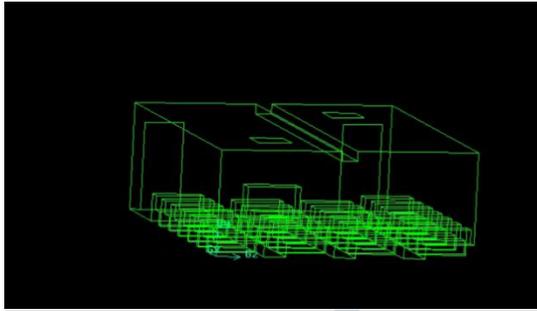


Figure 3 Line diagram of diagonally located AC in class room.

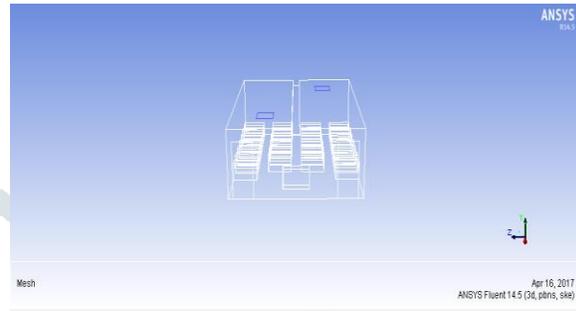


Figure 4 ANSYS model of diagonally located AC in class room.

Result Analysis

The air flow inside the room at two different front sections, at sections close to the window and middle of the room. Air enters the room with a high velocity and after circulating inside the rooms, it also leave the room through the vent with a relatively high velocity; but its exit velocity is lower than the entering velocity. In the sitting elevation level of the room's occupant, the high velocity zones are located in areas close to the rear glass window. The entire area in front of the window always experiences a higher air velocity. Following benefits are observed while placing air conditioners into diagonally:

- By placing one unit above the open space near door, it was possible to eliminate greater turbulence and velocity variations.
- A lower variation of velocity over the sitting area was observed.
- Variation between maximum temperature and minimum temperature was comparatively less.
- Thermal discomfort caused by the turbulence nature of air around the benches was reduced to some extent.

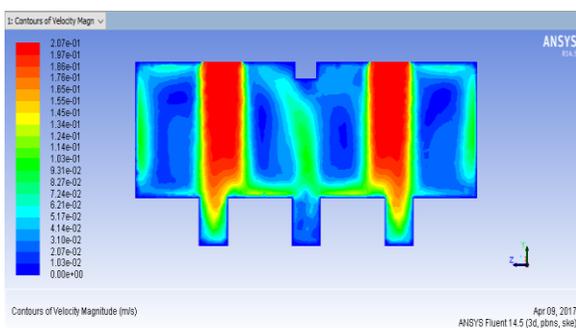


Figure 5 Velocity counter 1.5 m above the ground

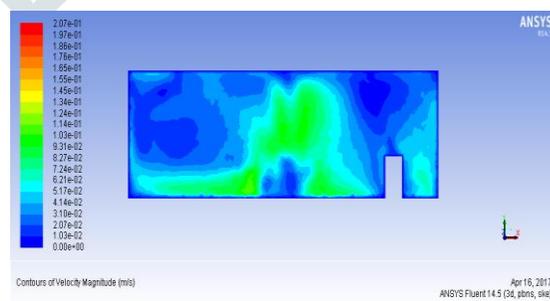


Figure 6 Velocity counter window to door.

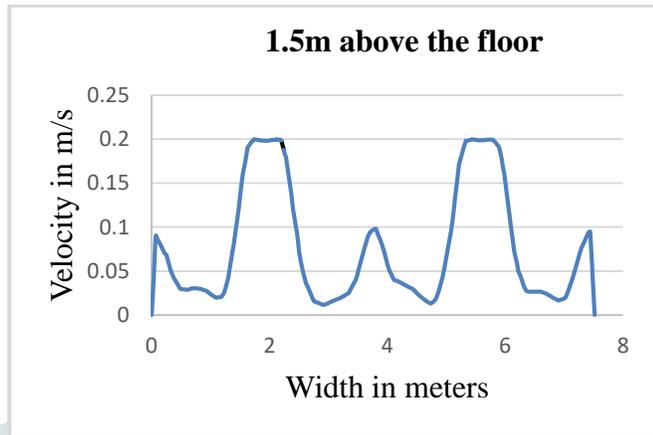
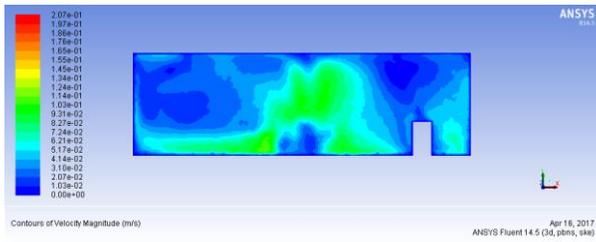


Figure 7: Velocity contour in center of classroom from door to window
 Figure 8 Velocity graph from left wall to right wall

Figure 7 shows the Velocity distribution of air in the actual classroom from a reference line drawn at 1.5meter above the ground and from left wall to right wall at the center of classroom. Velocity of air at inlet of air conditioner is considered as 0.2m/s (*from the study, range of velocity of air for Air conditioners is between .08-.30 m/sec depending on the fan speed). Graph is plotted against the velocity and width in meters.

Above graph shows the variation in Velocity which ranges from 0.015m/s to 0.2m/s. This shows that velocity distribution is uneven. Maximum velocity (i.e. 0.2m/s) lies at the point just below the two air conditioners i.e. 2m and 6m from left side wall. Minimum Velocity. (I.e. 0.15m/sec) lies at the point 3m and 7m from the left side of wall because flow of air is not distributed properly. This is due to vertically downward flow of air (assumption) as the benches and walls are responsible for turbulent flow of air. Graphs obtained from results are also symmetric because air conditioners are place at the same distance (2meter) from left side and right of the wall.

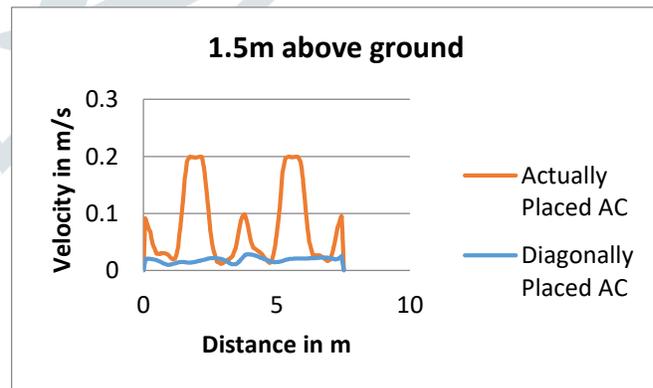
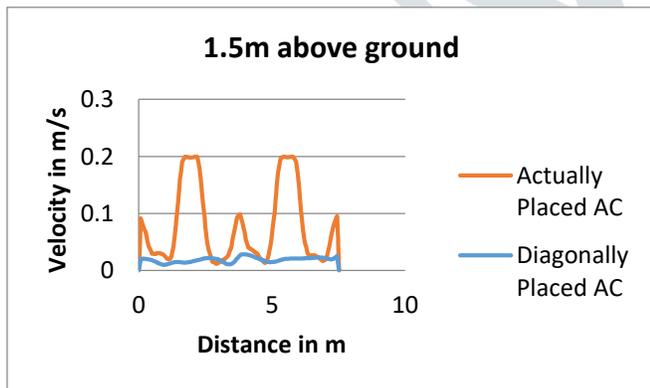


Figure 9 Velocity comparison from left wall to right wall.

Figure 9 Velocity comparison from left wall to right wall.



Figure 10 Velocity comparison from window to door.

Acknowledgment

Authors acknowledge the SVKM'S NMIMS, MPSTME, shirpur for providing facilities for doing research.

References

- [1] A. Holiday, S.Andreou, M. Hayder; "Air flow visualization in an office room"; European International journal of science and technology, Vol.4 No.4; April 2015.
- [2] Y.Li "Numerical simulation and analysis for indoor air quality in different ventilation" Health, Vol.4 No.12, 2012.
- [3] J. Sarkar, S. Mandal; "CDF modeling and validation of temperature and flow distribution in air conditioned space"; International Refrigeration and Air Conditioning Conference at Purdue, July 14-17, 2008.
- [4] K. Rahaman, S. Kumar; "CFD analysis of air conditioning equipment of CAD Lab for enhancing its performance"; International Journal of Recent Development in Engineering and Technology ISSN 2347 – 6435 Vol.4; November 2015.
- [5] Z. Sun, S. Wang. 2010. "A CFD-based test method for control of indoor environment and space ventilation. Building and Environment". Building and Environment 45, 1441–1447, 2010.
- [6] P. Aryala, T. Leephakpreedaa "CFD Analysis on Thermal Comfort and Energy Consumption Effected by Partitions in Air-Conditioned Building", Energy Procedia,79 183 – 188, 2015.
- [7] J. Srebric, Q. Chen, 'An example of verification, validation and reporting of indoor environment CFD analysis (RP-1133), ASHRAE Transactions, 108(2), 185-194, 2002