Computational Fluid Dynamics Analysis of a Classroom for Effective Utilization of Position of Air Conditioning System

Ashish Mogra 1, Pankaj Kumar Pandey 2, Krishna Kumar Gupta 3

1, 3 Department of Mechanical Engineering, SVKM’s, NMIMS, MPSTME, Shirpur, Maharashtra, India
3Department of Chemical Engineering, Amity University, Jaipur, Rajasthan, India
ashish.mogra@nmims.edu

Abstract. The productivity of an individual person affected to a great extent by indoor quality of air and the condition of thermal comfort. Students as well as professors spend almost half of their day in classrooms; hence the distribution of air flow from the air-conditioning systems plays a crucial role in determining whether the students will receive the proper velocity and temperature of air up to the comfortable accepted range. In present 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.

1. 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. Holiday et al. [1], 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 [2], conducted computational fluid dynamics simulation to investigate the effect of air flow inside a classroom. They selected two types of air conditioning system namely split type and centralized system. They analysed and compared the air flow of both types of system. They found better air flow distribution in centralized system as compared with the split air conditioning.

Sarkar and Mandal [3], 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 [4], did a study on CFD analysis of air conditioning of Computer Aided Design laboratory. Their result showed the air flow distribution of air conditioning system. Sun et al. [5] 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 Leepkhakpreedaa [6], performed computational fluid dynamics study to identify the thermal comfort and consumption of energy which was affected by in split unit of air conditioning in...
building. They found that in split unit level of thermal comfort decreased while consumption of energy increased by 24% in some part of building. They performed computational fluid dynamics analysis of a 4 way cassette air conditioner in office room. They also made a comparison between wall mounted typical air ventilation with 4 way cassette air conditioner. The result of numerical simulation showed higher performance of 4 way cassette air conditioner with compared to wall mounted typical air ventilation [8]. This study was performed in a classroom having an overhead air conditioning system with an aim to understand air flow and temperature distribution. The positions of the existing air conditioners were also changed to compare the effects with the existing one. For that purpose Computational Fluid Dynamics (CFD) tool has been used to perform the simulation of air flow distribution and flow pattern. This simulation will help to locate or identify the various points where air distribution is insufficient.

2. Methodology
The following assumptions are made for simulation:
1. The selected air conditioning system has been found in smooth and good operating condition.
2. There was proper arrangement of no air leaking in class room. Windows and door are assumed to be closed.
3. The temperature of atmospheric air was assumed to be constant.
4. Internal heat generation may be neglected due to less effect on properties of air.
5. All the material available in class room including tables, chair and board will be considered for simulation.
6. Atmospheric temperature and pressure take.
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: $P = P_{at_m}$
At side wall: Isothermal, $T = T_w$
At top and bottom wall: Constant heat flux, $\frac{\partial T}{\partial y} = 0$

![Figure 1 Boundary condition of Class room.](image)

**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.

| ITEM      | DIMENSION (cm) | THERMAL CONDUCTIVITY ($\frac{W}{m \cdot k}$) | 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.

**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 co-ordinates of the air-conditioners is as follows:
- Unit-1: $(x, y, z) = (2, 3.03, 2) \text{ m}$
- Unit-2: $(x, y, z) = (7.07, 3.03, 5.53) \text{ m}$
Figure 4 Line diagram of diagonally located AC in class room.

Figure 5 ANSYS model of diagonally located AC in class room.

3. 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 6 Velocity counter 1.5 m above the ground

Figure 7 Velocity counter window to door.
Figure 8 Velocity contour in center of classroom from door to window

Figure 9 shows the Velocity distribution of air in the actual classroom from a reference line drawn at 1.5 meter 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 0.2 m/s. It has been found from the literature review that the range of velocity of air for air conditioners should be in between 0.08-0.30 m/sec depending on the fan speed. Graph has been plotted between the velocity of air and width of room. The graph showed the variation in Velocity which ranges from 0.015 m/s to 0.2 m/s. This shows that velocity distribution is uneven. Maximum velocity (i.e. 0.2 m/s) lies at the point just below the two air conditioners i.e. 2m and 6m from left side wall. Minimum Velocity (i.e. 0.05 m/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 as the benches and walls are responsible for turbulent flow of air. Graphs obtained from results are also symmetric because air conditioners are placed at the same distance (2 meters) from left side and right of the wall.

Figure 10 showed the comparison of simulation result between two conditions. It has been found that the distribution of air velocity is almost constant when the position of air conditioner changed from centrally to diagonally. Figure 11 showed the comparison between the flows of velocity from window to door. This is showing almost same pattern with originally placed air conditioner.

Figure 10 Velocity comparison from left wall to right wall.

Figure 11 Velocity comparison from window to door.

4. Conclusion
Numerical simulation has been performed of a class room for determining the effect of position of air conditioner. ANYSYS fluent software has been used for simulation. Initially
the air conditioner was placed centrally and then its position was changed diagonally. It was found that the thermal comfort of the peoples are affected by position of the air conditioning unit. It was found that by placing the air conditioner diagonally the velocity of flowing air is constant when measured above the 1.5 m from the ground. It has showed no effect when placed centrally and diagonally.

Reference
[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.
[8] 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.
[9] 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.
[10] 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.
[11] 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.
[12] 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.
[13] Popovici, C. G. (2017). HVAC system functionality simulation using ANSYS-Fluent. Energy Procedia, 112, 360-365.