Investigation of air-flow distribution of an air conditioned space using Computational Fluid Dynamics (CFD) based approach

Dr. Sharad Chaudhary [1], Pankaj Singh Chouhan [2]

[1]Institute of Engineering & Technology, DAVV
[2] Institute of Engineering & Technology, DAVV

---***---

Abstract - Aim of this study is to investigate the air-flow pattern and temperature contours at different velocities in an air conditioned space using Computational Fluid Dynamics (CFD) approach. A computer lab is used as the space under consideration for the research purpose. Analysis is performed using the software ANSYS-Fluent. The results are presented as variation for the velocity and temperature at different planes and a comparative analysis is performed between different inlet velocities.

Key Words: Computational Fluid Dynamics (CFD) simulation, Air conditioner, Air flow pattern, Thermal Comfort

1. INTRODUCTION

Room air distribution gives the proper picture that how air is introduced into the system, how it flows through the room and how it is removed from the room. In the flow-field air-flow patterns, air velocity distribution and temperature distribution are essential information to understand the performance of an air conditioning system.

After getting the air velocity contours and temperature contours it is easy to estimate the correct fan speed, dimensioning of inlet and outlet duct, location of system and various other parameters which are best suitable for the thermal comfort of the occupants. For this purposes experimental studies are expansive and time consuming.

In this paper CFD simulation approach is implemented to identify the air velocity and temperature distribution by providing three different inlet velocity.

A computer lab with occupancy of 30 member and three window Air-conditioner (AC) units was chosen for the analyses. The dimension of room is 9.5 m x7.6 m x3.3 m (LxBxH). Air conditioning systems are fixed at height of 0.9m from the ground surface on 9.5 m long walls.

The CFD tool employs a very simple rule of discritization of whole system in small grids. Then governing equations were applied on these elements to get numerical solutions concerning flow parameters, pressure, temperature in less time and at reasonable cost because of reduced required experimental work [12].

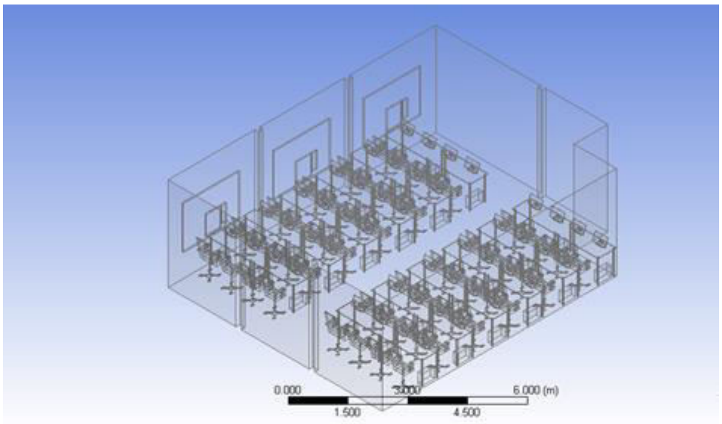


Fig.1

2. Body of Paper

2.1 CFD Simulation

The physical model of the computer lab for which the analysis has been performed is shown in Fig.1 the geometry of the test room, furniture used in the room is created using CREO parametric and assembled in the design modeler using ANSYS.

2.2 Boundary Conditions

1) Air inlet temperature is kept 288K.

2) Air inlet velocity is kept 2.5m/s, 5m/s, 7.5m/s in three different cases respectively.

3) Back flow temperature is kept at 313K.

2.3 Turbulence Sub model

For turbulent flow, a k-Ɛ model was considered in ANSYS. The k-Ɛ model is based on model transport equations for the turbulence kinetic energy(k) and its dissipation rate(Ɛ)[11]. Developed physical model of the room was meshed using ANSYS workbench contains 3390728 elements and 605888 nodes.

ANSYS-Fluent 19.0 is used in the analysis which uses Finite volume Method (FVM) to convert the governing equation in to numerically solvable algebraic equations.

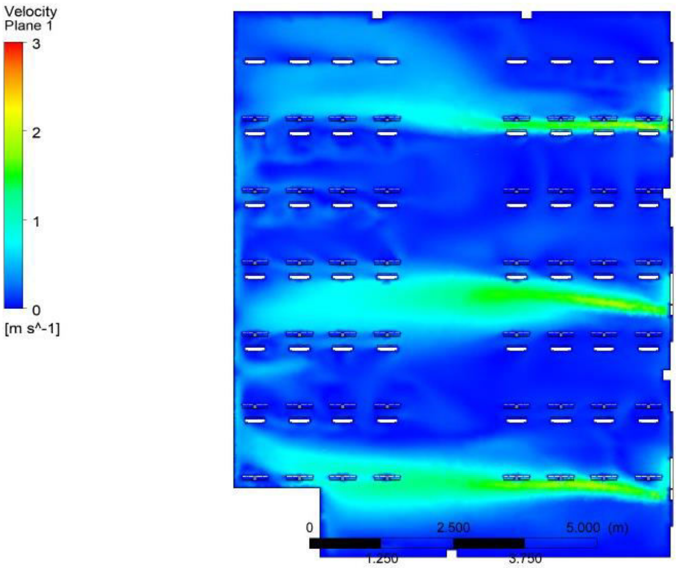
3. Results and discussion

The idea behind this work was to analyse the variation of velocity and temperature inside the room by providing different inlet velocity. Simulation was carried out at 2000 iteration using tool ANSYS-Fluent.

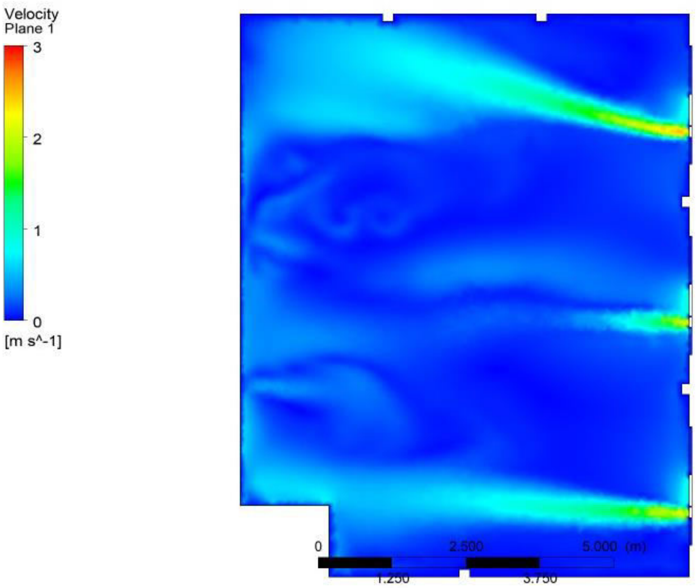
Case 1: Inlet velocity in all three AC is 2.5 m/s.

The initial temperature in the room was kept 300K and the back flow temperature was kept 313K. The temperature at which air is entered into the room is 288K. The result obtained in the terms of velocity and temperature contour at different plane, It can be seen in the in the Fig. 3 that effect of AC is not reaching on the Z plane at height of 0.5m from ground surface. The velocity at plane 1m, 1.5m, 2m are also not up to the requirement of human comfort.

1. **Velocity contours**



Y = 1 m



Y=1.5

Fig.2 Velocity contour in (X-Z) at Y=1 m and Y=1.5 m at V=2.5 m/s

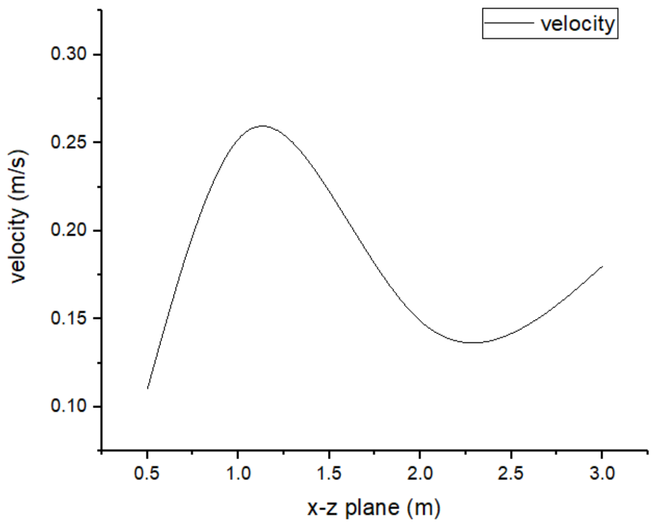
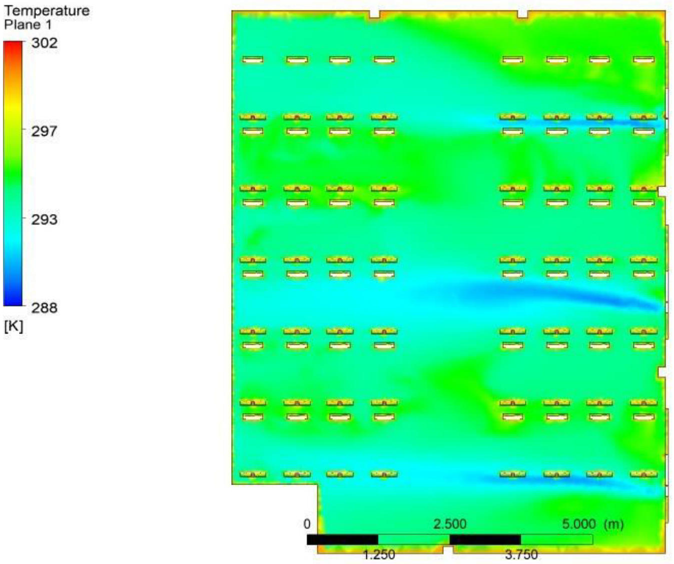


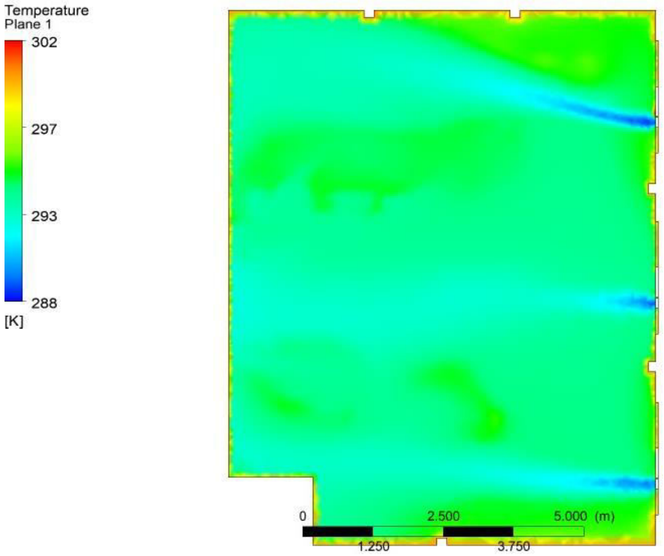
Fig.3 Velocity variation at different (X- Z) plane at V=2.5m/s

In this case the average velocity of room is .1639 m/s.

2. Temperature contour



Y=1



Y=1.5

Fig.4 Temperature contour in (X-Z) plane at Y=1 m and Y= 1.5 m at V=2.5 m/s

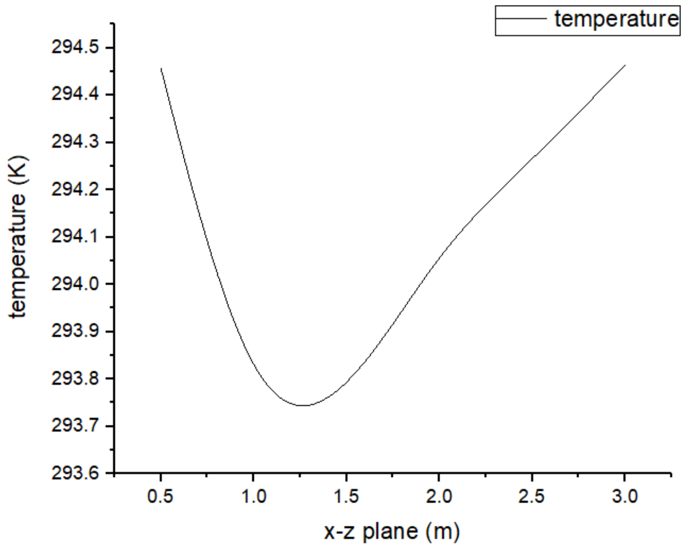
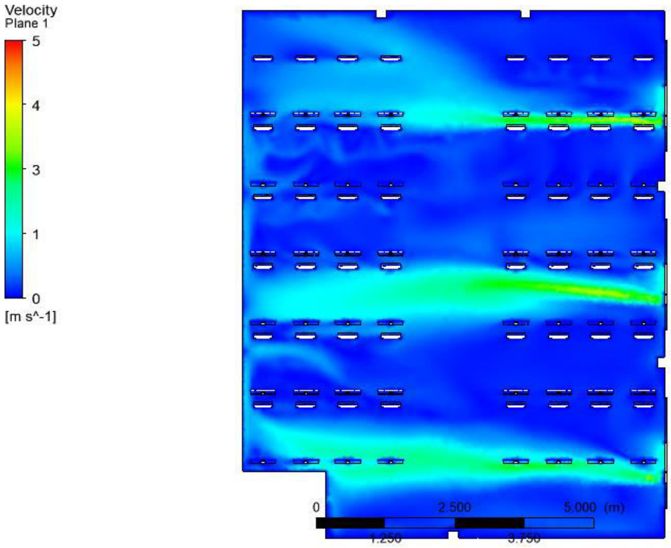


Fig.5 Temperature variation at different (X-Z) plane at V= 2.5 m/s

Average temperature of the room is 294.336 K.

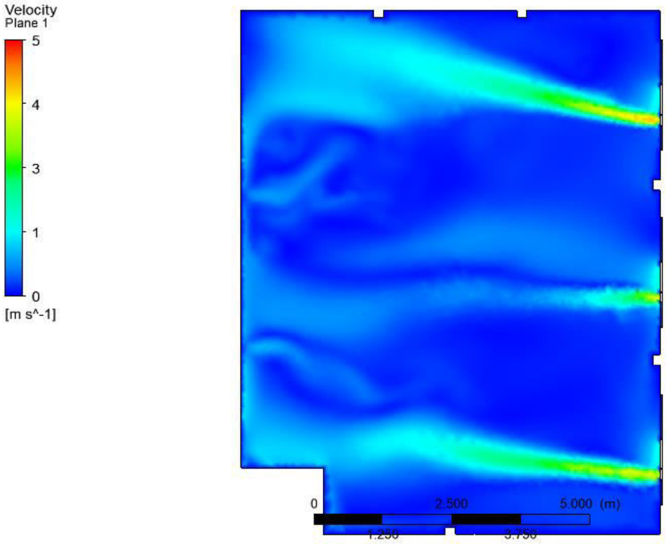
Case 2: Inlet velocity in all three AC is 5m/s.

1. Velocity contour



Y= 1 m

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 04 Issue: 09 | Sept -2020 ISSN: 2582-3930



Y= 1.5 m

Fig.6 Velocity contour in (X-Z) at Y=1 m and Y=1.5 m at V= 5 m/s

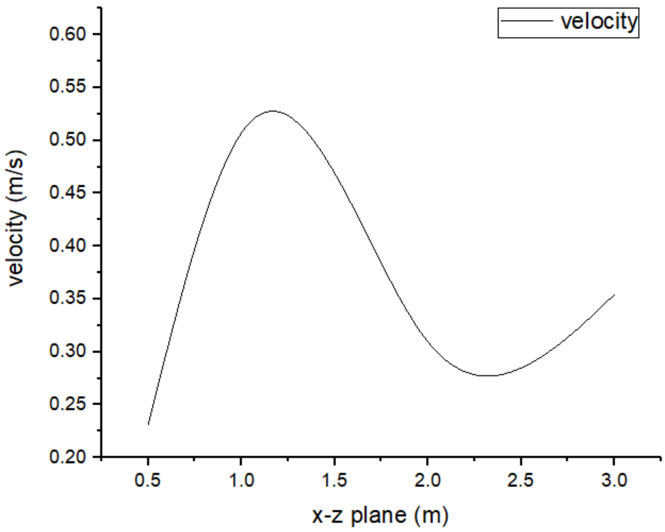
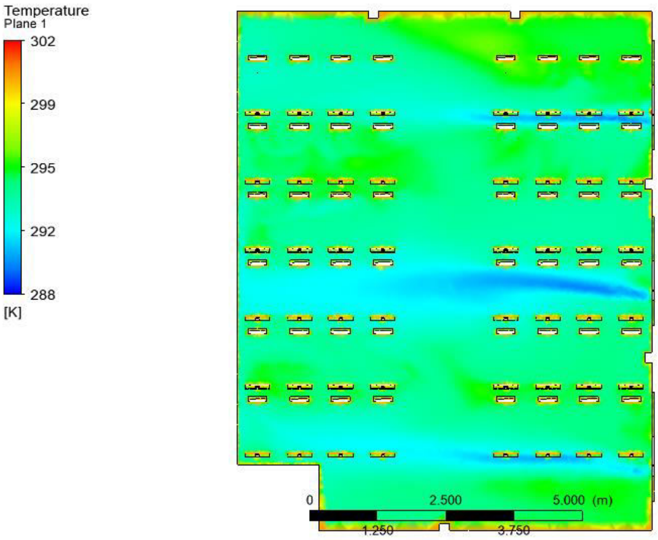


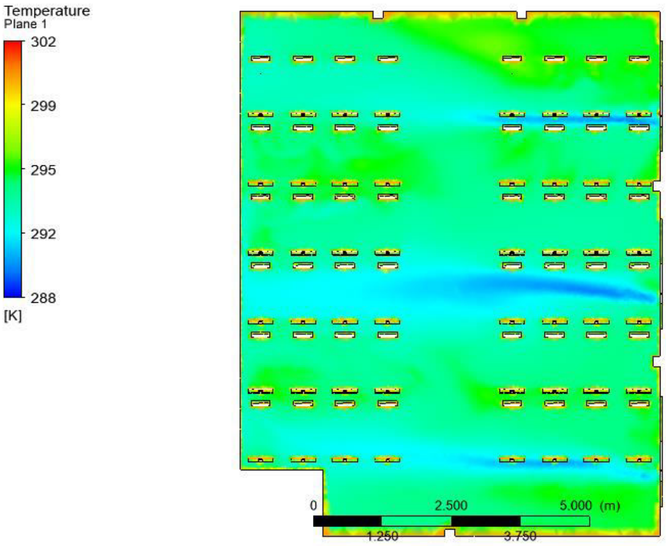
Fig.7 Velocity variation at different (X- Z) plane at V= 5m/s

In this case the average velocity of room is .3388 m/s.

2. temperature contour



Y = 1 m



Y = 1.5 m

Fig.8 Temperature contour in (X-Z) plane at Y=1 m and Y= 1.5 m at V= 5 m/s

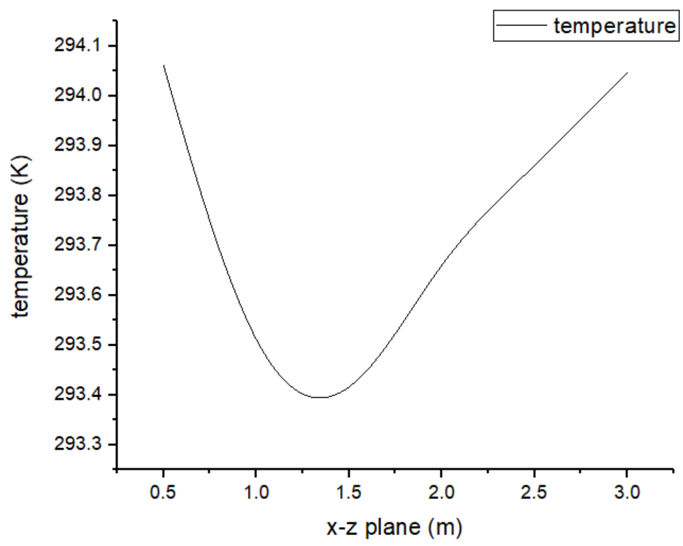
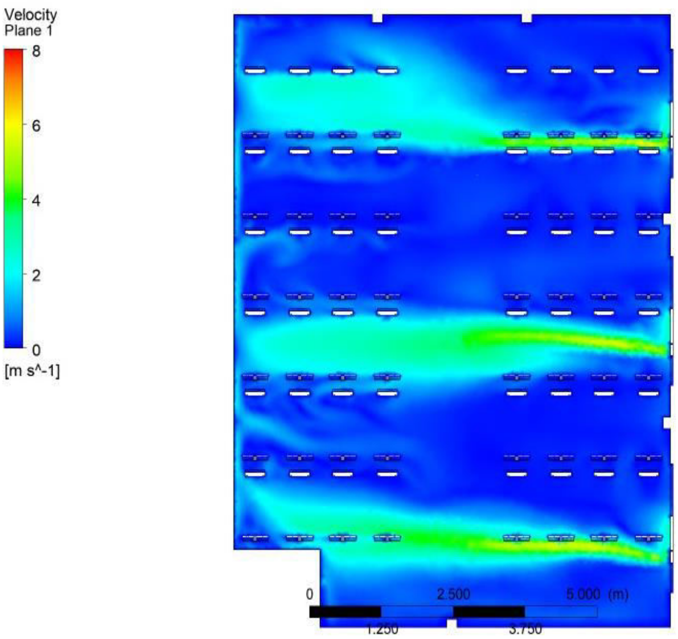


Fig.9 Temperature variation at different (X-Z) plane at V= 5 m/s

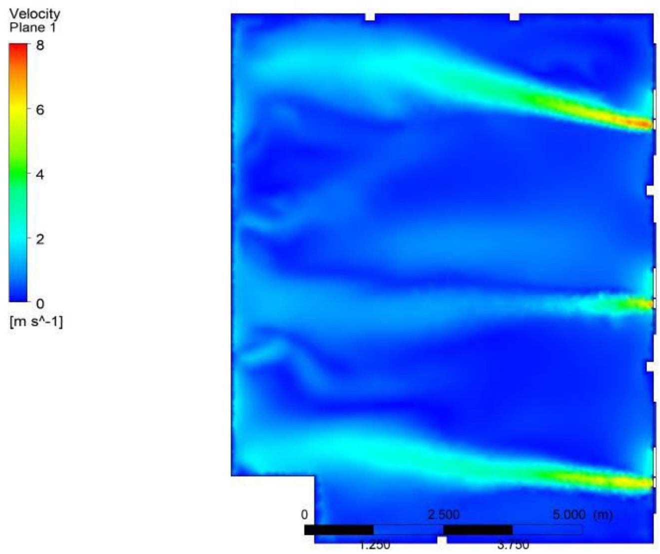
Average temperature of the room is 293.946 K.

Case 3: inlet velocity in all three AC is 7.5m/s.

1. velocity contour



Y = 1m



Y = 1.5 m

Fig.10 Velocity contour in (X-Z) at Y=1 m and Y=1.5 m at V= 7.5 m/s

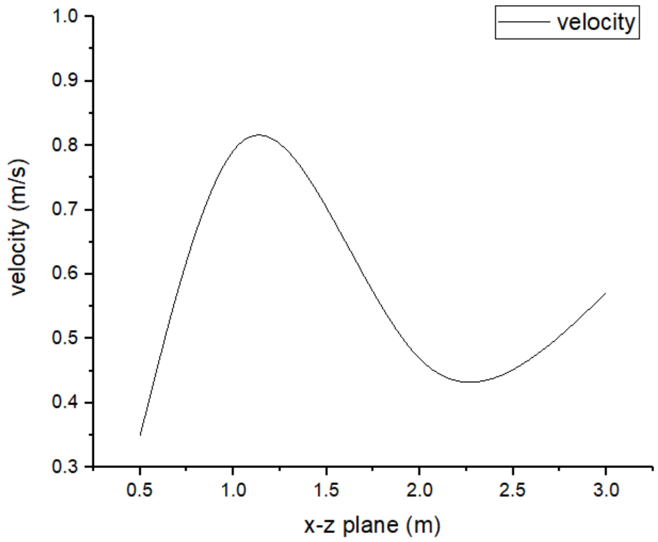
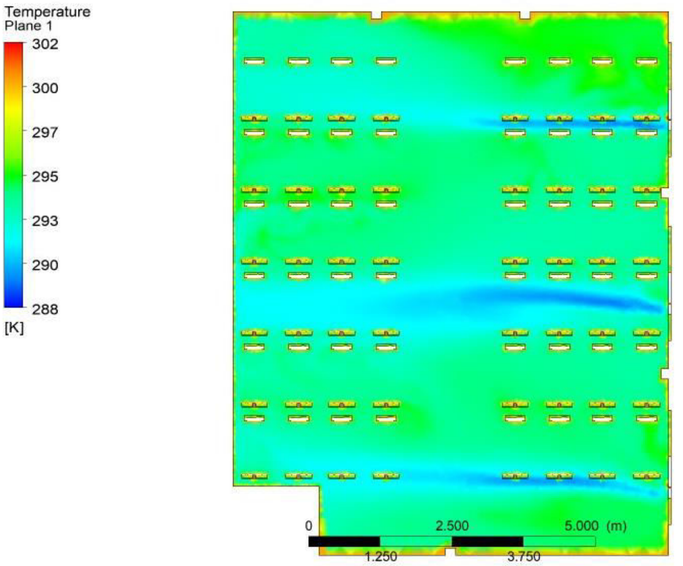


Fig.11 Velocity variation at different (X- Z) plane at V= 7.5m/s

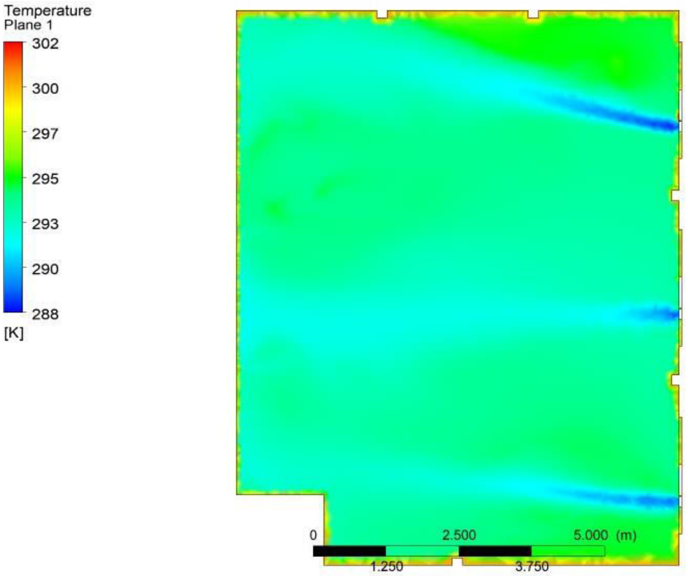
In this case the average velocity of room is .5217 m/s.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 04 Issue: 09 | Sept -2020 ISSN: 2582-3930

2. Temperature contour



Y = 1 m



Y = 1.5 m

Fig.12 Temperature contour in (X-Z) plane at Y=1 m and Y= 1.5 m at V= 7.5 m/s

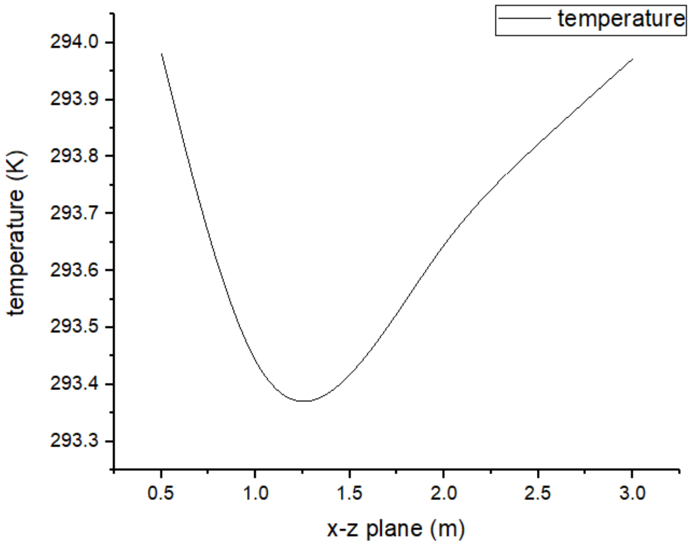


Fig.13 Temperature variation at different (X-Z) plane at V= 7.5 m/s

Average temperature of the room is 293.867 K.

4. Comparative analysis

A comparative analysis has been performed for the obtained average velocity and temperature at different plane in the room for all the three cases.

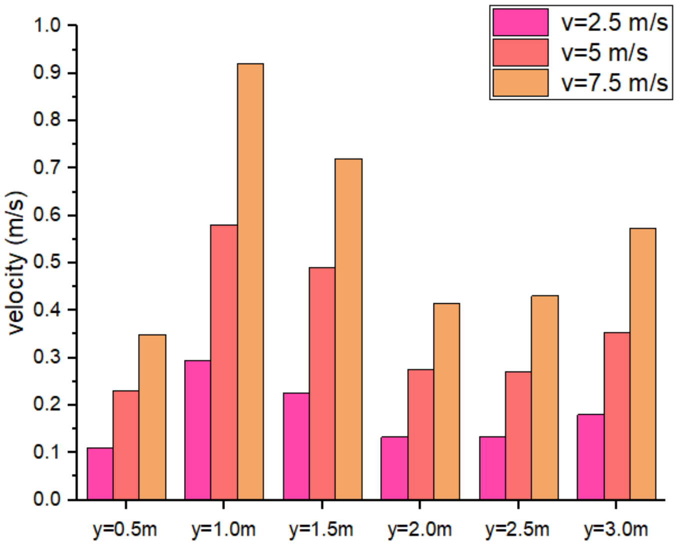


Fig.14 Comparative analysis of average velocity at different inlet velocity at different plane

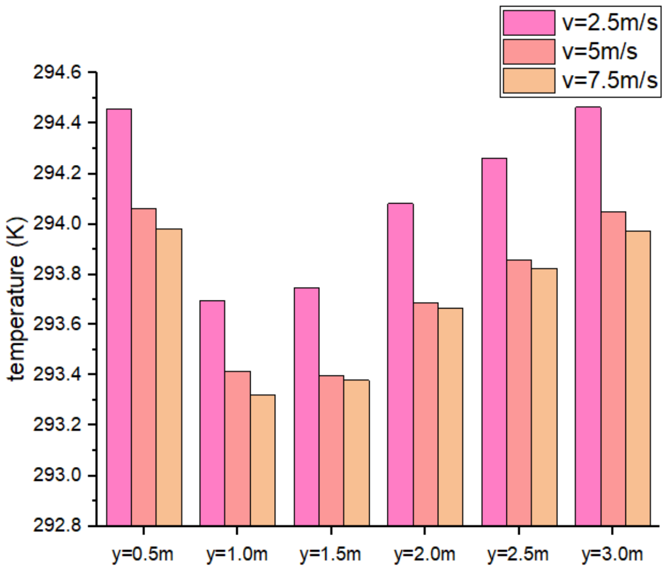


Fig.15 Comparative analysis of average temperature at different inlet velocity at different plane

5. CONCLUSIONS

The air-flow pattern in an AC space is an important factor for the achievement of satisfactory thermal comfort in the occupied zone of the room. This study aimed to simulate those flow patterns of an AC system in a computer lab under different inlet velocity condition while other parameters keeping constant.

Following conclusion are drawn from this simulation work.

[1] The average velocity obtained in the room for the inlet velocity of 5 m/s is 0.3388 m/s which is close to as per the requirement for human comfort.
[2] As higher the inlet velocity of AC, maximum will be the temperature drop in the room.

Simulation results obtained using tool ANSYS-Fluent.

6. REFERENCES

[1] Zhilin huang, Kaijun Dong, Qin Sun, (2017), “Numerical simulation and comparative analysis of different air flow distribution in data centers”, Procedia Engineering 205, 2378-2385.

[2] Cong Zheng, Jing liu, (2015), “Study on indoor air movment under the crowding condition by numerical simulation and particle image velocimetery small scale experiments”, Procedia Engineering 121, 1650-1656.

[3] Catalin George, (2017), “HVAC system functionality simulation using ANSYS-Fluent”, Energy Procedia, 360-365.

[4] Ahmed A, Ehab M. Mina, (2018), “Studying comfort in a room with cold air system using computational fluid dynamics”, Ain shams Engineering Journal 9 1753-1762.

[5] Z. Wiercinski, A. Siepsiak, (July 2008), “Application of CFD for temperature and air velocity distribution calculation in a ventilated room”, Institute of fluid flow machinery Poland.

[6] Abento ,J. Barreo, D. Reggio, M. Ozell, (2004), “Air flow modeling in a computer room , Building and environment”,vol.39, no.12:p.1393-1402.

[7] J. Sarkar, M Soumen, (2008), “CFD modeling and validation of temperature and flow distribution in air conditioned space”, International Refrigeration and Air conditioning conference, paper 972.

[8] Emmerich, S.J, (1997), “Use of CFD to analyze indoor air quality issues”, National institute of standards and technology, USA.

[9] Fan, Y. Zen, (1995), “CFD modeling of the air and contaminant distribution in rooms”, Energy and Building, vol.23, no.1:p.33-39.

[10] M Ali, Q Hasan, (2014), “Effect of workers and computers on the indoor air flow in office room”, Al-Rafidain university, issue no. 33/2014.

[11] N. Bansal, M. Sodha, S. Singh, A. Sharma, A.Kumar, (1985), “Evaluation of an earth air tunnel system for cooling/heating of a hospital complex”, Build. Environ, 20, 115-122.

[12] Rahul Khatri, A. Singh, V. Khare, (2017), “Identification of ideal air temperature distribution using different location for Air Conditioner in a room integrated with EATHE-A CFD based

approach", Energy Procedia 109, 11-17.

- *Dr. Sharad Chaudhary, Assistant Professor, Department of Mechanical Engineering, Institute of Engineering and Technology- Devi Ahilya Vishwavidyalaya, Indore (M.P.), India*
- *Pankaj Singh Chouhan is currently pursuing master degree program in Design and thermal in Department of Mechanical Engineering, Institute of Engineering and Technology- Devi Ahilya Vishwavidyalaya, Indore (M.P.), India.*