OPTIMIZATION OF INDOOR AIR CIRCULATION OF CLASSROOM USING CFD.

[1]B Vishal Eshwar Reddy ,[2]Adarsh S,[3]Akshay M, [4]Abhishek P Inamdar,[5] L R Jagadeesh

[1,2,3,4] Students [5] Faculty

School of Mechanical Engineering,

REVA University, Bengaluru, India

Abstract: India is a sub-continent where temperature ranges about 35°C-40°C. In summer the temperature shoots up to 40°C in many regions. Most of the technical institutions are having the classes during the summer. A considerable amount of heat is generated from the students and also heat transfer to the class room from solar radiation and other sources which will increases the dependency on electrical fan which are available in the class rooms. There is always a need for a proper ventilation system for indoor air circulation and to remove the heat from the class rooms. Therefore one of the class rooms is selected for study and analysis of temperature and air flow using CFD technique. It is found that ventilation is not sufficient to provide comfortable environment. After providing additional ventilation the temperature and air distribution with in comfortable condition which results in saving of power by running fans for minimum time with minimum speed.

Index Terms **- CFD, Natural convection, Air circulation, Air flow pattern, CFD Modeling.**

I. INTRODUCTION

Day by day environmental temperature is rising due to deforestation, more industrial activities and power generation which results in global warming. To get comfort environment electrical fans or air conditioners are preferred than natural ventilation system. These devices consuming electricity which will be responsible for ozone layer depletion and global warming. It is necessary to reduce the dependency on theses device by providing natural ventilation system. A class room capacity of 60 people is considered for this work to study and analysis of temperature and air distribution in the class room during extreme weather. CFD [1,2,3] is a one of the tool for simulation of fluids for different conditions. From the literature it is found that air flow circulation [4, 5, 6] and indoor air temperature can be studied using CFD and results are comparable with experimental results. In this study a class room is selected for study of temperature and air distribution due to mechanical ventilation.

II. PROBLEM FORMULATION

In the current design of buildings, natural ventilation is most important to maintain uniform temperature for indoor and outdoor atmosphere, especially for schools, colleges, auditoriums, concert halls, etc. Proper ventilation design, the positioning of mechanical vents like windows and air circulation within the environment plays an important role in designing the natural ventilation system.

For the study purpose a class room of sixty seating capacity has been identified where the mechanical ventilation fails to provide comfortable condition. The existing class room is in final floor of the building where it is exposed to solar radiation. The class room having four no fans, four fluorescent lamps and two glass windows exposed to west side and mechanical ventilation exposed to East side. From the experimental work it is found that temperature rises to 37°C which is uncomfortable for the students during class hours.

III. Methodology

To investigate the temperature and air circulation in the class room it is necessary to measure the temperature and air flow rate at different environmental condition and simulate using CFD and validate the results. Hence the following steps are adopted.

- Collection of class room details
- Measurement of temperature at different locations in the class rooms
- Development of prototype of class room for study of air circulation
- Development CAD model of class room
- Simulation using CFD.
- Validation of results.

IV. Experimental work

A class room in the final floor of the building has been selected for the study. The details of the class room are

Size of the class room:10*7.336*3.048 m^3 Number of Benches: 15

Seating Capacity: 60 students Grill size: 10*1.041.4 m^3

Window size: 10.6*.01*1.2 m^3

Room temperature was measured for various days and at different period of time with the help of dry bulb thermometer. A model prototype of the classroom was developed with medium density fibre board (MDF) for analysing the air flow pattern inside the classroom with bulb as heat source and smoke as the medium for visibility of airflow pattern.

Fig-1: Photo of the Classroom

Fig-2: Prototype of the Classroom

V. Simulation using CFD

A CAD model of class room was developed and meshing had been done using ANSYS Fluent tool. At first it was coarse meshed, later gradually increased the mesh size and it was found that when the number of elements exceeded above 100000 units the results were found to be accurate when compared with the experimental data. i.e., at any cross-section of the classroom the temperature difference from one end to the opposite end was found to be similar to that of analysed result.

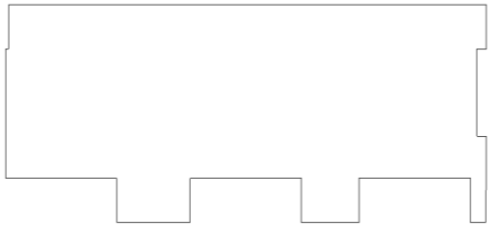

Fig-3: 2-D Model of the Classroom.

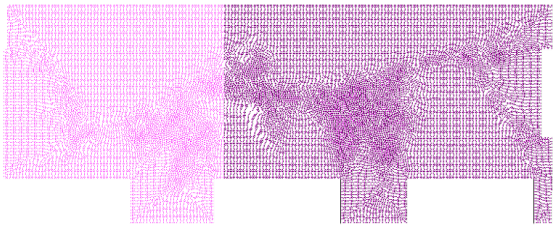

Fig-4: Meshing of the 2-D Geometry (Number of nodes: 166495)

Simulation was also done for additional mechanical vent is provided with existing mechanical vent. The figure 5 shows the size of the vent recommended.

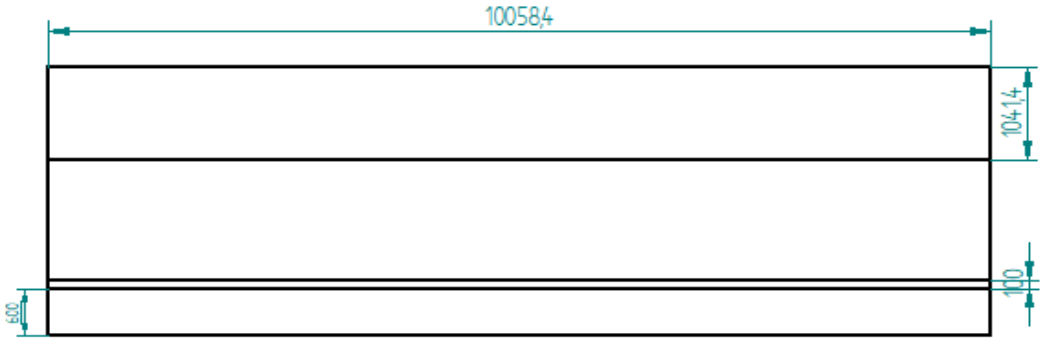

Fig-5: Elevation of the Classroom with Vent

During analysis boundary conditions have to be specified, hence the following assumptions had been made to make the model with less complexity.

- The flow is ideal gas flow.
- Solar radiation only through roof not from side walls
- No obstacles in the room.

Simulation had been done for four different cases using CFD tool CFX

Case – 1: Windows open; at 27°C.
Case – 2: Windows open; at 35°C.
Case – 3: Windows open with additional vent at 27°C.
Case – 4: Windows open with additional vent at 35°C.

VI. Results and discussions

Simulation was done using ansys CFX for applying various boundary conditions such as a windows opened for with ambient conditions of 27^0C and 35^0C. The simulation results of temperature variation and air circulation are obtained

Case – 1: Windows open; at 27°C.
Inlet conditions:

- Windows: Pressure-inlet condition at 300K.

Outlet conditions:

- Grill: Pressure- outlet condition at 300K.

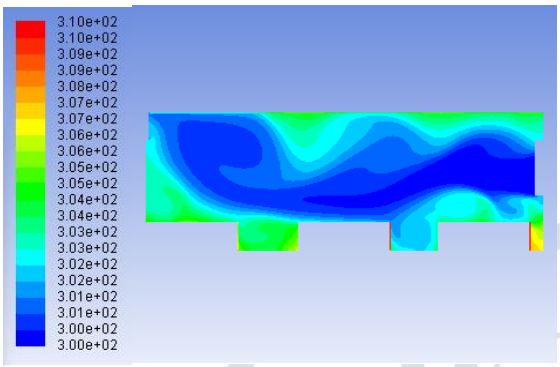

Fig-6: Temperature Contour Plot

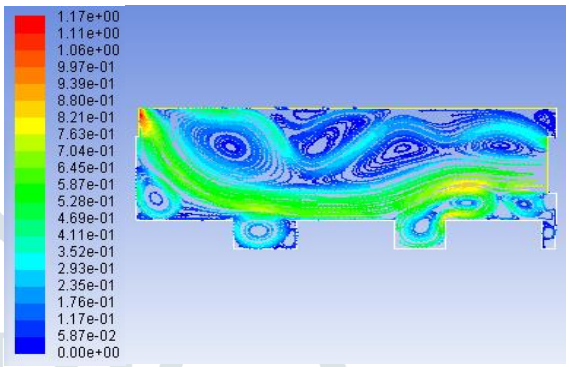

Fig-7: Path lines of Velocity Magnitude

The above figures shows temperature and air circulation in the class room when the window is kept open at ambient temperature of 27^0C. From the Fig 7 it can be seen that the air is getting re-circulated near the heat source. Near the window the air temperature is ambient temperature but as it progress towards the grill the temperature increased gradually. From Fig-6 it is clear that the heat is getting accumulated towards the lower part of the wall where grill is placed. From both Fig-6 and Fig-7 it is found that towards left the stagnation zone where temperature increases and the velocity is reducing to the negligible value. The temperature is about 31^0C-32^0C

Case – 2: Windows open; at 35°C.
Inlet conditions:

- Windows: Pressure-inlet condition at 308K.

Outlet conditions:

- Grill: Pressure- outlet condition at 308K.

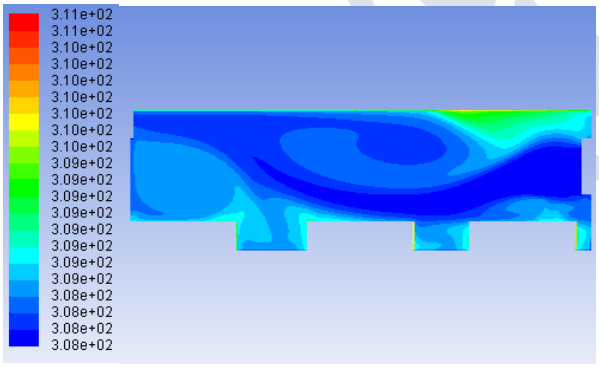

Fig-8: Temperature Contour Plot

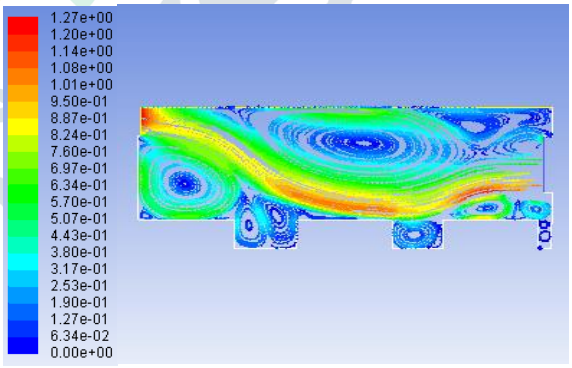

Fig-9: Path lines of Velocity Magnitude

This case is exactly similar to that of the first case but the ambient temperature is 308K, which was found to be high at the place of experimenting. In Fig-8 is clearly visible that the overall temperature inside the classroom is maintained approximately equal to ambient temperature because the difference between the temperature of heat source and the ambient temperature is very less i.e., 2^0C.Velocity is maintained sufficiently with the present conditions except of a stagnant spot i.e. near the wall towards the grill.

Case – 3: Windows open with vent open at 27°C.
Inlet conditions:

- Windows: Pressure-inlet condition at 300K.

Outlet conditions:

- Grill: Pressure- outlet condition at 300K.

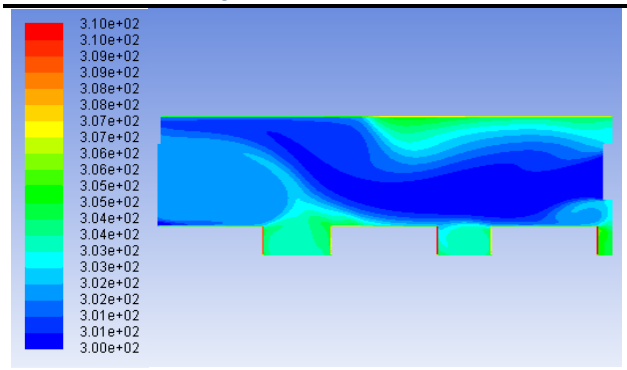

Fig-10: Temperature Contour Plot

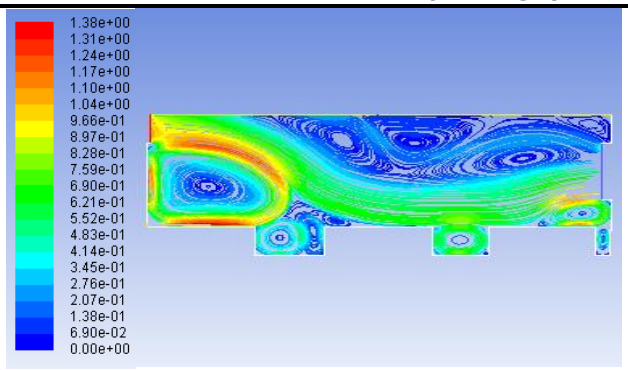

Fig-11: Path lines of Velocity Magnitude

The above figure shows the temperature and air flow circulation for outside temperature of 27^0C. After providing additional vent a small increase in temperature by 2^0C to 3^0C from room temperature of 27^0C. In case 1 with same outside temperature of 27^0C results in increase of temperature up to 32^0C due to recirculation of air. Additional vent improves the circulation of air so that there is temperature drop by 5^0C-6^0C.

Case – 4: Windows open with Vent at 35°C.

Inlet conditions:

- Windows: Pressure-inlet condition at 308K.

Outlet conditions:

- Grill: Pressure- outlet condition at 308K.

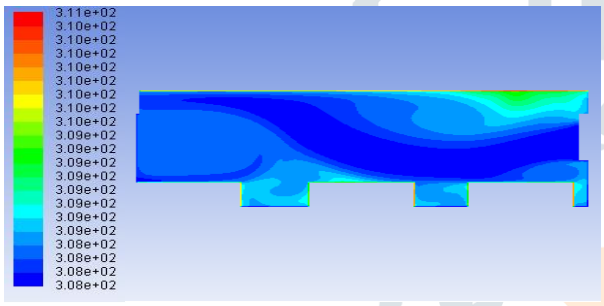

Fig-12: Temperature Contour Plot

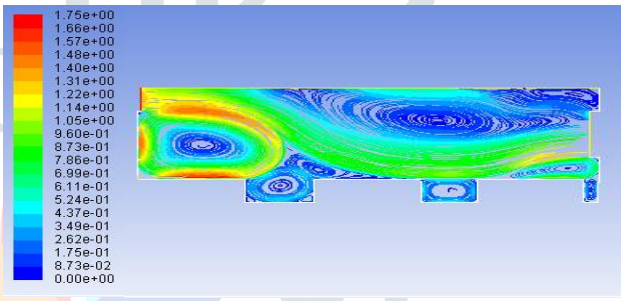

Fig-13: Path lines of Velocity Magnitude

After providing the vent the temperature is same as the ambient temperature of 35^0C in many locations of the class room. Without this vent the temperature rises by 2^0C i,e 37^0C(case-2) which is more uncomfortable for the students.

VII. Conclusion

- The temperature is rising in the classroom at present condition due to inadequate ventilation to the range of 33°C while the ambient temperature is 27°C.
- Due to improper ventilation, the air is re-circulated at certain points of the classroom, which is resulting in an increase of temperature at that particular point up to a maximum of 35^0C.
- To reduce the temperature additional vent is recommended. After providing the vent it has been found an effective drop in the temperature by 4^0C to 5^0C. As well as the recirculation of air has reduced considerably.

References

[1]H Yang, H Zhang, S You and P Wang "CFD simulation and study on office thermal environment" School of Environmental Science& Engineering, Tianjin University, Tianjin, 300072, China.

[2] S. Gilani, H. Montazeri, B. Blocken "CFD simulation of the stratified indoor environment in displacement ventilation:Validation and sensitivity analysis" Accepted for publication in Building & Environment, 9 September 2015

[3] Bjarne Bjerg, Guo-Qiang Zhang and Peter Kai" CFD Analyses of Methods to Improve Air Quality and Efficiency of Air Cleaning in Pig Production"

[4] T. P. Ashok Babu, G. S. Sriram, Aneesh S. Vadvadgi, Ravindra Siddeshwar, "Air Flow Modeling in a Mechanically Ventilated Room" International Refrigeration and Air Conditioning Conference, School of Mechanical Engineering 2008.

[5] Manoj Arya, Rajput.S.P.S "Monitoring and analysis of indoor air quality at different heights in the industrial room by using CFD" INTERNATIONAL JOURNAL OF ENVIRONMENTAL SCIENCES Volume 1, No 6, 2011 ISSN 0976 – 4402.

[6] D. Prakash and P. Ravikumar " Simulation of Indoor Air Flow for a Room with Windows at their Adjacent Walls under various Wind Flow Direction using CFD" ARPN Journal of Engineering and Applied Sciences VOL. 7, NO. 11, NOVEMBER 2012 ISSN 1819-6608.