Analysis of Lecturing Room using Computational Fluid Dynamics
Natural and Forced Ventilation

1,2 Department of Mechanical Engineering,
BTL institution of Technology and Management, Bangalore, 560099, India

Abstract--Investigating the main parameters contrite to the air quality and thermal comfort has been highlighted by different studies. Doing so will provide the suitable measure for environmental quality that receive considerable attention by building designers. It comes along with the different international and regional standards intended to foster environments that are acceptable to occupants. Although there is considerable field data on thermal comfort, there is far less data that assesses the level of thermal comfort in universities. This has not been investigated in the literature, therefore, this study aims at simulating the ventilations of lecturing room and determining the number of people allow for a sufficient air conditioning.

Computer simulations offer an exciting opportunity for validating aspects of ventilation, smoke movement, natural air flow and thermal comfort right at the design stage. In this work CFD simulation of natural and force ventilation on lecturing room was carried and thermal comfort of the human being is studied using ANSYS Fluent. Results shows that for 234 human load case the average Temperature contours in lecturing room is approximately 301K (28 C) which is higher than comfort temperature of 296K (23 C) due to the flow stagnant zone back side of the lecturing room. So the human load is reduced to 198 to achieve the comfort zone for human being of 296 K (23C). Then the natural ventilation is simulated. It shows average temperature of the flow stagnant zone back side of the lecturing room is approximately 305 K (32 C) for 198 human case, also shows that some areas of peak temperature of 308 K due to stagnant region. Finally smoke simulation is carried out from front door stack releases smoke that is dispersed into the lecturing room with an oncoming side wind of 1 m/s. the total time required to evacuate the smoke is approximately 3 min.

I. INTRODUCTION

It is necessary to provide buildings with adequate ventilation to ensure both removal of stale air and the supply of fresh air for occupants. This can be provided in different ways, which include: mechanical ventilation, in which fans and ducts are used to move large volumes of air with or without heating the air; air-conditioning, in which the temperature and humidity of air, supplied via fans and ducts, is fully controlled; and natural ventilation which harnesses the naturally occurring driving forces of wind and buoyancy. It is also possible to use a hybrid approach which uses both natural forces and mechanical means (usually fans). These are known as mixed-mode systems.

The main disadvantages of air-conditioning are: cost, in terms of capital, running costs and maintenance; and the large amount of space required to house the necessary equipment. Furthermore, there is evidence to suggest that air-conditioning systems are more likely to cause occupant health complaints than natural or mechanical ventilation.

Given today’s efforts to reduce energy usage and CO2 emissions, there is an increasing trend to move away from air-conditioned buildings. Consequently, many architects and building designers have turned their attention towards naturally ventilated or mixed mode techniques. Although such buildings use innovative techniques, many have been completed with a high degree of success. Due to the non-predeterminate nature of natural ventilation systems, much care is needed at the design stage to ensure that the ventilation strategy will perform successfully under foreseeable climatic and occupancy scenarios. In particular that acceptable air change rates will be obtained. Predictions of such parameters are much easier in a mechanically-driven system since the designer knows, to a high degree of accuracy, the volume flow rates produced by the various components that make up the system. When designing natural ventilation strategies however, the only way to predict the flow rates is by means of either a physical or mathematical model.

Since the advent of more powerful, affordable, desk-top computers, another tool has become accessible to designers - Computational Fluid Dynamics (CFD). This technique considers the airflow in a space by dividing the space into small cells and solving the equations which govern the airflow and temperature distribution in each one. This offers the immediate advantage over the salt bath modeling approach of being able to provide information about the flow at many positions throughout the domain of interest. It also enables changes to the geometry and operating conditions to be made more easily, offering an ideal tool for investigating many ventilation options early in the design process.

M. Zajicek et.al [6] is focused on the numerical analysis of ventilation of building for broilers during the summer period
Numerical computations were performed using commercially available software package Fluent developed by ANSYS.

Frédéric Conte et al. [9] studied "Early detection in rooms with high ceilings." This project consists of the study of smoke production, smoke detectors, and two problems detecting fires and will be finished by full-scale experiments.

Jelena Srebric et al. [10] demonstrated how to use two simplified methods, the box and momentum methods, to simulate complex diffusers in room airflow modeling by computational fluid dynamic.

The main aims of the research were as follows:

- to evaluate the accuracy of CFD for modeling buoyancy-driven displacement ventilation flows by considering the flows in simple geometries and comparing the results with experimental and analytical predictions of bulk airflow patterns, stratification, and vertical temperature gradients;
- to provide guidance on the use and reliability of CFD techniques for modeling buoyancy-driven flows.
- CFD simulation lecturing hall using ANSYS Fluent-Natural ventilation
- CFD simulation lecturing hall using ANSYS Fluent-Forced ventilation
- CFD Simulation smoke evaluation in lecturing hall

II. FORCED CONVECTION VALIDATION STUDY

A well documented forced convection benchmark study by Nielsen et al. [4][5][1978] is used herein to validate CFD flow field predictions. Geometrical arrangement used in the experiment is presented in Figure 3.1, where H=3m (height in meters). The experiment was conducted in a room with a single slot inlet/single slot outlet configuration. The inlet velocity distribution was a uniform velocity profile with a magnitude of 0,455 m/s (meters/second). The inlet and outlet heights were 0,168 m and 0,48 m respectively. The air velocity was measured by a laser Doppler anemometer along the centerlines of the four cross sections as shown in Figure 2, namely x/H = 1, x/H = 2, y/H = 0.028, y/H = 0.972.
A. 2.1 Geometric Model of the building

Geometries of the model created Top-down approach due to simple geometry. In this approach, the computational domain is created shown in Fig.3. Geometry files are then transferred into imported into ANSYS Meshing to create CFD Mesh. The geometric model of the lecturing room is shown in Fig.4. It shows blue color with 15 inlet supply duct location and red color with 6 outlet return duct. The amount of supplying air is 12000 CFM or 604000 BTU, air speed supply duct 2 m/s, temperature supply 17°C and temperature set 23°C. Outside temp 28°C, 15% return air ventilation. There are 15 supply duct and 6 return duct.

Boundary Conditions
CFD problems are defined in terms of boundary conditions, and it is important to specify them correctly. The material that initially fills the entire solution domain was selected as the air at 23°C, 1atm and treated as incompressible fluid. The ambient conditions were set to be 28°C with 10 W/m²K and 0 Pa relative to 1atm. Walls and ceiling boundaries within the domain were of type “Intermediate” and modeled as adiabatic on both sides

While compact fluorescent lights are relatively new, straight-tube fluorescent lights have been around since the 1930s. Fluorescent lights have been used widely in commercial and industrial buildings, largely because of their dramatic energy
savings over incandescent lights and the fact that they typically last ten to twenty times as long is [4]. Fig.5 shows the Florence Light Heat Source red color, here each Florence light is modeled with 50 W applied as heat-flux boundary conditions selected based on Fig.3.8. Then total heat applied is $80 \times 2 \times 50 = 8000$ W.

Fig.6 shows 30 Globe Light-Heat Load locations. in ANSYS it is applied as point source $W/m^3$. Similarly From the Fig.3.8 heat load for the globe light has taken has $75$ W. So to total heat load applied is $30 \times 100 = 3000W$.

Total 75 W is applied for each human. In this analysis room has 234 chairs, so total $234 \times 75 = 17550$ W which is shown as point source in Fig.3.18.

III. RESULTS AND DISCUSSIONS

1. Validation study

Sectional view of the flow field solution obtained using CFD is presented in Figure 8, while velocity distributions along the X-direction taken through the middle of the room are shown in Figure 9. Solution contours represent flow velocity component computed along the X axis and are colored by its magnitude. As expected, the flow velocity magnitude diminishes upon incoming airflow propagating into the main room domain, from right to left in Fig.10. Airflow forms a large recirculation pattern inside the room, with the bulk-entering airflow moving from right to left along the ceiling, turning towards the floor, partially exiting the room at the outlet and partially proceeding from left to right along the
floor completing the recirculation pattern by turning upwards at the right lower corner of the solution domain. An important characteristic of the flow confirmed by experimental results includes two secondary recirculation regions, one in the upper left and one in the lower right corners of solution domain. Normalized experimental and numerical results are shown in Figure 3.7 for the two measured cross sections. All numerical solutions show close adherence to experimental measurements.

**CASE1: - Simulation of the ventilation for a lecturing room with 234 humans**

The predicted velocity distribution is shown in Figure 11 for three inlet planes. In this case, the flow contains a free jet in the inlet region and downstream a wall-affected area.

**Fig. 11. Velocity contours on inlet plane**

The temperature contours on inlet plane is shown in Figure 12. It shows average temperature of the flow stagnant zone back side of the lecturing room is approximately 301K (28°C) which is higher than comfort temperature of 296K (23°C).

**Fig. 12. Temperature contours on inlet plane**

**CASE2: - Simulation of the ventilation for a lecturing room with 198 humans**

Based on earlier results, the number of people allow to ensure a sufficient air conditioning is reduced to 198 means last two row of human loading is not considered for the simulations. The comparisons of temperature contours on inlet plane is shown in Figure 13. It shows average temperature of the flow stagnant zone back side of the lecturing room is reduced for 198 human load approximately 296K (2°C) which is equal to the comfort temperature of 296K (23°C).

**Fig. 13. Comparisons Temperature contours on inlet plane**

The temperature contours on horizontal plane is shown in Figure 14. It shows average temperature of the flow stagnant zone back side of the lecturing room is approximately 301K (28°C) for 234 human case for 198 human case is reduced to 296 K which is equal to comfort temperature of 296K (23°C), also shows that some areas of peak temperature of 300 K due to stagnant region.

**CASE 3: - Lack in the Air-Condition unit Natural Ventilation Opening Front and back door**

Natural ventilation is recognized as a traditional technique that relies on wind and thermal buoyancy to improve indoor thermal environment and air quality. As an alternative to fan-assisted systems, natural ventilation is more energy efficient and is regarded as cleaner, healthier, and “naturalness” way.
for the connection with outside. In case 3 is lecturing room is considered as natural convection and air flow inside to room from front and back doors, in case of failure of AC and 198 human load is considered for the simulation.

![Image](image1)

198 human load with Air conditioning

![Image](image2)

198 human load without Air conditioning

Fig.15. Comparisons Temperature contours on inlet plane with and without AC

The Comparisons of temperature contours on inlet plane is shown in Fig 15 fro with and without AC. It shows average temperature of the flow stagnant zone back side of the lecturing room is reduced for 198 human load approximately 306K (33°C) which is higher than comfort temperature of 296K (23°C), may be in this lights should be switched off to feel more comfortable.

**Case 4. Simulation of effects of dangerous aspects considering smoke entering into the lecturing room from front door.**

In this case from front door stack releases smoke that is dispersed into the lecturing room with an oncoming side wind of 1 m/s. Initially, no smoke is being released. Subsequently, the front door starts to release smoke is shown in Fig.16. As can be seen, the smoke concentration rises to its asymptotic value reaching 90% of its final value at around 7 seconds and releases until 30 seconds.

![Image](image3)

Fig.16 Smoke inlet conditions for lecturing room

Fig.17 and 18 shows the Smoke Evacuation of from 30 sec to 3 min, so minimum time required to remove the smoke is approximately 3 min from the lecturing room.

**IV. CONCLUSION**

In this work, the influence of human loads, room lights, globe lights on Forced and natural ventilation and outdoor temperature on the indoor airflow pattern was examined based on CFD analysis. Airflow patterns mainly caused by Air conditioning natural ventilation, and smoke propagations were analyzed.
For 234 human load case the average Temperature contours in lecturing room is approximately 301K (28 C) which is higher than comfort temperature of 296K (23 C) due to the flow stagnant zone back side of the lecturing room.

So the human load is reduced to 198 to achieve the comfort zone for human being of 296 K (23C).

Then the natural ventilations is simulated. It shows average temperature of the flow stagnant zone back side of the lecturing room is approximately 305 K (32 C) for 198 human case, also shows that some areas of peak temperature of 308 K due to stagnant region.

Finally smoke simulation is carried out from front door stack releases smoke that is dispersed into the lecturing room with an oncoming side wind of 1 m/s. the total time required to evacuate the smoke is approximately 3 min.

REFERENCES

[9] Frédéric Conte, CFD simulations of smoke detection in rooms with high ceilings. SP AR 2002:30 Brandtechnik Borås 2002