International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 3, March - 2014

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

[1.] Md Atiqur Rahman, [2.] Narahari. G. A
[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 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.

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 with the use of computer fluid dynamics (CFD) software from Fluent Inc. The summer period is particularly critical.

Rong Li et.al [7] investigated buoyancy-driven natural ventilation of a room with large lower and higher level openings by both theoretical analysis and CFD simulation

R. Ramponi et.al [8]validated based on detailed wind tunnel experiments with Particle Image Velocimetry. The impact of a wide range of computational parameters is investigated, including the size of the computational domain, the resolution of the computational grid, the inlet turbulent kinetic energy profile of the atmospheric boundary layer, the turbulence model, the order of the discretization schemes and the iterative convergence criteria. Specific attention is given to the problem of oscillatory convergence that was observed during some of these coupled CFD simulations.

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; and

- 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.

Numerical computations were performed using commercially available**.** software package Fluent developed by ANSYS

.

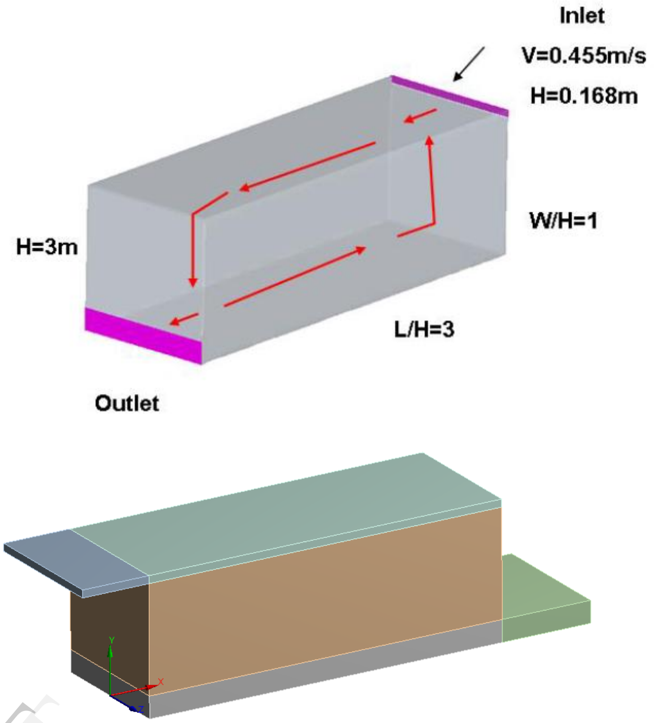


Fig.1 Geometry and CFD domain of Forced Convection Validation Study

CFD Meshing
The whole computational domain has to be divided into small control volumes, called grid cells in order to solve the discretized transport equations. Constructing a computational grid is a constant tradeoff between accuracy and CPU-time; when a grid is coarse the systems that have to be solved are small which implies short-CPU times. The downside is that a coarse grid is unable to represent small velocity or pressure gradients in the flow field. A very fine grid will be more accurate but can take undesirably long CPU-times. An additional disadvantage of a fine grid is that discretization gives a small round off error for every grid cell; more grid cells imply more round off errors
The computational grid used in this thesis can be found in Fig.2 . The grid is created with the program “AMP”. This is a standard mesh-generator compatible with several CFD packages, including Fluent. The main advantages of this program are the automatic mesh generator and the extended options to adapt the model to user preferences. AMP defines the model, grid and all boundary types.

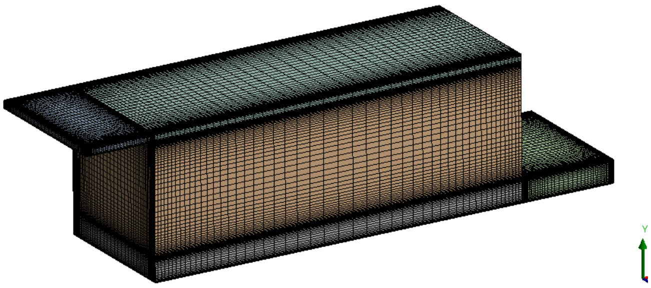


Fig.2. CFD Meshing Room model

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, 15% return air ventilation. There are 15 supply duct and 6 return duct.

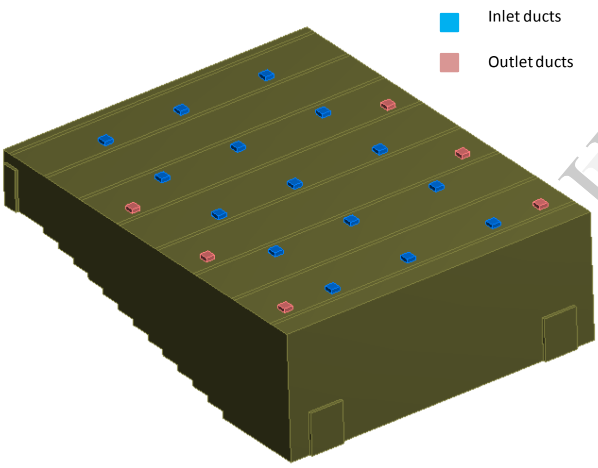


Fig.3. CFD Model Lecturing room Inlet and Outlet Location

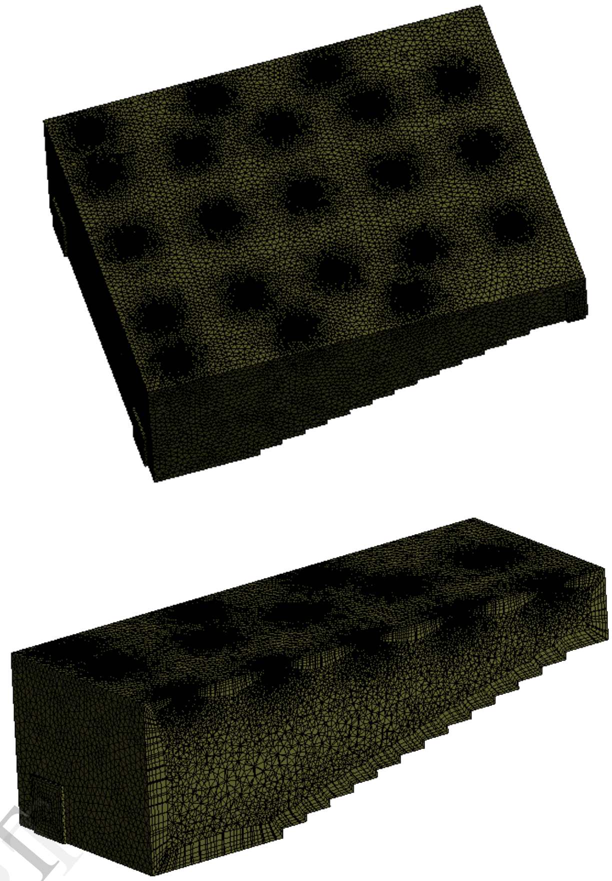


Fig.4.CFD Meshing details

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^{o}C, 1atm and treated as incompressible fluid. The ambient conditions were set to be 28^{o}C with 10 W/m^{2}K and 0 Pa relative to 1atm. Walls and ceiling boundaries within the domain were of type "Intermediate" and modeled as adiabatic on both sides

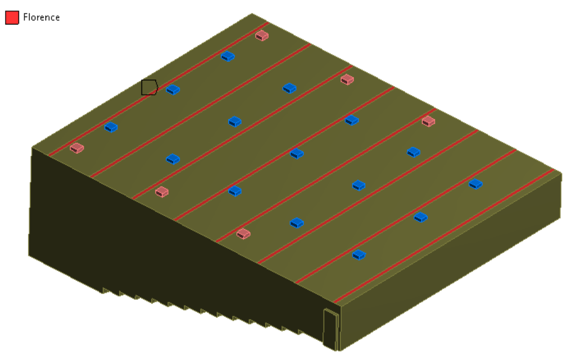


Fig.5.Florence Light Heat Source

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*2*50 = 8000 W.

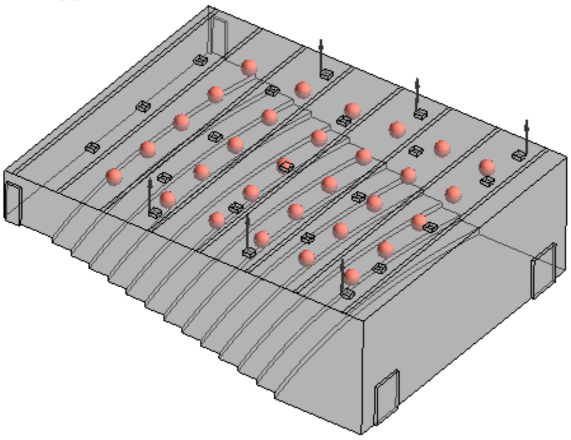


Fig.6 Globe Light-Heat Load-Isometric View

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*100 = 3000W.

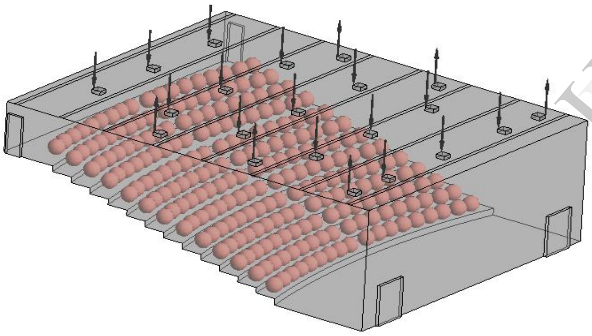


Fig.7 Human-Heat Load

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

III. RESULTS AND DISCUSSIONS

1. Validation study

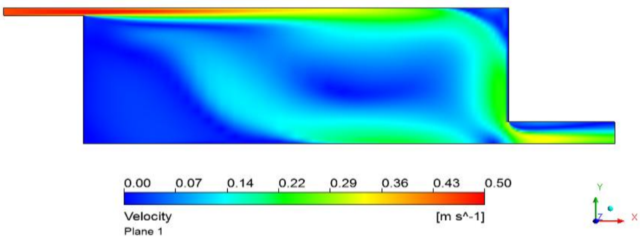


Fig.8.Velocity Magnitude contours

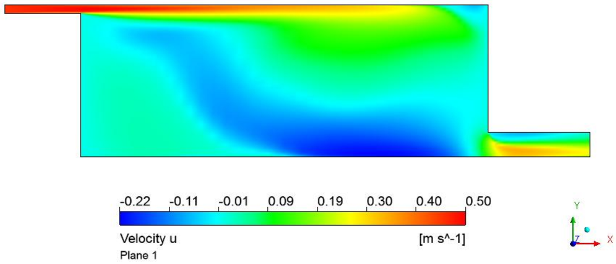


Fig.9.X-Velocity Magnitude contours

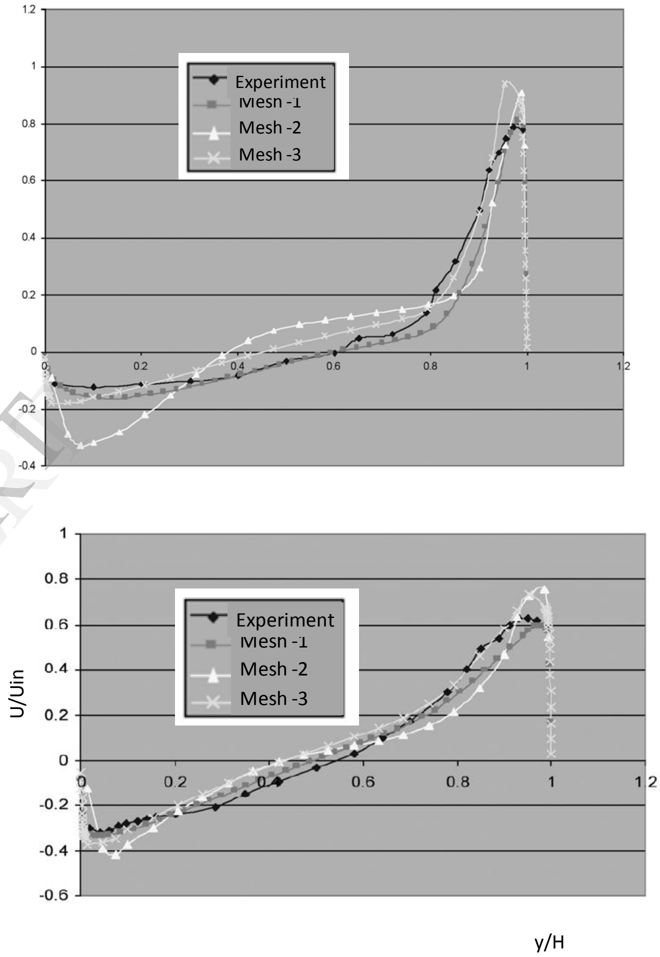


Fig.10. Comparison of dimensionless mean velocity profiles predicted by the investigated turbulence models to the experimental data at planes x/H = 1 and 2

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

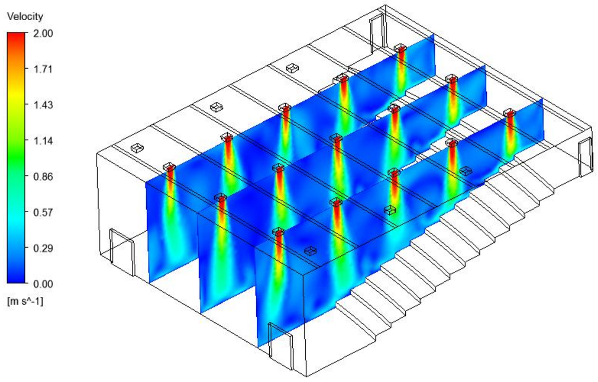


Fig.11.Velocity contours on inlet plane

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

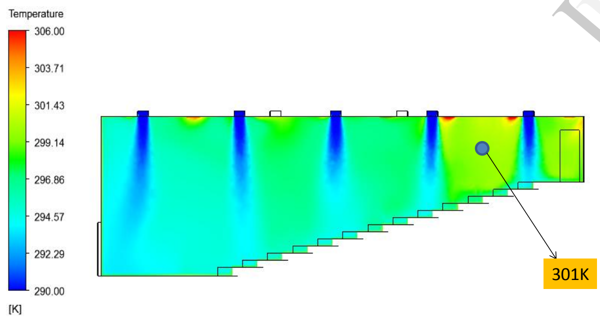


Fig.12 Temperature contours on inlet plane

The Temperature contours on inlet plane is shown in Fig 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).

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.

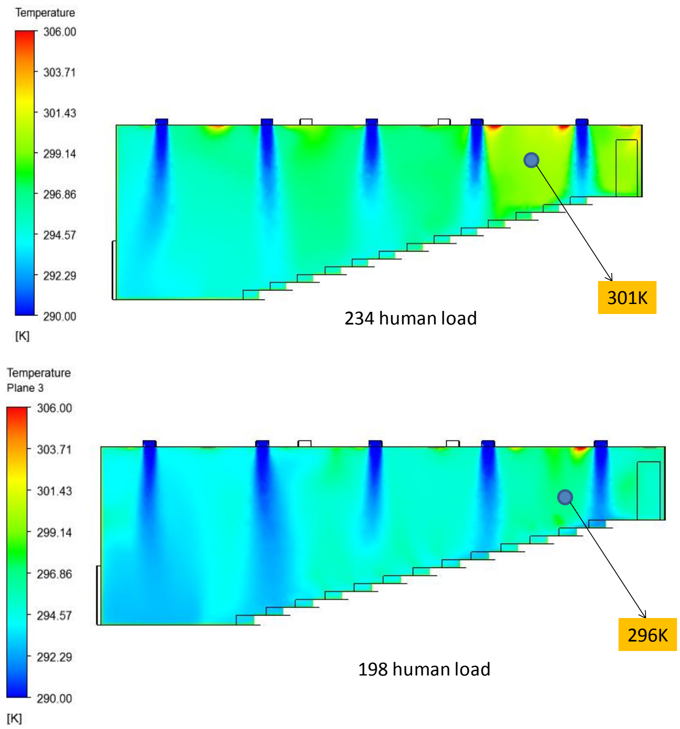


Fig.13. Comparisons Temperature contours on inlet plane

The Comparisons of temperature contours on inlet plane is shown in Fig 13. It shows average temperature of the flow stagnant zone back side of the lecturing room is reduced for 198 human load approximately 296K (2C C) which is equal to the comfort temperature of 296K (23 C).

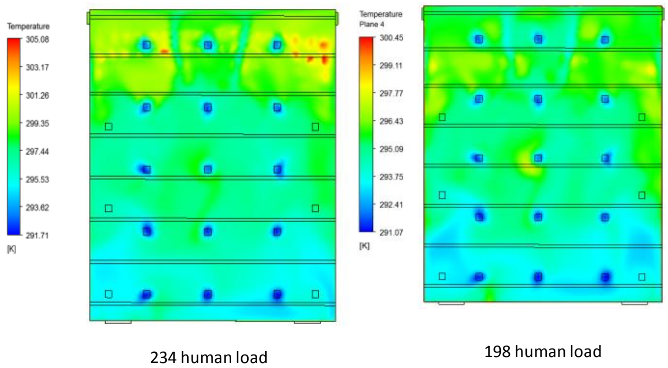


Fig.14 Comparisons of Temperature contours on Horizontal plane

The Temperature contours on horizontal plane is shown in Fig 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.

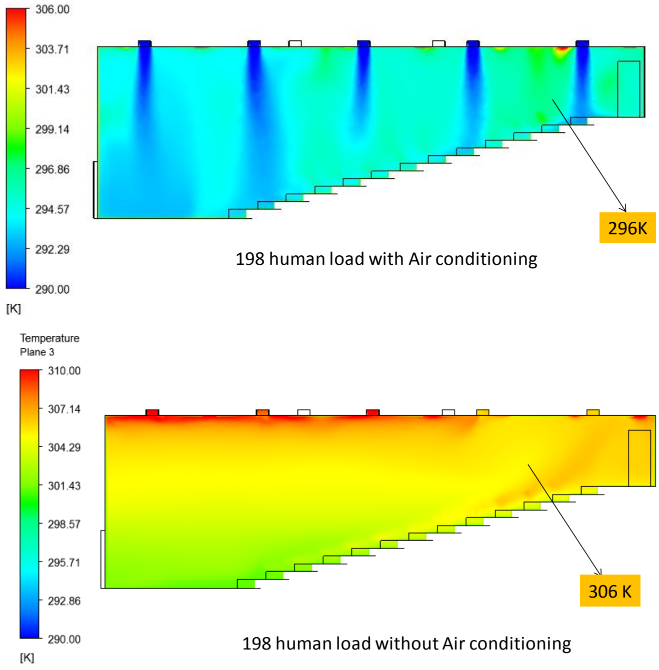
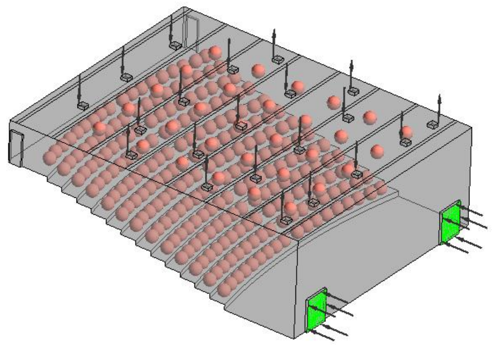


Fig.15. Comparisons Temperature contours on inlet plane with and without AC

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 (33C) 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 seonds.



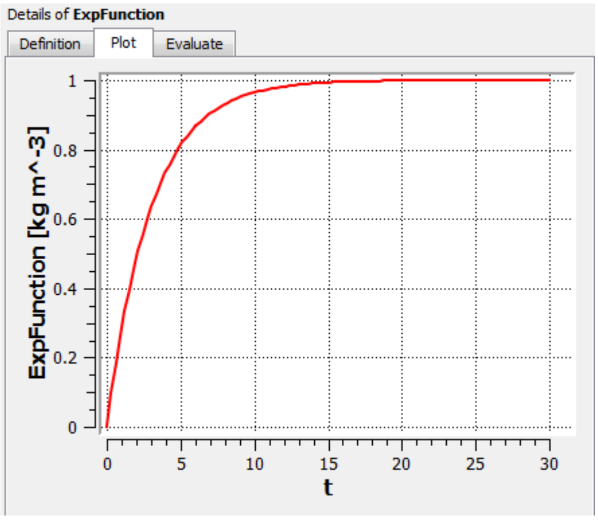


Fig.16 Smoke inlet conditions for lecturing room

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.

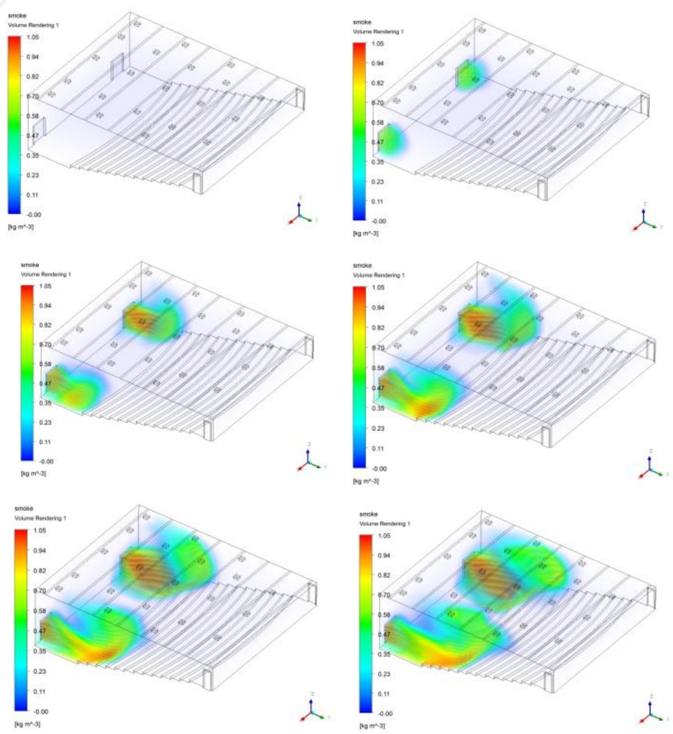


Fig.17. Smoke evaluation for 30 seconds

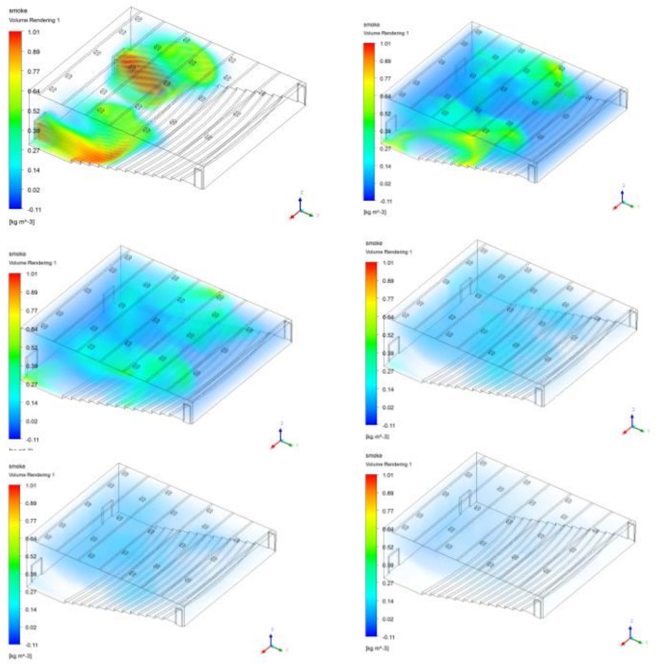


Fig.18. Smoke Evacuation of from 30 sec to 3 min

- 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

[1] Frank P. Incropera, David P. DeWitt, Fundamentals of heat transfer, Wiley, 1981
[1] Y.A. Cengel, Heat Transfer: A Practical Approach, 2nd ed., McGraw-Hill, New York, 2003.
[2] ASHRAE Handbook 2001 Fundamentals, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Atlanta, GA, 2001
[3] http://www.dmme.virginia.gov/DE/LinkDocuments/HandbookLighting.pdf
[4] Nielsen, P. V., Restivo, A., & Whitelaw, J. H. (1978). The velocity characteristics of ventilated rooms. Journal of Fluids Engineering, 100, 291-298.
[5] Nielsen, P. V. (1998). The selection of turbulence models for prediction of room airflow. ASHRAE Transactions, 104(Part 1), 1119-1127
[6] M. Zajicek, and P. Kic, Improvement of the broiler house ventilation using the CFD simulation, *Agronomy Research* Biosystem Engineering Special Issue 1, 235-242, 2012.
[7] Rong Li, Adrian Pitts, and Yuguo Li, Buoyancy-Driven Natural Ventilation Of A Room With Large Openings, Proceedings: Building Simulation 2007.
[8] Ramponi R, Blocken B. 2012. CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters. Building and Environment 53: 34-48.
[9] Frédéric Conte, CFD simulations of smoke detection in rooms with high ceilings. SP AR 2002:30 Brandteknik Borås 2002
[10] Srebric, J. and Chen, Q. 2001. "A method of test to obtain diffuser data for CFD modeling of room airflow," *ASHRAE Transactions*, 107(2), 108-116.
[11] Robert N. Meroney, CFD Prediction of Airflow in Buildings for Natural Ventilation, 11th wind engineering conference.2009.
[12] Shafqat Hussain and Patrick H. Oosthuizen, Numerical Modeling of Buoyancy-driven Natural Ventilation in a Simple Three Storey Atrium Building, 20012.
[13] Ryoichi Kajiya1, Kodai Hiruta2, Koji Sakai1, Hiroki Ono1, Toshihiko Sudo Thermal Environment Prediction Using Cfd With A Virtual Mannequin Model And Experiment With Subject In A Floor Heating Room Proceedings of Building Simulation 2011: 12th Conference of International Building Performance Simulation Association, Sydney, 14-16 November