International Journal on Emerging Technologies **6**(1): 89-97(2015)

ISSN No. (Print) : 0975-8364
ISSN No. (Online) : 2249-3255

Numerical study on the effect of Supply Air Position in a Hall for Thermal Comfort

Shamir Daniel* and Dr. V.N. Bartaria**
**P.G. Scholar, Department of Mechanical Engineering, LNCT Bhopal, (MP), INDIA*
***Professor, Department of Mechanical Engineering, LNCT Bhopal, (MP), INDIA*

(Corresponding author: Shamir Daniel)
(Received 04 March, 2015 Accepted 04 April, 2015)
(Published by Research Trend, Website: www.researchtrend.net)

ABSTRACT: Conventional air conditioning systems in buildings are designed to create a uniform environment across the entire occupied zone. Individual thermal preferences cannot be accommodated, and the fresh supply air is polluted by indoor contaminants before its inhalation by the occupants. An urgent demand of comfortable indoors environment have resulted in a number of numerical studies related to the prediction of indoor air distribution. At present, the computational fluid dynamics (CFD) technique has become a powerful tool for obtaining information about the indoor environment with the increase of PC performance. CFD provides us the tool that enables us to gain greater understanding of likely airflow and heat transfer processes within the space under consideration specified by boundary conditions such as effects of climate, internal energy sources and air-conditioning systems used [5]. By using CFD we can analyze how the location of supply air port and exhaust air port will affect the overall indoor environment under consideration. Thermal stratification is very important issue for energy saving. Increasing stratification may save energy, while decreasing stratification may improve occupant comfort. The temperature gradient is of great interest as a person does not like to be exposed to large temperature differences between the feet and the head. Further, CFD modeling could save cost, effort and time involved and give better visualization of results.

Keywords: Conventional, conditioning, ventilated room

I. INTRODUCTION

In modern society, most people spend a great part of their lifetime staying indoors. As a result, creating comfortable indoor1 environment plays an important role on people's indoor activities. Among the factors, to create comfortably indoor environment: making comfortable thermal environment is one of the most important factors to satisfy human's sensation of "comfort". Thermal comfort is defined as "that condition of mind which expresses satisfaction with thermal environment" [19]. It is evaluated by two personal factors, comprising: activity level and thermal insulation of clothing, as well as four environmental parameters including: air temperature, means radiant temperature, air velocity, and air humidity. This thesis is motivated by creating the indoor environment with thermal comfort for humans. Thermal comfort has close relation to the indoor air movement [7,15]. However, air movement within a room depends upon several factors [13]. For example, indoor air movement is often induced by the forced convective airflow [4] supplied by an air-conditioning system.

Also, temperature difference between the wall of a room and indoor air may cause indoor air movement, such as natural convection. Air movement caused by a differential pressure across the indoor structure may be considerable. The existence of doorways and apertures inside a room could have great impact on the indoor air movement. The opening and closing of doors coupled with people's movement may have important influence on the indoor air distribution. Owing to the urgent demand of comfortable living indoors, a number of numerical studies related to the prediction of indoor air distribution [8, 9, 14, 17] within a ventilated room have been conducted in the past decades. This thesis mainly focuses on (1) inspecting the influence of different turbulence models, and various numerical schemes and algorithms on the simulation results; and (2) the verification of simulation and theoretical results. However, most of this thesis concentrated on predicting steady-state indoor air distribution within a ventilated hall. In fact, transient behavior of indoor flow characteristics could have great influence on thermal comfort of human beings.

The purpose of a ventilating system is to remove heat generation within a room, creating a thermally comfortable environment. One of the active approaches for regulating heat transfer in a convection-dominated cavity system is to use a periodic moving object [6]. Air-conditioning systems have been used in many parts of the world. Computer fluid dynamics (CFD) analysis is a recent achievement of technology and is finding growing application in the field of air conditioning, enabling us to predict the indoor environmental conditions through simulation [1,5]. Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. A large number of equations are required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that can improve the accuracy and speed of complex simulation scenarios. Initial theoretical validations of such software are performed using a wind tunnel and final validation by performing a full-scale testing. Computational Fluid Dynamics (CFD) describes a family of numerical methods terms used to calculate the temperature, velocity and various other fluid properties throughout a given region of space under consideration.

A. Air Conditioning System

If the theory is properly applied, every system can be successful in any building. However, in practice, factors such as initial and operating costs, space allocation, architectural design, location, and the engineer's evaluation and experience limit the proper choices for a given building type [18]. Heating and air-conditioning systems should be: Simple in design, of proper size for a given building, of generally fairly low maintenance, of low operating costs and of optimum inherent thermal control as is economically possible [10]. For buildings the following parameters are to be considered, as shown in Fig. 1.

Fig. 1. Harmonization of energy terms in building technology.

B. Load Characteristics

Sun Gain. Office and conference rooms usually have windows and door opening; the walls are subjected to direct sunlight. East and west walls are more subjected to direct sunlight. As India is located in the northern hemisphere, the southern walls are slightly hotter than the northern wall.

Transmission. In winter, effects on objects located close to outside walls and possible condensation of moisture on the objects and the surface of outside walls must be evaluated. In summer, possible radiant effects from exposure should be considered.

People. Some areas may have concentrations as high as 1.0 m2 per person, while office space will have closer to 10 or 15 m2 per person. When smoking is permitted, return air should be contained, and the re-circulated part of the air should be deodorized with activated charcoal and similar odour-removal devices or exhausted.

Lights. Careful analyses of the required lighting intensity should be made in a hall and in view of day-lighting availability.

Stratification. In reading rooms, large lecture halls, and huge offices with high natural or false ceilings, air temperature may stratify.

C. Design Concepts

All-air ducted systems are preferred in library public areas, careful evaluation of relative humidity is essential. A number of people with different perception towards heat and cold may react differently to a given thermal comfort. Thus, individually controlled zones may be required to maintain optimal environmental conditions. Another problem is the location of room thermostats and humidistats.

D. Design Criteria

While considering an enclosed office, more stringent design criteria are usually provided for achievers, because the final output provided by persons sitting in office very valuable to the company. A high-efficiency air filters are often provided. Relative humidity is held below 55%. Room temperature is held within the 20 to 26°C range. Air velocity inside the room shall be less than 0.5m/sec.

E. Building Contents

The effect of equipments on the indoor thermal conditions should be carefully considered and critically examined. The number of skylights required for properly illuminating the room, requirement of wall mounted fans, if required, the amount of heat dissipated by these to the indoor environment shall be considered.

F. Effect of Ambient Atmosphere

The temperature and, particularly, the relative humidity of the air have a marked influence on the surrounding air[19, 20]. The object humidity is usually defined as the relative humidity of the thin film of air in close contact with the surface of an object and at a temperature cooler or warmer than the ambient dry bulb. If objects in a an office or hall are permitted to cool overnight, the next day they will be enveloped by layers of air having progressively higher relative humidity. These may range from the ambient of 45 to 60% to 97% immediately next to the object surface, thus effecting a change in material regain or even condensation. If the particular material is warmed, however, the object's humidity will be lower than the humidity of the surrounding pace. This warming may be caused by spotlights or any hot, radiating surface.

G. Sound and Vibration

Air-conditioning equipment should be treated with sound and vibration isolation to ensure quiet comfort for visitors and person as per the ASHRAE standards and local environmental laws.

H. Ventilation and Indoor Air Quality

All houses need ventilation to remove stale interior air and excessive moisture and to provide oxygen for the inhabitants. There has been considerable concern recently about how much ventilation is required to maintain the quality of air in homes. While it is difficult to gauge the severity of indoor air quality problems, building science experts and most indoor air quality specialists agree that the solution is not to build an inefficient, "leaky" home. Research studies show that standard houses are as likely to have indoor air quality problems as energy efficient ones [3, 11, 12]. While opening and closing windows offers one way to control outside air for ventilation, this strategy is rarely useful on a regular, year-round basis. Most building researchers believe that no house is so leaky that the occupants can be relieved of concerns about indoor air quality [16]. The researchers recommend mechanical ventilation systems for all houses. The amount of ventilation required depends on the number of occupants and their lifestyle, as well as the design of the home. The ANSI/ASHRAE standard, "Ventilation and Acceptable Indoor Air Quality in Low-Rise Residential Buildings" (ANSI/ASHRAE 62.2-2007) recommends that houses have 7.5 natural cubic feet per minute of fresh air per bedroom + 1, plus additional air flow equal to (in cubic feet per minute) 1% of the house conditioned area, measured in square feet. In addition, the standard requires exhaust fans in the kitchen and bathrooms that can be operated when needed. Air leaks are not a reliable source of fresh air and they are not controllable. Air leaks are unpredictable, and leakage rates for all houses vary. For example, air leakage is greater during cold, windy periods and can be quite low during hot weather. Thus, pollutants may accumulate during periods of calm weather even in drafty houses. Concerns about indoor air quality are leading more and more homeowners to install controlled ventilation systems for providing a reliable source of fresh air.

II. RESEARCH METHODOLOGY

This work has been carried out in two stages. In the first stage, a theoretical and conceptual study is carried out to explore how the supply air position can affect the closed environment in terms of thermal comfort in terms of temperature and velocity. The effect are measured by varying supply air temperature and supply air velocity so that inside environment is at thermal comfort level. In the second stage CFD modeling is used, and the objective data from the theories would be used to validate the CFD models. The CFD models would be used to test other combinations of inlet supply air temperature and inlet air velocity and its effect on the overall indoor environment under consideration. CFD will be uses to test supply air temperature and air flow rates and to investigate their impact on local air quality and thermal environment.

A. CFD model

Computational Fluid Dynamics (CFD) modeling is employed to further study the effect of supply air position in a closed environment under consideration. The main focus of the CFD study is to assess the effect of the air flow rate and supply air temperature on the performance of the system under study. Meanwhile, the results from CFD study could provide more information on the system and better visualization than the theoretical study, such as the tracking of the path lines, temperature profiles and velocity profiles. The CFD results are able to visually display the air flow pattern in the hall. Figure 2 shows the simplified model of the studied chamber. It consists of a three dimensional model (a large hall) with an inlet port, an outlet port while considering all the doors and windows as tightly closed walls with no leakages. The models is a hall and is considered empty for simplicity. The position of outlet port is kept fixed at upper corner of the wall. The inlet supply air position is varied to analyze its effect in the enclosed environment.

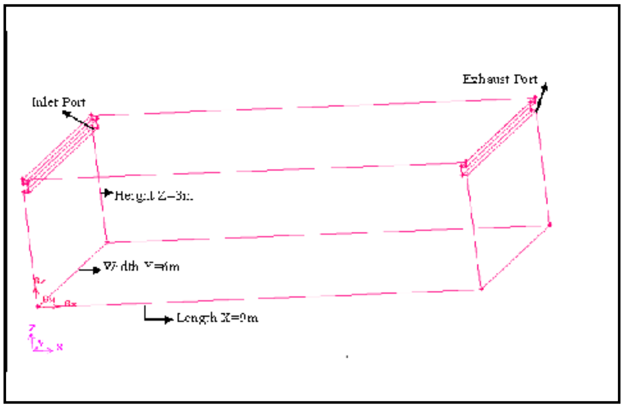

Fig. 2. The Geometrical model.

The study is done by considering three cases for inlet supply air positions: Case I: Inlet supply air is positioned at the upper part of a wall which is in front of the exhaust port wall, the exhaust port is at the upper part of the wall, Case II: Inlet supply air is positioned at the lower part of a wall which is in front of the exhaust port wall, the exhaust port is at the upper part of the wall, Case III Inlet supply air is positioned at the ceiling close to the wall which is in front of the exhaust port wall, the exhaust poet is at the upper part of the wall. CFD model is prepared in the preprocessor software called GAMBIT. The basic model is prepared by using bottom-top approach. Once the basic model is prepared, meshing of the model is done. An hexahedral mesh is chosen based on literature survey done. Then the inlet and outlet ports were fixed as per the three cases under consideration.

The files were saved as .mesh file in the GAMBIT. Then the post-processing is done using CFD software FLUENT 6.3 [21].

B. Boundary conditions

The boundary condition is another important factor deciding the success of a CFD simulation. Right setting of boundary conditions is crucial in two ways, i.e., it may affect convergence and it may affect the correctness of results. The boundaries involved in this study include three types: wall boundary, mass flow inlet and pressure outlet. The surfaces of the room wall, ceiling and floor, are all set as wall boundaries. The Inlet supply air post is set as mass flow inlets and the exhaust port is taken as pressure outlet. The details of wall boundary parameters of the surfaces are listed in Table 1.

Table 1: Boundary conditions of manikin surface.

Segments	(K) (Temp.)	(°C) (Temp.)
East Wall	305	32
West Wall	305	32
North Wall	300	27
South Wall	300	27
Ceiling Insulated	300	27
Bottom Insulated	300	27

C. Simulation settings

All simulations are performed for three dimensional steady-state airflows in the thermal chamber. The turbulence model used is RNG k-ε model with enhance wall treatment. Pressure gradient effects and thermal effects are turned on for the enhanced wall treatment. Buoyancy effect is enabled by turning on gravity and set the fluid density as ideal-gas. Under-relaxation factors are used and reduced from the default values in the software to ensure convergence. SIMPLE algorithm is used for pressure-velocity coupling. Standard algorithm is used for pressure discretization, and first order upwind algorithm is used for the discretization of the other parameters.

III. SIMULATION

Simulation was carried out for all the three cases, temperature profiles were plotted.

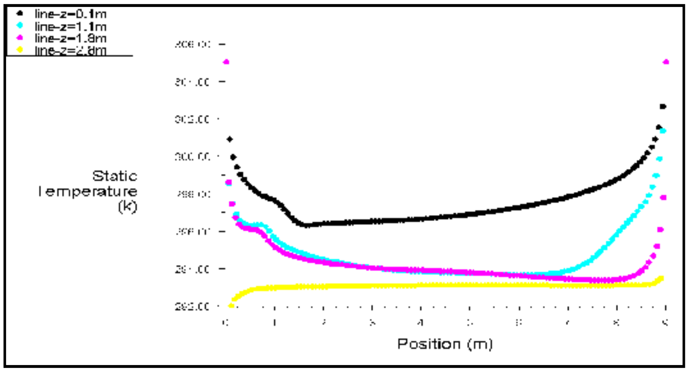

Fig. 3. Temperature profile at different heights for 292°K supply air temperature Case I.

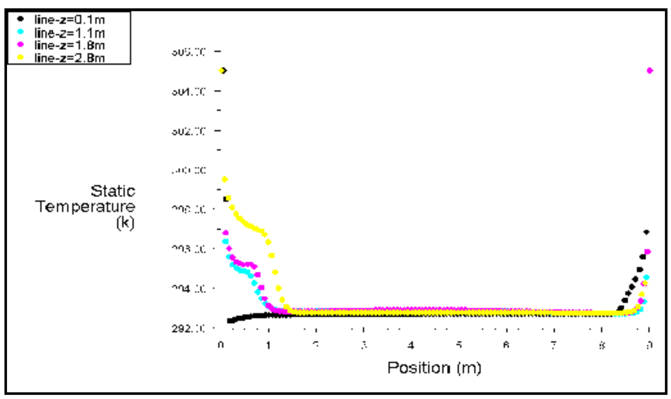

Fig. 4. Temperature profile at different heights for 292°K supply air temperature Case II.

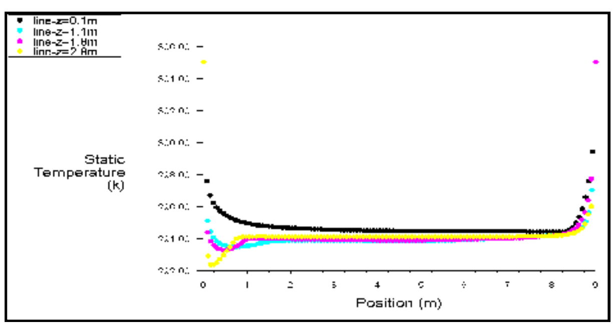

Fig. 5. Temperature profile at different heights for 292°K supply air temperature Case III.

As observed from figures 1, figure 2 and figure 3, Case III provides best results compared to the other two cases considered. In Case III, the temperature and velocity profiles on X-Y plane (ie inside a closed hall) at various heights (Z=0.1m, 1.1m, 1.8m, and 2.8m) are plotted. Stratification is observed in the analysis, the temperature difference between ankle position (Z=0.1m) and to the head position while person is sitting (Z=1.1m) is observed to be less than 1°C, which is within the acceptable limit for a person to feel comfortable as per ASHRAE standard. The increase in velocity helps in slight drop in temperature at a particular point under consideration. Thus a slight increase in velocity can be effective in maintaining the overall inside temperature of the hall at peak summer seasons with slight increase in energy consumption, while the overall system can provide thermal comfort with less energy consumption during rest of the year. Further the temperature profiles for Case III are plotted for different Inlet supply air temperatures varying from 292°K, 294°K, 296°K and 298°K (i.e. 19°C, 21°C, 23° and 25°C) with fixed Inlet supply air velocity 0.2m/s.

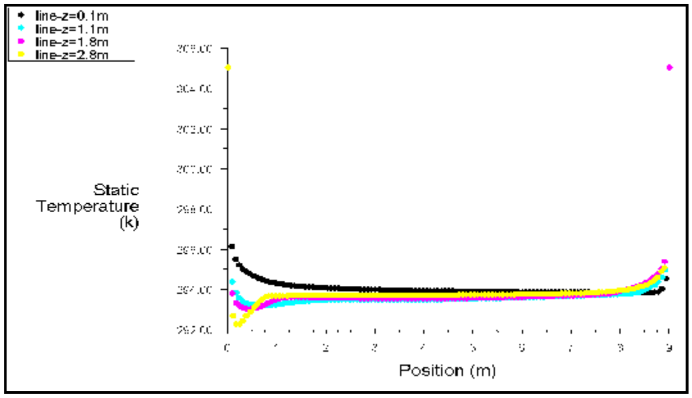

Fig. 6. Temperature profile at different heights for 292°K supply air temperature.

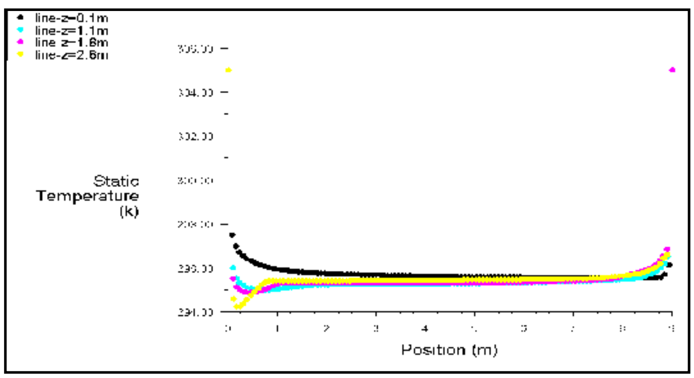

Fig. 7. Temperature profile at different heights for 294°K supply air temperature.

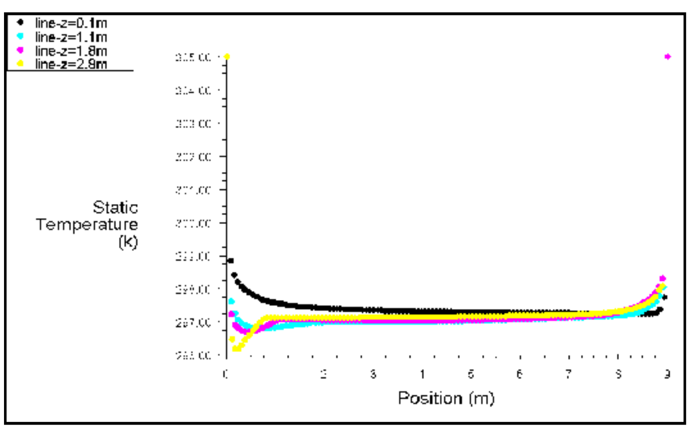

Fig. 8. Temperature profile at different heights for 296°K supply air temperature.

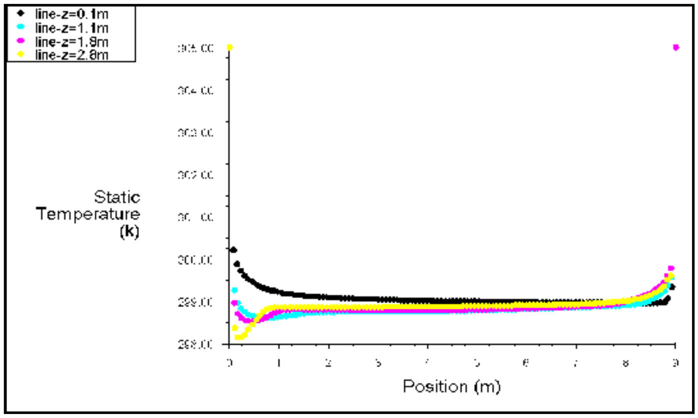

Fig. 9. Temperature profile at different heights for 298°K supply air temperature.

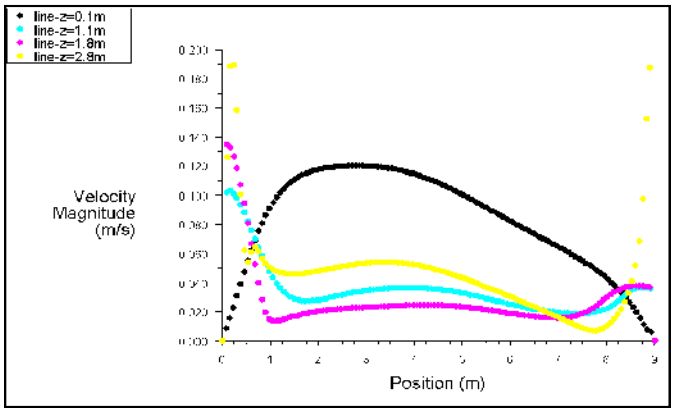

Fig. 10. Velocity profile at different heights for 0.2m/s at 298°K supply air temperature.

As observed from the fig. 5.35, 5.36, 5.37 and 5.38, condition of thermal comfort inside the hall is achieved. The temperature difference between ankle height (z=0.1m) and head while person is sitting (z=1.1m) is <1°C. Stratification is also observed, resulting in proper mixing and movement of hot air towards the exhaust port. Increase in velocity further reduces the temperature at a point and also increases working zone slightly.

Case III, the velocity profiles are within the comfortable limit (ie < 0.2m/s).

Thus Input Supply air velocity of 0.2m/s and Input supply air temperature of 298°K provides the optimum condition in terms of power consumption. The Overall velocity inside the hall is within 0.2m/s and temperature is within the range of 295°K- 299°K (i.e. 22°C - 26°C).

IV. RESULTS

In Case III (i.e. inlet at ceiling of the room at length x=0.2m and outlet at opposite side wall top i.e. length x=8.9m)

(1) The maximum temperature inside the room remains with 300K (< 27°C) which is under comfort condition for all three velocities.

(2) The temperature difference between ankle (z=0.1m) and Head level while person is sitting (z=1.1m) is less than 1K (<1°C) this is also within comfortable condition.

(3) The stratification is also observed, resulting in proper mixing and movement of hot air towards the exhaust port.

(4) Working zone reduction is about 0.8m at the Inlet wall and about 0.4m at the outlet wall.

(5) Increase in velocity helps in decreasing the uncomfortable zone at the inlet, though there is cost associated for increase in velocity. .

V. CONCLUSION

To validate simulation, results cited from ASHRAE and numerical simulation were compared and analyzed. By using the RNG k-ε Renolds Average Navier- Stokes Turbulence model three cases were analyzed numerically. Conclusions regarding the effects of locations of the diffuser on the air flow field in an air conditioning hall have been drawn:

(1) It is reliable to predict the air flow field of the air conditioning hall by RNG k-ε model. The regions of higher temperature and greater velocity are closer to the experimental results. Because of the complexity of boundary conditions, the temperature calculation results are underestimated.

(2) When the supply air inlet port is at the lower part of office, the effects of different locations on the temperature distribution of office is not significant. The stratification is not achieved as desired. Influence of the supply air inlet port with height is more obvious than with locations at a certain height to the temperature. When the supply air inlet port is at the top of the hall, stratification is observed but as observed it is on the higher side, the temperature difference between ankle position and to the head position while person is sitting is observed to be about 3°C, which is on a much higher side for a person to feel comfortable as per ASHRAE standard.

(3) When the diffuser is at the ceiling of the hall. The stratification is observed, the temperature difference between ankle position and to the head position while person is sitting is observed to be less than 1°C, which is within the acceptable limit for a person to feel comfortable as per ASHRAE standard.

(4) Thus it is observed from the simulation that out of the three cases considered the Case III provides best results. It is observed that in this case increase in velocity helps in slight drop in temperature at a particular point under consideration. Thus a slight increase in velocity can be effective in maintaining the overall inside temperature of the hall at peak summer seasons with slight increase in energy consumption, while the overall system can provide thermal comfort with less energy consumption during rest of the year.

VI. FUTURE WORK

While the objective of this research was to use three-dimensional CFD analysis to determine a set of principles to guide the design of heating and cooling systems, many difficulties were encountered in devising, evaluating, and validating a three-dimensional model. As such, this leaves much room for future work in this area. Not only do problems such as computational limits and times complicate the matter of performing a three-dimensional investigation, but the sheer number of possible model configurations makes the development of a meaningful model more difficult as there are unlimited variations of actual physical hall designs. Additionally, while a single model may be able to provide insights into the temperature and flow patterns within a hall, changes as simple as an open doorway or any obstruction added within the room could dramatically affect the observed patterns within a physical building. Areas of interest in future simulations would include changing of vent locations and changing of air injection methods. For example, a pulsed injection with a mean volumetric flow equal to the injection flow rate of a steady flow may offer more mixing and better results. Also, changing the number of inlets and outlets that are active in any given case may yield desirable results.

REFERENCES

[1] Liu, J., Aizawa, Y., and Yoshino, H., (2002). Experimental and numerical study on simultaneous temperature and humidity distributions, *ROOMVENT 2002*, 169-172.

[2] Yang Ming, Yuan Dongsheng and Ren Xiaoli (2012). Numerical simulation on the influence of air-supply velocity in air-conditioning office, IPCSIT vol. **47**, 2012.

[3] Essam E. Khalil, (2002). Energy-Efficiency in Air Conditioned Buildings: The Green Buildings Dream, *Int. J. of Thermal & Environmental Engineering* Vol. **2**, No. 1 ventilation, ROOMVENT 2002, page 209-212.

[4] Kana Horikiri, Yufeng Yao and Jun Yao, Numerical Simulation of Convective Airflow in an Empty Room, *International Journal Of Energy And Environment*, Vol. **5(**4), 2011.

[5] Kameel, R., and Khalil, E.E., (2001). Numerical computations of the fluid flow and heat transfer in air-conditioned spaces, NHTC2001-20084, *35th National Heat Transfer Conference*, Anaheim, California.

[6] CC Tsao, S W Nien, W H Chen, Y C Shih, (2013). The Impact of an air supply guide vane on the Indoor air distribution, World Academy of Science, Engineering and Technology, 2013.

[7] Nakamura, Y., and Fujikawa, A., (2002). Evaluation of thermal comfort and energy conservation of an ecological village office, ROOMVENT 2002, page 413-416.

[8] Kameel, R., and Khalil, E.E., (2002). Prediction of flow, turbulence, heat-transfer and air humidity patterns in operating theatres, ROOMVENT 2002, page 69-72.

[9] Kameel, R., and Khalil, E.E., (2002). Prediction of turbulence behavior using k- ε model in operating theatres, ROOMVENT 2002, page 73-76.

[10] Alexander Kayne, Ramesh K Agarwal, (2013). Computational Fluid dynamics modeling of mixed convection flows in building enclosures, *International Journal of Energy and Environment*, 2013.

[11] Van Dike, D. and Khalil, E. E., (2009). Energy Efficiency in Buildings, ISO Focus, September 2009, page.16-20, 2009.

[12] K David Huang, Nguyen Anh Tuan, (2009). Numerical and Experimental analysis of an Air conditioning energy saving mechanism, Asian Symposium on Computational Heat Transfer and Fluid Flow, 2009.

[13] Tibaut, P., and Wiesler, B., (2002). Thermal comfort assessment of indoor environments by means of CFD, *ROOMVENT 2002*, page 97-100.

[14] Braconnier, R, Fontaine, J. R., and Bonthoux, F., (2002). Use of predictive ventilation to evaluate the emission rates of pollutant sources in an enclosure and to reconstruct the associated concentration field, ROOMVENT 2002, page 133-136.

[15] Khalil, E.E., (2010). Thermal Comfort and Air Quality in Sustainable Climate Controlled Healthcare Applications, AIAA-2010-0802, Orlando, Jan 2010.

[16] C.H. Halios and C.G. Helmis, (2007). “On the estimation of characteristic indoor air quality parameters using analytical and numerical methods,” *Science of the Total Environment*, **380**(1-3), 2007, pp. 222-232.

[17] Q. Chen, (2009). “Ventilation performance prediction for buildings: A method overview and recent applications,” Building and Environment, **44**(4), 2009, pp. 848-858.

[18] Q. Chen, K. Lee, K. Mazumdar, S. Poussou, L. Wang, M. Wang and Z. Zhang, (2010). “Ventilation performance prediction for buildings: Model assessment,” *Building and Environment*, **45**(2), 2010, pp. 295-303.

[19] ASHRAE (American Society of Heating, Ventilating and Refrigerating Engineers) ASHRAE Handbook, Fundamentals, Atlanta, USA, ASHARE 2009.

[20] ASHRAE Handbook, HVAC Applications, Atlanta, USA, 2011.

[21] Fluent Inc., Fluent 6.3 User guide, Lebanon NH 03766, 2006.