International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 4, Issue 11, November 2015)

CFD Analysis of Air Conditioning Equipment of CAD Lab for Enhancing its Performance

K Vaseemul Rahaman[1], Prof. Shankar Kumar[2]
Mechanical Engg Dept., LNCT/RGPV, Bhopal (MP) - India

***Abstract*— Thermal displacement ventilation (TDV) is a promising technology for all types of buildings. The potential energy efficiency, health, and acoustic benefits offer promise government of various countries are prepared to spend billions on new educational institutes and major modernization. This paper presents some results from Computational fluid dynamics (CFD) analysis which is needed in order to model and study the complex flows associated with TDV.CFD is an established, state-of-the-art scientific approach for quantitative prediction and analysis of fluid flow, heat and mass transfer in a multiplicity of conditions.**

In 1995, the Government Accounting Office concluded that 25% of the nation's schools are inundated by indoor air quality (IAQ) problems. A few years later the Environmental Protection Agency reported that an even higher percentage of buildings have IAQ problems. Most of these IAQ problems can be recognized to poor ventilation. TDV, which has been used since the late 1980s in Northern Europe and only more recently in U.S., disproves the common perception that improving IAQ in an air-conditioned space should effect the higher energy consumption.

***Keywords*— TDV, CFD simulation, ventilation, heating, temperature, thermal comfort.**

I. INTRODUCTION

Thermal displacement ventilation (TDV) is a promising technology for every building.

Most classroom air conditioning systems deliver air at the ceiling at a temperature of about 55º F. Air is supplied at high velocity to provide efficient mixing of supply air with room air to provide uniform temperature throughout the space and to dilute contaminants with fresh supply air. By contrast, thermal displacement air conditioning systems deliver air near the floor instead of at the ceiling. The air is heated or cooled so that it enters the room at about 65º F, comparatively warmer than with a conventional air conditioning system. Control is provided by varying the volume of air that is delivered to the space. The diffusers are larger so that air speed is lower. Since the 65ºF air is cooler than the rest of the air in the room, it spreads by gravity all along the floor, thereby forming a continuous layer of cool air. The cool air rises as it is heated by the occupants and other heat generating sources. The effect of air rising up over a warm object is called a thermal plume.

Since the air moves straight up and all supply air is delivered at the floor level, each person basically has his or her own private supply of outside air ventilation.

Traditional design methods are not effective for classrooms with TDV, because they are based on the assumption that the space has a uniform temperature from floor to ceiling and that the air is well mixed. TDV works because there is a substantial temperature difference between the floor and the ceiling which drives air movement through buoyancy.

When the room requires heating, warm supply air (75 to 80°F) is provided from the diffusers, at a sufficient velocity to provide a fairly uniform temperature throughout the occupied zone.

Computational fluid dynamics (CFD) analysis is needed in order to model and study the complex flows associated with TDV. CFD is an established, state-of-the-art scientific approach for quantitative prediction and analysis of fluid flow, heat and mass transfer in a multiplicity of conditions. For this study, a commercially-available CFD software tool was used (Airpak 2.1.10 2002) whose solver engine is Fluent. The Airpak/Fluent package employs a finite-volume formulation of the governing differential equations used in CFD. The package can model basic fluid flow, heat transfer, turbulence, radiation heat transfer, and contaminant transport.

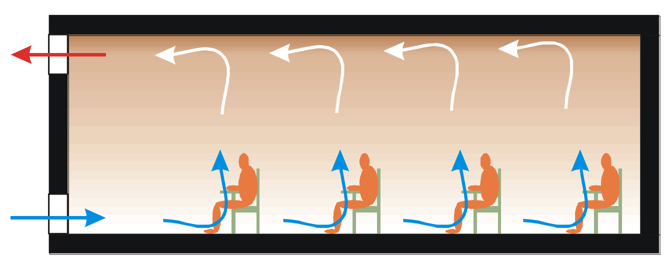


Fig 1. TDV Concept

II. WHAT IS CFD?

Computational Fluid Dynamics (CFD) is a computer-based analysis technique used to predict various physical and chemical phenomena like fluid flow, heat transfer, mass transfer, chemical reactions, phase change, combustion, and flow acoustics etc.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 4, Issue 11, November 2015)

It solves the mathematical equations governing these phenomenons. CFD is becoming a powerful tool, in combination with usual design techniques, to analyze the problems.

Advantages of CFD simulation

- Visualizing designs
- Complete Information
- Making prediction using results.
- Improved design ability
- Low Cost
- Ability to Simulate Real Conditions

Steps of CFD

- Preprocessing
- Solving
- Specifying the fluid and flow properties
- Choosing the discrete scheme
- Post processing

Fluids Dynamics are derived from the basic physical laws of conservation of mass, momentum, and energy. Logical solutions of such problems are possible only for very simple flow domains with certain assumptions made. For usual designs designers have to trust the empirical formulae, rules of thumb, and experimentation. Though, there are many intrinsic problems associated with these usual designs. Empirical formulae and rules of thumb are extremely precise to the problem and are not globally used because of the non-linearity of the governing equations thus making experimentation the leading design technique.

Limitations of experimentation technique:

- Measurement of flow variables may cause these variables to change by themselves, which is not possible at all (in very small or unreachable spaces), and can be expensive.
- Experimental data has less detail.
- Experimentation may take a long time to set up, which sometimes lasts for a very short time, and may cost more, as in the case of supersonic wind-tunnel runs.

All these limitations are overcome by CFD, as it is a numerical simulation technique which does not require a model to be built, is not dissatisfied by measurement capabilities, and can provide extremely detailed data as and when required.

Using CFD, you can create a computational model that represents a system or device that you want to study.CFD analysis not only complements testing and experimentation, but leads to a saving of time as a large number of options can be tested much before the prototyping stage.
CFD is, thus, a tool for compressing the design and development cycle.

Preprocessing:

It consists of input of the flow problems to CFD program by means of operator and to prepare a cad model and geometry dimensions as shown in figure

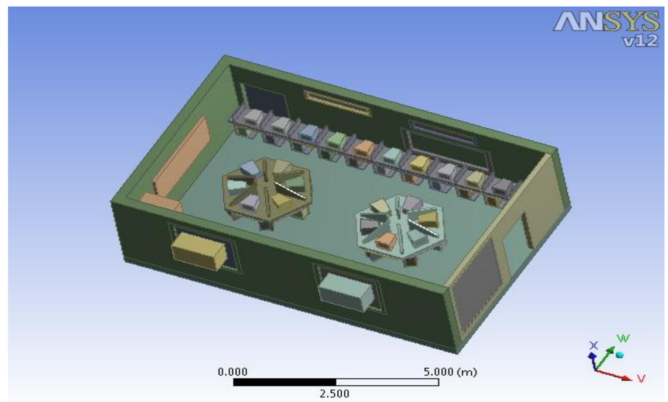


Figure 2.1 Geometry dimension of our solution domain

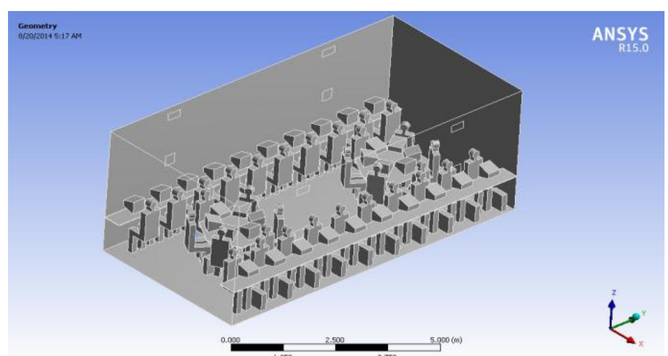


Figure2.2 Geometry for CFD analysis

Meshing

The meshing of cad model is symmetrically and elementary length is 0.1 m and mesh type is tetrahedral.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 4, Issue 11, November 2015)

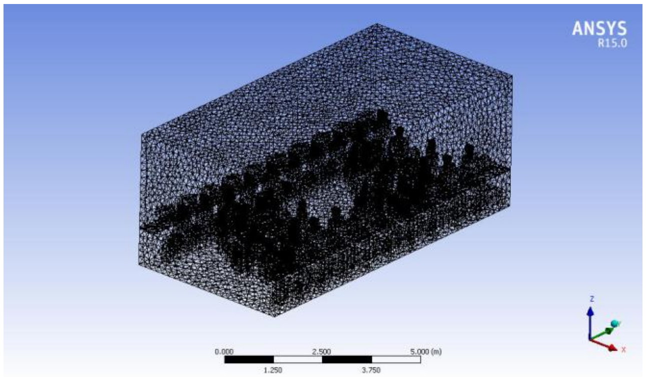


Figure 2.3 Meshing of domain air distribution system

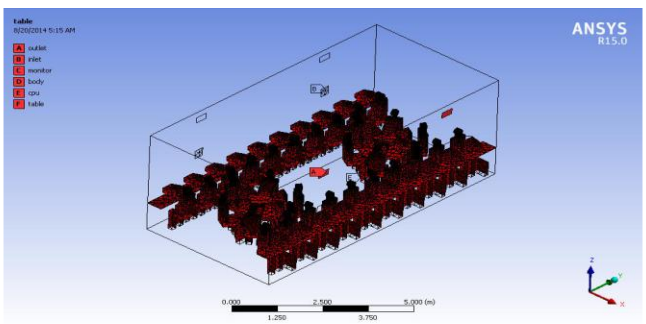
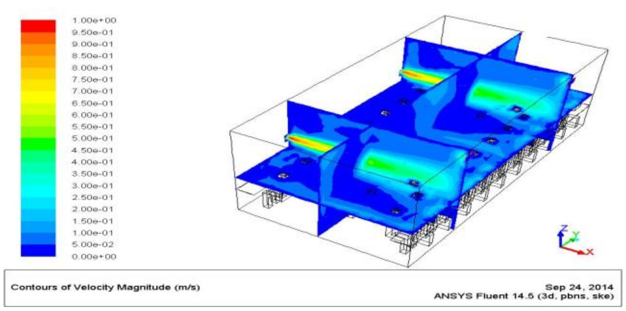


Figure 2.4 Meshing of interior involved in analysis

No. of nodes=107360
No. of elements= 600782

III. RESULTS AND DISCUSSIONS

3.1 Result of rainy season air distribution system



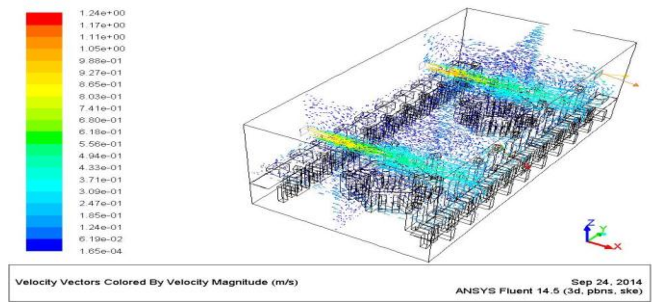


Figure 3.1 Analysis of air velocity and distribution

Graph between velocity and temperature

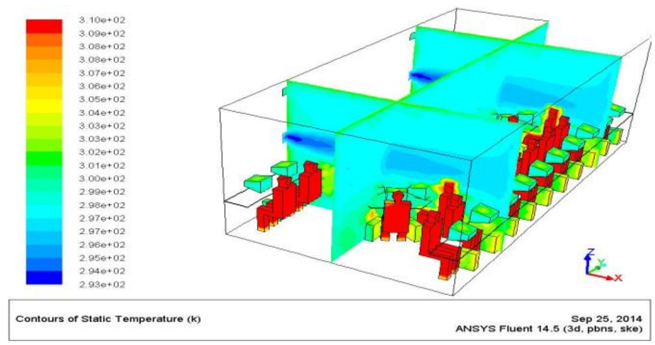


Figure 3.2 Analysis of air temperature

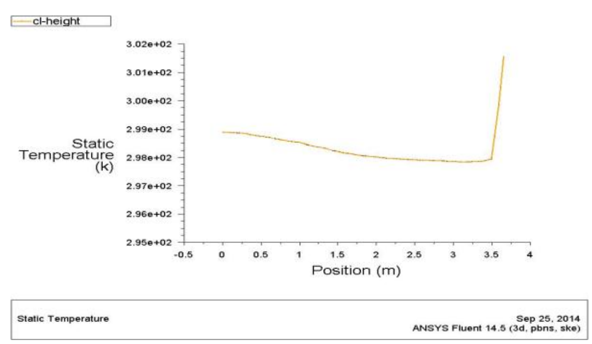


Figure 3.3 Analysis of air temperature with respect to ceiling height

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 4, Issue 11, November 2015)

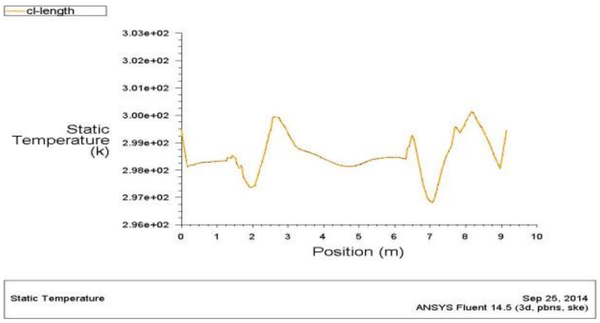


Figure 3.4 Analysis of air temperature with respect to ceiling length

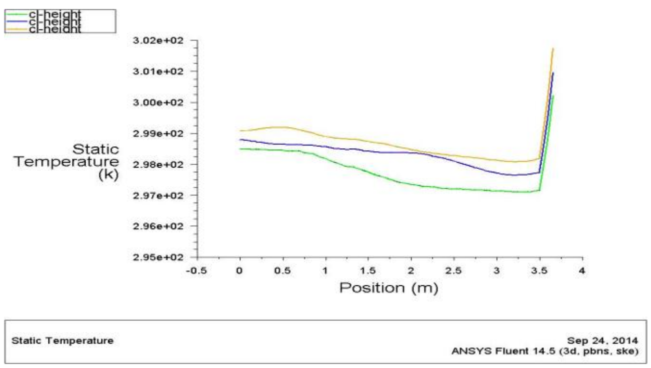


Figure 3.5 Analysis of air temperature with respect to ceiling height at different velocities

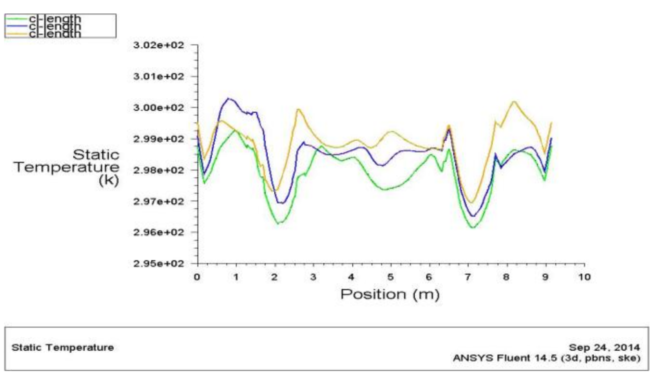


Figure 3.6 Analysis of air temperature with respect to ceiling length at different velocities

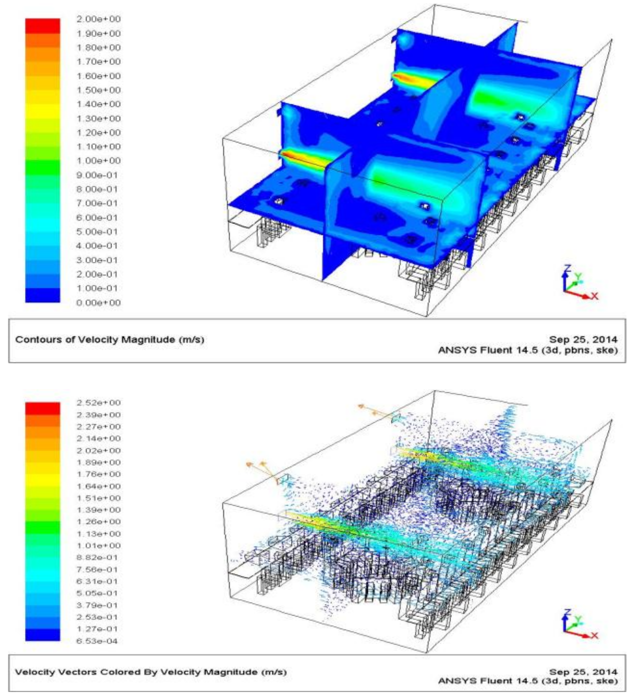


Figure 3.7 Analysis of air velocity and distribution

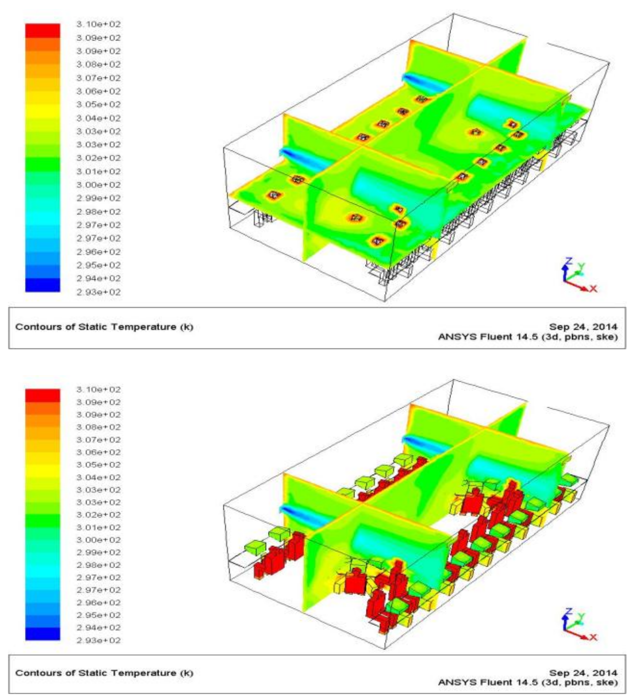


Figure 3.8 Analysis of air temperature

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 4, Issue 11, November 2015)

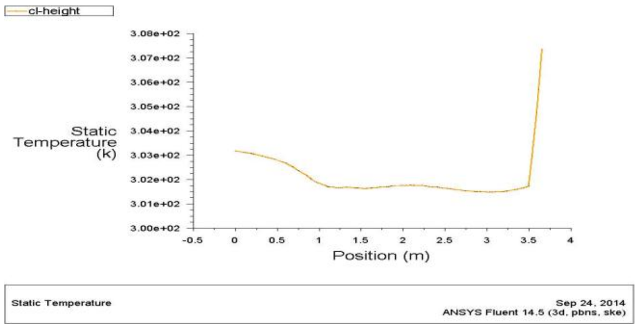


Figure 3.9 Analysis of air temperature with respect to ceiling height

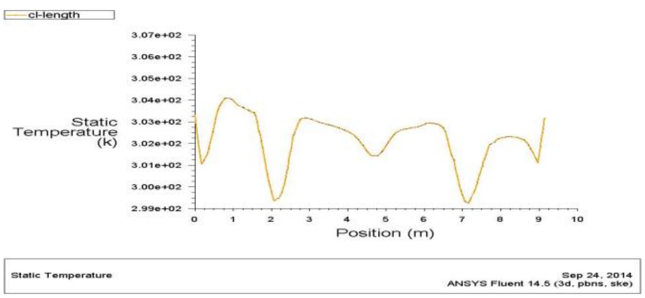


Figure 3.10 Analysis of air temperature with respect to ceiling length

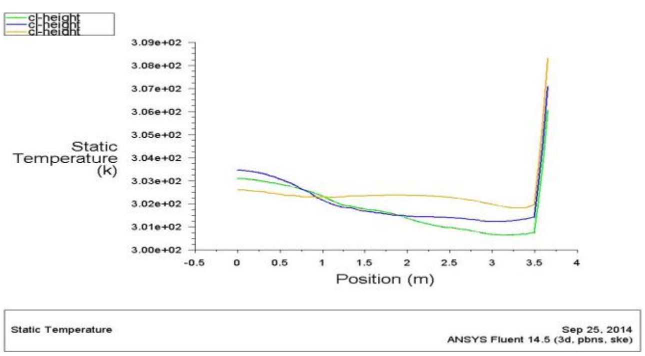


Figure 3.11 Analysis of air temperature with respect to ceiling height at different velocities

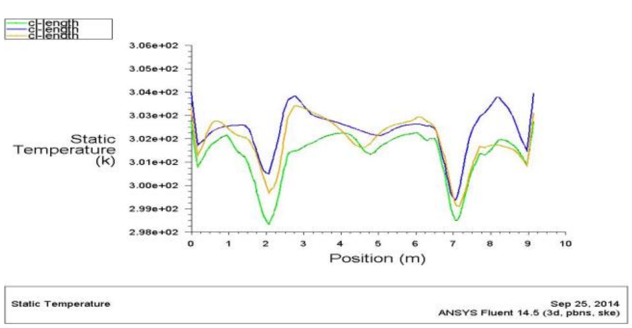
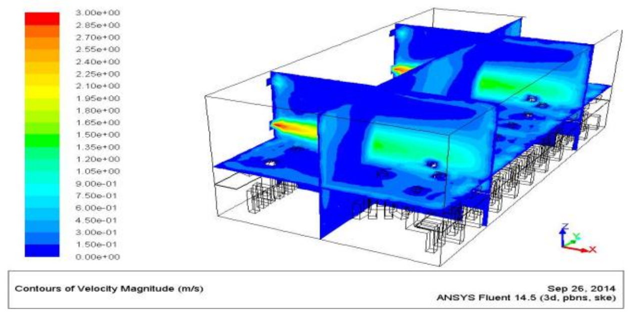
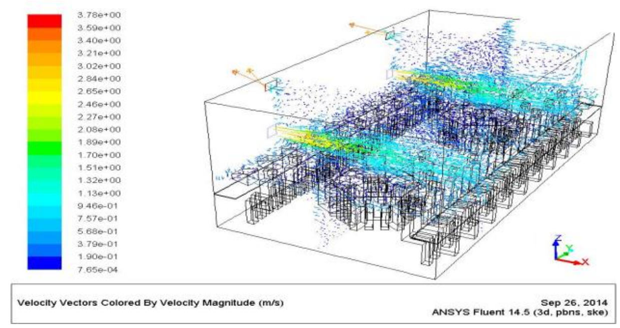


Figure 3.12 Analysis of air temperature with respect to ceiling length at different velocities

3.2 Results of winter season air distribution system





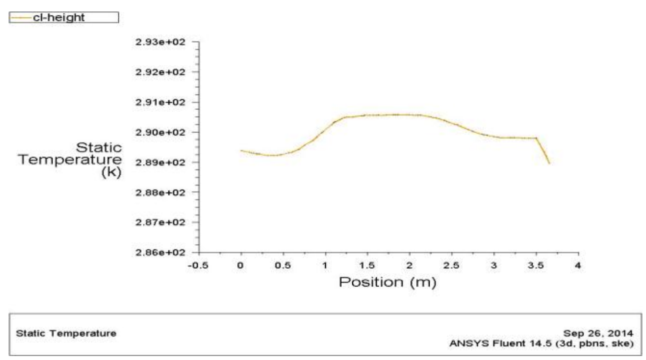


Figure 3.13 Analysis of air temperature with respect to ceiling height

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347 - 6435 (Online)) Volume 4, Issue 11, November 2015)

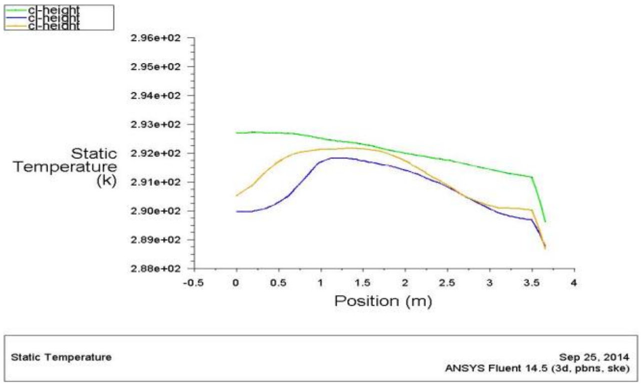


Figure 3.14 Analysis of air temperature with respect to ceiling height at different velocities

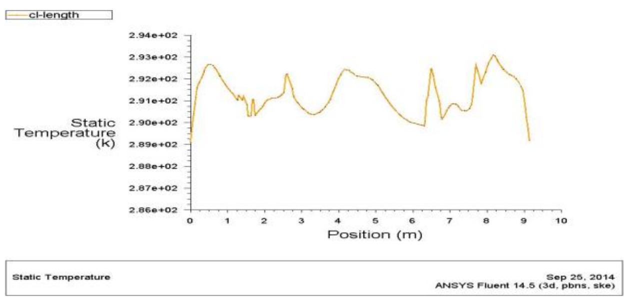


Figure 3.15 Analysis of air temperature with respect to ceiling length

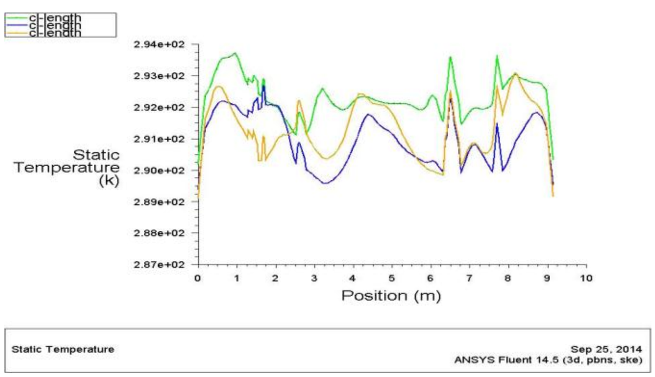


Figure 3.16 Analysis of air temperature with respect to ceiling length at different velocities

IV. CONCLUSION

There are several different points that must be taken into account while determining a mechanical system. The velocity of air and the position of air system in terms of height and length will be able to provide the elasticity required for future technology within the facility and is favorable on an operating cost basis.

For this reason, enthusiastic recommendations come from the project team which discuss the priorities and determine which option suits the project better. This study used the methodology and field measurements in order to develop a 3D CFD model which gives complete vision on environmental conditions of a naturally ventilated office meeting room. Moreover, the validated CFD model and field measurements were used to evaluate thermal comfort index for the occupants of the meeting room. The satisfactory and optimal operation of thermal environment has been confirmed. This paper is done in order to study the indoor airflow characteristics and thermal comfort of a room with nearby window openings under general position of window openings.

All range of the analysis will allow good resident comfort. Comfort and control both are achieved in different conditions depending on the type of work inside the system. The air velocity fields obtained from the CFD study show that a local discomfort at feet/ankle zone was observed but with good values of air velocity for the rest of the test room. The results reported confirm that the cooling ceiling has advantages like low vertical air gradient and that the thermal comfort is obtained even for higher metabolism rates or clothing insulation. The specification of confirmation criteria based on the purpose of the model proved to be a clear and straightforward way of validating CFD results with measurements. Solar radiation played a significant role in the highly-glazed room, even if only the diffuse part of solar irradiance was considered in the analysis. The indoor air temperature was mostly influenced by the inlet velocity vertical component, solar radiation through the windows, inlet velocity horizontal component and outdoor air temperature. In short, the CFD model calibration methodology bridges the gap between understanding and application of CFD simulation and field measurement for natural ventilation systems. This is done through a systematic utilization of the best practice guidelines and standards. This study was conducted to provide enhanced predictions to implement design strategies for radiant cooling ceilings with mechanical ventilation systems employing CFD, and showed that the proposed system operated with reasonable satisfaction.

REFERENCES

[1] Guangyu Cao , HazimAwbi , Runming Yao, Yunqing Fan , Kai Sirén , RistoKosonen ,Jianshun (Jensen) Zhang , A review of the performance of different ventilation and airflow distribution systems in buildings, Building and Environment 73 (2014) 171-186.

[2] Li Yang , Miao Ye, Bao-Jie he, CFD simulation research on residential indoor air quality, Science of the Total Environment 472 (2014) 1137–1144.

[3] Delia D'Agostino*, Paolo Maria Congedo, CFD modeling and moisture dynamics implications of ventilation scenarios in historical buildings, Building and Environment 79 (2014) 181-193.

[4] Simon J. Rees a, Philip Haves , An experimental study of air flow and temperature distribution in a room with displacement ventilation and a chilled ceiling, Building and Environment 59 (2013) 358-368.

[5] Xiaozhou Wu a,b, Bjarne W. Olesen b, Lei Fang b, Jianing Zhao a,*, A nodal model to predict vertical temperature distribution in a room with floor heating and displacement ventilation, Building and Environment 59 (2013) 626-634.

[6] Tomohiro Kobayashi, TomoyukiChikamoto, KeishiOsada, Tomohiro Kobayashi, TomoyukiChikamoto, KeishiOsada, Building and Environment 63 (2013) 20-30

[7] R. Tomasi, M. Krajˇcik , A. Simonec, B.W. OlesencExperimental evaluation of air distribution in mechanically ventilated residential rooms: Thermal comfort and ventilation effectiveness, Energy and Buildings 60 (2013) 28–37.

[8] Wei-Hwa Chiang, Chia-Ying Wang, Jian-Sheng Huang, Evaluation of cooling ceiling and mechanical ventilation systems on thermal comfort using CFD study in an office for subtropical region, Building and Environment 48 (2012) 113-127.

[9] Fatima ZohraChafi, Stephane Halle, Three dimensional study for evaluating of air flow movements and thermal comfort in a model room: Experimental validation, Energy and Buildings 43 (2011) 2156–2166.

[10] R. Whalley, A. Abdul-Ameer, Heating, ventilation and air conditioning system modelling, Building and Environment 46 (2011) 643-656.

[11] Fatima ZohraChafi, StéphaneHallé, Evaluating of air flow movements and thermal comfort in a model room with Euler equation: Two dimensional study, Building and Environment 46 (2011) 448-456

[12] K.-S. Kwon ,d, I.-B. Lee a , H.-T. Han, C.-Y. Shin, H.-S. Hwang, S.-W. Hong ,Jessie. P. Bitog , I.-H. Seo , C.-P. Han , Analyzing ventilation efficiency in a test chamber using age-of-air concept and CFD technology, i o s y s t ems engi n e e r i n g 1 1 0 (2 0 1 1) 4 2 1 -4 3 3

[13] A.C.K. Lai, K.W. Mui, L.T. Wong, L.Y. Law, An evaluation model for indoor environmental quality (IEQ) acceptance in residential buildings, Energy and Buildings 41 (2009) 930–936.

[14] Anita Coulter Flowe, Ashok Kumar, Analysis of velocity fields and dispersive cavity parameters as a function of building width to building height ratio using a 3-D computer model for squat buildings, Journal of Wind Engineering and Industrial Aerodynamics 86 (2000) 87-122.

[15] G. Einberg, K. Hagstrom, P. Mustakallio, H. Koskela, S. Holmberg, CFD modelling of an industrial air diffuser—predicting velocity and temperature in the near zone, Building and Environment 40 (2005) 601–615.

[16] H. Lee, H.B. Awbi, E ect of internal partitioning on indoor air quality of rooms with mixing ventilation—basic study, Building and Environment 39 (2004) 127 – 141.

[17] K. Visagaval, P.S.S. Srinivasan, Analysis of single side ventilated and cross ventilated rooms by varying the width of the window opening using CFD, Solar Energy 83 (2009) 2–5.

[18] K.W. Mui, L.T. Wong, W.L. Ho, Evaluation on sampling point densities for assessing indoor air quality, Building and Environment 41 (2006) 1515–1521.

[19] MiroslawZukowski, Modeling and designing heating and ventilation system withunderfloor air distribution, Energy and Buildings 38 (2006) 600–609

[20] Liang Zhou, FariborzHaghighat, Optimization of ventilation system design and operation in office environment, Part I: Methodology, Building and Environment 44 (2009) 651–656.

[21] L.T. Tan, S.C. Sekhar , Optimization of cooling coil performance during operation stage for improved humidity control, Energy and Buildings 41 (2009) 229–233.

[22] L.T. Wong_, K.W. Mui1, K.L. Shi, Energy impact of indoor environmental policy for air-conditioned offices of Hong Kong, Energy Policy 36 (2008) 714–721.

[23] M.A. Hassan, N.M. Guirguis, M.R. Shaalan, K.M. El-Shazly, Investigation of effects of window combinations on ventilation characteristics for thermal comfort in buildings, Desalination 209 (2007) 251–260.

[24] Monika Woloszyn, Joseph Virgone, Stephane Melen, Diagonal air-distribution system for operating rooms: experiment and modeling, Building and Environment 39 (2004) 1171 – 1178.

[25] N.P. Gao, J.L. Niu, M. Perino, P. Heiselberg, The airborne transmission of infection between flats in high-rise residential buildings: Tracer gas simulation,Building and Environment 43 (2008) 1805-1817.

[26] P. Rohdin, B. Moshfegh, Numerical predictions of indoor climate in large industrial premises.A comparison between different k–e models supported by field measurements, Building and Environment 42 (2007) 3872–3882.

[27] P.S. Hui, L.T. Wong, K.W. Mui, Evaluation of professional choice of sampling locations for indoor air quality assessment, Building and Environment 42 (2007) 2900–2907.