Indoor Air Quality Optimization by Thermal Displacement Air-Conditioning Methods

S. Shanmuga Priya¹, M. Premalatha² and I. Thirunavukkarasu³

¹Department of Chemical Engineering, Manipal Institute of Technology, Manipal University, Manipal - 576104, Karnataka, India; shanpriyanitt@yahoo.co.in, shan.priya@manipal.edu
²Department of Energy and Environment, National Institute of Technology, Tiruchirappalli – 620015, Tamil Nadu, India; latha@nitt.edu,
³Department of Instrumentation and Control Engineering, Manipal Institute of Technology, Manipal University, Manipal - 576104, Karnataka, India; it.arasu@manipal.edu

Abstract

Background/Objectives: We have intended to investigate air velocity, CO₂ concentration and temperature distribution by computational fluid dynamics software for comfort issues. Methods/Statistical Analysis: For designing purpose, the experimental data on temperature, humidity and CO₂ concentration were collected for an internet room. The same geometry of the room is simulated with same existing method of mixing ventilation. The profile of the experiment and the simulation are found as same for CO₂ concentration and Temperature. Findings: Air supply is a critical factor to evaluate the heat distribution, air velocity and solid contaminants inside the room. The air supply diffuser location, contaminant source location and air supply method are tested numerically by comparing mixing ventilation and displacement ventilation methods. ANSYS CFX is used to obtain the temperature, velocity, and CO₂ concentration. In this work the displacement ventilation method shows better distribution of air and the ventilation efficiency with less consumption of electrical energy for cooling load compared with the existing mixing ventilation method. Applications/Improvements: In the present system the temperature of inlet air is 15°C but if the system is modified to displacement ventilation efficiency the temperature of inlet air is 18°C and this itself provide better comfort condition. Hence the saving of electrical energy is also possible.

Keywords: Computational Fluid Dynamics, Displacement Air-Conditioning, Humidity, Indoor Air Quality, Temperature

1. Introduction

The important objective in design of the buildings we live and work and the transport we travel is ensuring the Indoor Air Quality (IAQ). Ramy H. Mohammed¹ investigated the characteristics of airflow, indoor air quality (IAQ), and thermal comfort in a room which has displacement ventilation system using CFD. W. Chakrouna et.al.² has conducted experiments to recorded time of the air temperature and CO₂ concentration and compared with values predicted by the model and it matches well.

Atila Novoselac et.al.³ has tested that building airflow, thermal and contaminant simulation programs for Displacement Ventilation (DV) systems.

Zhang Lin et. al.⁴ has carried out a numerical simulation using CFD and compared the performance of mixing and displacement ventilation with importance on indoor air quality. Shengwei Wang et. al.⁵ has conducted CFD simulation results to evaluate the construction of splitter vanes in the airflow channels and scrolls of the fans. Yuanhui Zhang⁶ has used the approach of using the CFD before and after fan retrofitting. Netra Naik⁷ has studied the indoor environmental quality effect on shopping areas with a focus on shopping arcades.

2. Evaluation of Indoor Air Quality - CFD Simulation

The model considered for an internet room in a computer centre for a hot and dry region. The indoor air quality is evaluated in the internet room of the computer centre.
Indoor Air Quality Optimization by Thermal Displacement Air-Conditioning Methods

Length of the room = 11.6 m
Width of the room = 5.2 m
Height of the room = 2.5 m
Window size (3 no’s) = 1 x 2 m
No. of persons = 36
No. of computers = 36
No. of Lights = 5
Total flow rate of Air in the Room = 3907 m$^3$/hr
No. of supply Grill (inlet Diffusers) = 10
Size of inlet = 0.31 x 0.31m
Opt. 3 No of Return Grill (outlet) = 12
Size of outlet = 0.56 x 0.15 m
$\mathrm{CO}_2$ concentration outside the room = 240 ppm
$\mathrm{CO}_2$ concentration near the entrance of room = 300 ppm
$\mathrm{CO}_2$ concentration in the corridor of the room = 1000 ppm

The ventilation of the room is the mixing ventilation, the indoor air quality in the internet room is above the acceptable limits. More amount of concentration is in the occupied zone about 1900 ppm. Also the temperature distribution is not uniform inside the room. Air distribution is also not uniform. Some amount of inlet air is bypass through the return grill.

CFX Simulation is done for three models
Model 1 ----- Mixing Ventilation
Model 2 ----- Displacement ventilation with wall mounted diffuser
Model 3 ----- Displacement ventilation with floor mounted diffusers

2.1 Creating the Material
Material 1 - Oxygen
Material 2 - Nitrogen
Material 3 - $\mathrm{CO}_2$
Material 4 - Water vapour at 25
Material 5 - Air mixture ($\mathrm{O}_2 + \mathrm{N}_2 + \mathrm{CO}_2 + \mathrm{Water vapour}$)

2.1.1 Creating the Domain

2.1.1.1 General option
Name : Room
Location : Assembly
Domain Type : Fluid Domain.

2.2 Fluid Models
Heat Transfer Model : Thermal Energy.
Reaction Or Combustion Model : None
Turbulent Wall Functions : Scalable.
Thermal Radiation Model : Monte Carlo.

2.2.1 Component Details
$\mathrm{CO}_2$ : Transport equation
Kinematic Diffusivity : $1.25 \times 10^{-5}$ m$^2$/s
$\mathrm{O}_2$ : Transport equation
Kinematic Diffusivity : $2.27 \times 10^{-5}$ m$^2$/s
Water Vapour : Transport Equation
Kinematic Diffusivity : $4 \times 10^{-5}$ m$^2$/s
$\mathrm{N}_2$ : Constraint

2.3 Boundary Conditions
2.3.1 INLET Boundary Condition
There are 10 inlets in the room

2.3.1.1 Basic Settings
Boundary Type : Inlet
Location : Inlet

2.3.1.2 Boundary Details
Flow Regime Subsonic.
Mass and Momentum : Mass Flow Rate : 0.14105 kg/s
Turbulence : Medium (Intensity = 5%).
Heat Transfer:
Static Temperature : 288 k
Thermal Radiation : Local Temperature.

2.4 Component Details
$\mathrm{CO}_2$ : Mass fraction $600 \times 10^{-6}$
$\mathrm{O}_2$ : Mass fraction 0.232
Water vapour : Mass fraction 0.017
2.4.1 OUTLET Boundary Condition
There are 12 outlet in the domain

2.4.1.1 Basic Settings
Boundary Type to Outlet
Location to outlet

2.4.1.1.1 Boundary details
Flow Regime : Subsonic.
Mass and Momentum : Average Static
Pressure and Relative pressure : 0 pa
Thermal Radiation : Local Temperature.

2.4.2 CO₂ INLET Boundary Condition
There are 36 (Students) CO₂ inlets in the room

2.4.2.1 Basic settings
Boundary Type : Inlet
Location : CO₂ Inlet

2.4.2.2 Boundary details
Flow Regime Subsonic
Mass and Momentum:
Mass Flow Rate : 0.00001487kg/s
Turbulence : Medium (Intensity = 5%).
Heat Transfer:
Static Temperature : 303 k
Thermal Radiation : Local Temperature.

2.5. Component details
CO₂ : Mass fraction : 1
O₂ : Mass fraction : 0
Water vapour : Mass fraction 0

2.5.1 WINDOW Boundary Conditions
To model incoming radiation at the window boundaries, a directional radiation source will be created. The windows will also contribute heat to the room via a fixed temperature of 26 °C.

2.5.1.1 Basic Settings
Boundary Type : Wall
Location : Windows

2.5.1.1.1 Boundary details
Wall Influence On Flow : No Slip.
Wall Roughness : Smooth Wall.
Heat Transfer : Temperature and Fixed
Temperature to 26 °C
Thermal Radiation:
Radiation Source : Isotropic Radiation Flux
Radiation Flux : 400 [W m^-2].

2.6 COMPUTER Boundary Condition
The computers will also contribute heat to the room via a fixed temperature of 40°C

2.6.1 Basic Settings
Boundary Type : Wall
Location : System

2.6.2 Boundary Details
Wall Influence On Flow : No Slip.
Wall Roughness : Smooth Wall.
Heat Transfer : Temperature and Fixed
Temperature to 40 °C
Thermal Radiation:
Radiation Source : Isotropic Radiation Flux
Radiation Flux : 40 [W m^-2].

2.7 LIGHTS Boundary Condition
The lights will also contribute heat to the room via a fixed temperature of 45 °C

2.7.1 Basic Settings
Boundary Type : Wall
Location : Lights

2.7.2 Boundary Details
Wall Influence On Flow : No Slip.
Wall Roughness : Smooth Wall.
Heat Transfer : Temperature and Fixed
Temperature to 45 °C
Thermal Radiation:
Radiation Source : Isotropic Radiation Flux,
Radiation Flux : 60[W m^-2]

2.8 WALL Boundary Conditions
Fixed temperature of 26 °C will be assumed to exist at the wall during the simulation.
2.8.1 Basic Settings
Boundary Type : Wall
Location : Walls, floor

2.8.2 Boundary Details
Wall Influence On Flow : No Slip.
Wall Roughness : Smooth Wall.
Heat Transfer : Temperature and Fixed
Temperature to 24 °C

2.9 Table Boundary Conditions
2.9.1 Basic Settings
Boundary Type : Wall
Location : Walls, floor

2.9.2 Boundary Details
Wall Influence on Flow : No Slip.
Wall Roughness : Smooth Wall.
Heat Transfer : Temperature; Fixed
Temperature to 24 °C

2.10 HUMAN Boundary Conditions
Human beings also emit heat

Basic Settings
Boundary Type : Wall
Location : Human

Boundary Details
Wall Influence On Flow : No Slip.
Wall Roughness : Smooth Wall.
Heat Transfer : Temperature; Fixed
Temperature to 26 °C

2.11 Global Initialization
Velocity Type : Cartesian.
Cartesian Velocity Components,
U : 0 [m s⁻¹]
V : 0 [m s⁻¹]
W : 0 [m s⁻¹]
Static Pressure : Relative Pressure to 0 [Pa].
The room will be at an initial temperature of 23 °C
Temperature : 23 °C
Turbulence Kinetic Energy : Fractional Intensity: 0.05.
Turbulent Eddy Dissipation : Eddy Length Scale : 0.25 [m].
Blackbody Temperature : 22 °C

2.12 SOLVER
Advection Scheme : High Resolution.
Transient Scheme : First Order Backward Euler.
No of iteration : 500

For displacement ventilation give the inlet temperature as 18°C. The total flow rate is 1.410 kg/s. The model is created by using ANSYS ICEM CFD Simulation and is shown in Figure 2-4.

3. Results and Discussion

For designing, the experimental data on temperature, humidity and CO₂ concentration were collected from an internet room. The same geometry of the room is simulated with same existing method of mixing ventilation. The profile of the experiment and the simulation are found as same for CO₂ concentration and Temperature. The air supply method is an important factor for distribution of heat, air velocity and solid contaminants inside the room. The influence of air supply diffuser location, contaminant source location and air supply method are tested numerically by comparing the mixing ventilation and displacement ventilation methods. Figure 1 shows the layout of an internet room.

A commercial CFD code, ANSYS CFX is used to obtain the temperature, velocity, and CO₂ concentration. In this work the displacement ventilation method shows better distribution of air and the ventilation efficiency with less consumption of electrical energy for cooling load compared with the existing mixing ventilation method.

Figure 1. Layout of internet room.
Simulation is done for three models as shown below:

Figure 2. Model 1 - Mixing ventilation.

Figure 3. Model 2 – Displacement ventilation with wall mounted diffusers.

Figure 4. Model 3 – Displacement ventilation with floor mounted diffusers.

The velocity, temperature, CO$_2$ concentration distribution inside the room is analyzed using CFX POST.

Figure 5. Height vs. Temperature–Mixing.

Figure 6. Height vs. ventilation O$_2$ concentration.

From the Figure 5 and Figure 6, the temperature distribution for the mixing ventilation is not uniform and the supply air temperature is 15°C. But in the human occupied zone the temperature is around 24°C. The pollutant concentration is more in the occupied zone most of the pollutant will stagnant in the breathing zone itself.

Figure 7. Height vs. velocity.
Indoor Air Quality Optimization by Thermal Displacement Air-Conditioning Methods

From the Figure 7 and Figure 8, the velocity distribution in the occupied Zone is around 0.09 m/s. The temperature distribution for the Displacement ventilation is uniform the supply air temperature is 18°C But in the human occupied zone the temperature is around 24°C. There is rise in temperature from the floor level. The CO$_2$ concentration in the breathing zone is less. The pollution concentration is rising up to the outlet.

Internet room considered for the study showed that the mixing ventilation does not provide uniform temperature and air distribution, the CO$_2$ concentration is very high i.e. 1900 ppm. Where the outside concentration is 250 ppm. In this work numerical simulations have been carried out to study the effect of air distribution inside the room. The effect of temperature distribution CO$_2$ concentration and the velocity of air have been investigated for two cases mixing ventilation and displacement ventilation. Experimental measurements carried out for the temperature and CO$_2$ concentration showed the similar profile that is obtained in mixing ventilation model by CFX software.

Displacement ventilation is carried out by two methods one is wall mounted diffusers and another one is floor mounted diffusers. The simulated results of displacement ventilation of both wall mounted and floor mounted diffusers shows better air distribution, temperature and CO$_2$ concentration compared with existing model. However for economical point of view it is suggested that displacement ventilation with wall mounted diffusers will improve the comfort conditions. The present system the temperature of inlet air is 15°C but if the system is modified to displacement ventilation efficiency the temperature of inlet air is 18°C and this itself provide better comfort condition.

5. References

1. Mohammed RH. Numerical investigation of indoor air quality and thermal comfort in a ventilated room. International Journal of Environmental, Chemical, Ecological, Geological and Geophysical Engineering. 2013; 7(12):876-82.
2. Chakrouna W, Ghali K, Ghaddar N. Air quality in rooms conditioned by chilled ceiling and mixed displacement ventilation for energy saving. Energy and Buildings. 2011; 43:2684–95.
3. Novoselac A, Burley BJ, Srebric J. Development of new and validation of existing convection correlations for rooms with displacement ventilation systems. Energy and Buildings. 2006; 38:163-73.
4. Lin Z, Chow TT, Fong KF, Tsang CF, Wang Q. Comparison of performances of displacement and mixing ventilations. Part II: Indoor air quality. International Journal of Refrigeration. 2005; 28:288–305.
5. Wang S, Zhub D. Application of CFD in retrofitting air-conditioning systems in industrial buildings. Energy and Buildings. 2003; 35:893–902.
6. Zhang Y. Indoor air quality. Engineering. CRC–PRESS; 2005.
7. Naik N, Dhamankar N, Karve S. Assessing the indoor environmental quality in shopping areas with a focus on shopping arcades case of Pune. Indian Journal of Science and Technology. 2015; 8(28):IPL0762.