Numerical analysis of displacement ventilation for different locations of the outlets

A H Naidu, A Singh, T V S Yaswanth Mohan, K K Murthy, M Mallela, R Choudhary

School of Mechanical Engineering, Lovely Professional University, Punjab (India) - 144402

E-mail: rajesh.22677@lpu.co.in

Abstract. In the earlier phase of construction of simple buildings, natural ventilation was the best ventilation method used in those times. It was a simple system which includes windows, doors, chimneys etc. where forces like buoyancy force acts due to thermal and density variations of indoor and outdoor air. Later, due to increase in working population in offices, industries, natural ventilation failed to satisfy the comfort levels of the working people. A combination of mixing ventilation and natural ventilation introduced satisfying industries and large shopping outlets in both comfort levels and also improved air distribution quality guaranteeing the safety of people working by clearing out pollutants efficiently. But the shortcomings of this ventilation were such that the higher jet velocities causing loud noises, less efficiency causing pollutants concentration to rise in air, etc. Therefore, at places where noiseless, efficiency and moderate comfort is preferred displacement ventilation came in the present scenario. Displacement ventilation supplies cold air at low velocities from the floor level, while hot air rises to the ceiling due to buoyancy while hot air escapes through the exit outlets placed at proper positions to create a way for the circulation of fresh air. The displacement ventilation in an office model with different variations that depend on heat sources, outlet positions etc. is studied in the present study. Different cases have been simulated for the different locations of the outlet and number of heat sources.

1. Introduction
The displacement ventilation is a widespread use in hospitals, office buildings, airports etc. It is a passive mechanical type of ventilation. Displacement ventilation has serious advantage take over by achieving low loading conditions. This is possible by making stratification layer over the working height. This is due to buoyancy force of hot air which rises to the top of the ceiling which is due to positive temperature gradient or due to change in density of air molecules. Heat loads are taken into consider which forms a layer between cool air in stratification layer and hot air rise as heat plumes. The heat plumes act as a single point buoyancy force which was proposed by Sandberg & Lindstrom in 1990 [1]. Stratification layers develops two layers namely upper and lower layer in which upper layer is independent of height and only increases with increase of heat plumes. The lower layer is at uniform ambient temperature and upper layer is at uniform high temperature. The typical setup of displacement system has inlet diffuser, outlet exhaust and assuming height of stratification for good environment. The air enters the room at very low velocity and evenly distributed on the floor. This
makes positive temperature gradient to rise to the ceiling level and exhaust hot air to the outlet. This makes to create a comfortable environment within working space.

The primary function of ventilation is to maintain Indoor Air Quality (IAQ), while also maintaining the temperature of the room and the human body. Temperature of the human body alters due to metabolic activities conducted in the body and it tries to maintain it at approximately 37 °C despite the change in ambient temperature of the surroundings to a certain limit as evaporative cooling of sweat becomes ineffective at higher temperatures [2]. During these temperature changes body temperature might not change but skin temperature alters for a small change in ambient temperature. The challenge is to maintain the skin temperature at thermal equilibrium and comfort. Heat transfer occurs through convection, radiation, conduction, evaporation, respiration. It is further affected by clothing we wear, metabolic activity of the body.

Air temperature, mean temperature, air velocity, water vapor pressure, heat flux of the warm bodies are the important factors that must be taken into consideration while giving input to the experiment analysis. Displacement ventilation is the result of change in temperature at upper and lower regions of a space due to buoyancy and viscous forces acting on the air that change due to climatic conditions. Various factors are responsible for this change. Some of them are:

- Room layout
- Location of supply and return-air outlets within the room
- Air flow rates and characteristics of supply-air outlets
- Supply air temperature
- Internal heat generation due to occupants, electronic devices and their distribution within the room
- Convection currents generated by the wall’s due stratification
- Physical obstruction to the air movement
- Boundary conditions for the wall
- Stack effect
- Air leakages

1.1. Ventilation Rate
A lot of factors are taken into account while determining the ventilation rate for number of people and space availability. These include odor, water vapor, pollutants like carbon dioxide (CO2), carbon monoxide (CO), formaldehyde (CH2O), aerosol, ozone (O3), etc. As ventilation rate increases concentration of pollutants, odor concentration decreases to the safest level possible [2]. Quality of air should be maintained along with ventilation rate. Air inlets should be placed away from traffic pollutants which are at major concentration at lower areas of the land and there should be sufficient space between the buildings for a proper air circulation while taking ventilation design for the building into the account [3].

Airflow in displacement ventilation is influenced by thermal plumes. When the air jet is released from the outlet and passes through the heat sources at floor level and is exited from the upper zone. Due to the action of buoyancy, cold air spreads along the floor and distributes itself along the lower zone of the room. The cool air nearer to the heat source gets heated up and rises upward due to convection. This convection stream spreads along the ceiling. The boundary of lower occupant zone depends on the air discharged and number of heat sources and heat plumes released by them. This convective stream reaches to a certain height where balancing of the temperature gradient between lesser heat sources plumes and ambient air present in the room in the presence of more heat sources. Turbulence generated due to the heat sources can be neglected as it is very less when compared to the heat released by the sources with certain room ventilation parameters [4].

1.2. Energy Requirements
Diffuser should be able to alter the temperature, moisture content of the entering air while maintaining the proper circulation of air. Building materials, radiating gases, aerosols, Total volatile organic
compounds etc. concentration must be diluted while supplying air. This improves the ventilation effectiveness of the system which will be able to provide comfort to the people. A reduction in 2.4K of the indoor temperature can be tolerated by the occupants as per the studies conducted as per the ISO 7730 and ASHRAE standard 55 [5]. Proper positioning of the diffuser outlets ensures the supply of air to all the locations decreases the energy consumption of the system.

1.3. Vertical Temperature Gradient variation

It depends on the factors like room height, location and its climatic conditions, kind of workplace etc. An example can be given by the study of Porras-Amores et. al [6] on vertical temperature gradient variation in warehouses in Spain. They considered five kinds of warehouse facilities with and without air-conditioning above and below the ground. Average temperature was found to be around at 15.1˚C throughout the year in Spain. So, the air-conditioned warehouses were maintained at the temperature of 16˚C, while maintaining relative humidity between 75-85% throughout the year [6]. Non-conditioned warehouses undergo natural ventilation.

From the observations of the temperature gradient values throughout the year in the warehouses, temperature gradient increased in hotter months while decreased during winters. Stratification is highest at summers while lowest at winters. The main area is the ceiling where the highest heat transfer occurs as it is the most disclosed part of the system other than windows and doors. The same goes for the underground warehouses due to poor insulation of soil (dry conditions) covering the ceiling. During this period hot air goes up while cooler air settles down being more denser achieving the desired effect. Wall plumes are one of the most important factors that must be not neglected during summers. Temperature gradient is highest close to 0.7˚C/m in the case of underground warehouse due to less heat transfer, while in winters it was in range of 0-0.1˚C/m due to less outdoor temperature [6].

Air-conditioned facilities shown less changes in temperature gradient irrespective of the outdoor conditions while stratification is less during summers in non-air-conditioned warehouses and more in winters [6]. Ceiling temperature plays an important role in maintaining the temperature gradient near the ceiling region, but has less effect on the lower areas of the room space [7].

The air distribution has a significant effect on temperature gradient due to positioning of ventilation system. A down-supply up-return and down-supply down-return ventilation system were compared in the experimental study of under flow study of air distribution system in summer [7]. In down-supply up-return ventilation system Air inlets are positioned below while return outlets are positioned above the inlets. There was a significant change in temperature gradient variation at different heights. More variations are seen in lower cooler areas of the room in the case down-supply up-return as there is more stratification in this case [7]. Return air outlets must be placed above 7 ft and below 1 ft of ceiling for better stratification as it eases the exhaustion of warm and pollutant air above the heat sources [8]. In the case of large spaces, it’s better to evenly position the exhaust outlets above the heat sources.

Airflow is affected by contaminant movement with respect to their origins. They are originated from non-buoyant sources like from the paint of the wall which is known as contaminant diffusion occurs in all the directions, buoyant forces due to heat sources in the confined space that occur due to temperature differences that leads to generation of buoyant forces while the third one is dynamic source where contaminants flow with higher speeds like in the case of emissions from the metal cutting machines.

Airflow near the hood of the outlet can be influenced by other factors like eddies generated by turbulent air flow, convection flow due to buoyancy forces, number of occupants, airflow from the windows and doors. Enclosing hood outlets are good at controlling contaminant flow when compared to non-enclosing hood geometry. Enclosed hood outlets are efficient and at the same economical.

An CFD experiment conducted on a model based on office building by Menchaca-Brandan in simulation software with different Archimedes number and different occupants in different cases [9]. Archimedes numbers of 60 and 0.5 were used with 3 and 5 occupants office models. In this CFD
analysis inlet temperature is given as 19°C and suitable inlet velocity is maintained constant throughout the analysis, atmospheric pressure was maintained. Ceiling with a heat flux of 8 W/m² and other walls are adiabatic. Heat emitted by a sitting occupant is given as 120W at 1 metabolic rate. They used Shear stress transport k-omega is used for its better performance because of supply jet behavior in this case. From the above formula it is clear that Archimedes number is inversely proportional to the supply air velocity. At higher velocities, with lower Archimedes number increase the throw length of the jet along with more occupants resulting in Higher temperature gradient. In this case two-structured zone with upper-recirculating zone above heat sources and lower-stratified zone around heat source level due to convection from human metabolic activities (heat release). Heat radiation from the ceiling also affects the temperature gradient. At low Archimedes number case with low velocity jet stream, reaches the floor due to heat plumes released by occupants as warm air is denser than cool air. This results in forming of up-recirculating zone and lower-stratified zone due to buoyancy forces and larger throw.

2. Data Reduction

Procedure and calculation are done as per as per the ASHRAE research project-949 and ASHRAE manual method [6]. A temperature gradient of 2°C/m is assumed and following loads are considered in this analysis.

The heat released by occupants, Q_o (W)
The heat conduction through the room Q_c (W)
Overhead lightning is assumed to be zero due to design complications

Cooling Load Ventilation Flow Rate: The air flow rate required for cooling,

\[ Q_{dy} = \frac{0.295}{60c_p\rho c_f} + 0.185 Q_c + 0.132Q_i \]  

(1)

Flow Rate of Fresh Air, Q_{oz}

\[ Q_{oz} = \frac{Q_{dy} + \rho c_f d x z}{\rho c_f} \]  

(2)

Required Supply Air Flow Rate, Q_s: Choose the greater of the required flow rate for cooling loads

\[ Q_s = \max(Q_{dy}, Q_{oz}) \]  

(3)

Supply Air Temperature, t_s: The supply air temperature is given by

\[ t_s = t_{sp} - \Delta t_{sh} - \frac{\rho c_f d x z}{2.45Q_{dy} + L 0.08Q_s} \]  

(4)

Exhaust Air Temperature, t_e

\[ t_e = t_s + \frac{Q_e}{1.08Q_s} \]  

(5)

Based on the above calculation for the selected space and thermal comfort conditions, it is found that the minimum supply air temperature should be at 17.2°C.

3. Problem Description

The basic model of office is taken for analysis which has the dimensions 9 m length, 4m width and 3 m height. The analysis is done in two different arrangements of outlet positions with two and three heat sources and observations are noted accordingly. The space of the geometry is considered as fluid (air) and three heat sinks (models) are placed evenly 1.83 m apart where middle one is standing at the center of the room.

Case 1: Inlet is placed at bottom having ground clearance of 0.1 m. The outlet is on top far away from inlet (prevents short circuiting). It has clearance of 0.3 m from another end of wall from inlet, as shown in figure 1. It has three heat sources, and the parameters are mentioned in table 2.
Case 2: Inlet is placed at bottom having ground clearance of 0.1m. The outlet is on top and exactly at the center of the room. It has clearance of 4 m from another end of wall from inlet, as shown in figure 2. It has three heat sources, and the parameters are mentioned in table 2.

Case 3: The geometry in this case is similar to the case 1, except that one of the heat body is suppressed, which is near to the inlet, shown in figure 3.

Case 4: The geometry in this case is similar to the case 2, except that one of the heat body is suppressed, which is near to the inlet, shown in figure 4.
Figure 4: Three – dimensional view of geometry used in case – 4

3.1. Meshing
The space in which heat body is placed and to be ventilated, is considered as fluid and meshed, as shown in figure 5. Mesh independence test was conducted for the default fine and coarse mesh quality, and a negligible variation in the temperature was found. Therefore, coarse mesh has been used for the simulation for accurate results at less computational time.

Figure 5: Three – dimensional view of the space to be ventilated with fine mesh.

All the cases, described in the previous section, are meshed and shown in the below figure 6.

Figure 6: View of the meshing of geometry used in case – 1.
4. Thermo-physical Properties and Boundary Conditions

The properties of air are taken as shown in table 1 and the boundary conditions at the inlet, outlet and walls are mentioned in table 2.

| Table 1: Thermo-physical properties of air taken for the CFD analysis |
|-----------------------------|-----------------------------|
| Density of air | 1.225 kg/m$^3$ |
| Temperature of air | 295.15 K |
| Specific heat | 1006.3 J/kg-K |
| Viscosity | 1.7894×10$^{-5}$ kg/m-s |
| Thermal conductivity | 0.0242 W/m-K |
| Velocity of air | (0.25, 0.4) m/s |

| Table 2: Boundary Conditions applied in the CFD analysis |
|-----------------------------|-----------------------------|
| **Inlet conditions** |
| Length | 0.75 m |
| Breath | 0.5 m |
| K-epsilon | 1 |
| **Outlet conditions** |
| Length | 0.5 m |
| Breath | 0.375 m |
| K and epsilon value | 1 |
| **Wall conditions** |
| Thickness | 0.1 m |
| Temperature | 300 K |
| Heat flux | 0.84 W/m$^2$ |
| **Heat source** |
| Thickness | 0.01 m |
| Diameter of model | 0.36 m |
| Height of the model | 1.68 m |
| Heat flux | 2.84 W/m$^2$ |
| Temperature of heat source | 310.15K, 304.15K |

The energy equation is on and the solving viscous equation is given as k-epsilon method. The $k$ – $\epsilon$ is a simple equation to solve where $k$ is used to solve turbulence energy and $\epsilon$ for the rate of change in turbulence energy. Two different cases have been analyzed in the Fluent software.

5. Results & Discussion

The simulation of the problem discussed in the previous sections along with the thermo-physical properties of air and boundary conditions have been conducted in the Fluent Software, and the contours and vectors obtained are discussed in the present section.

5.1. Temperature Contours

From the figure 7, it is observed that the air temperature is distributed evenly at most of the zones except nearer to the inlet and around heat sources. In this case as the outlet is at the other end, the warmer air moves along the walls moving towards the outlet while maintaining the temperature at isothermal state at most of the cross section of the room. It also shows that model which is assumed as human can be seen that the one closer to the inlet has cooler surface than the one at extreme. This shows that the skin of human is sensitive to the thermal changes as it comes in direct contact to the air. It also shows that bottom regions of the room section are cooler than upper regions.
Figure 7: Temperature contours for heat source at temperature of 310.15 K and inlet velocity of condition air equal to 0.25 m/s in Case – 1.

In figure 8, all the heat sources are at lower temperature than case – 1, while all being at the same temperature. Overall temperature of the room is colder than the case – 1. These two cases show that there is a need to take the account of heat loads while designing the ventilation system. The figure 9 shows that the regions around the heat sources are at same temperature range, following the foot to head of the heat sources.

Figure 8: Temperature contours for heat source at temperature of 304.15 K and inlet velocity of condition air equal to 0.25 m/s in Case – 1

Figure 9: Temperature contours for heat source at temperature of 304.15 K and inlet velocity of condition air equal to 0.25 m/s in Case - 3

From the above cases, figure 8 and figure 9 have similar air distribution profiles while their boundary conditions being different asper the temperature of human models being. This shows the effect of position of outlet on the temperature of air distribution. If there are more heat sources, it is better to keep the outlet far away from the inlet instead fitting them above the heat sources or ceiling. This shows that case of figure 8 is the better ventilation system keeping the occupants comfortable.
Figure 9 arrangement is applicable for loosely packed people like office rooms or cabins. These temperature profiles also show the effect of number of heat sources in the space. More heat sources in figure 8 and figure 9 gives some degree of comfort when compared to other cases. Although their skin temperature varies, but their temperature ranges in their particular rooms is closer. Depending on the number of heat sources the case – 1 is better system as it is able to provide better temperature distribution.

In the case – 4 shown in figure 10, one of the heat sources is at higher temperature than the other by a difference by 3 K. This shows that even though the temperature distribution is slightly even throughout the room but the higher heat release effects the ventilation system. It is unable to cool the heat loads completely as per the requirements.

5.2. Flow field velocity Vectors

It shows the flow pattern that is being altered at every obstruction it is facing and changing its flow field accordingly. The velocity vector profiles described, shown in figure 11 and 12 for case – 1 and case – 3, respectively, the flow pattern in different cases when there is an obstruction. In figure 12 there is less turbulence when compared to Figure 11. Uniform air flow can make the domain to obtain desired temperature distribution. The number of heat sources and size of the domain must be taken into control when deciding the displacement ventilation system.
6. Another section of your paper
In the present study, the effect of outlet location and number of heat sources in a space to be ventilated is simulated and the results have been discussed in the form of temperature contours and velocity profiles.

Based on the results obtained, following points are found as the conclusions:

- Displacement ventilation depends on ventilation loads i.e., number of people, load from electronic devices, chimneys etc. As the loads increases number of air inlets and outlets should be managed properly. Inlets must be placed at the floor level and outlets should be positioned far away from the inlet wall on the ceiling for better air distribution maintaining comfort levels.

- Case – 2 and 4, where the outlet is located at the center, offered better results in terms of temperature distribution and velocity and air flow when compared to other cases where outlet is placed at the center.

- Proper care must be taken when initiating the pre-processing. Geometry, describing the domain (fluid or solid), mesh generation, giving proper Boundary tags (labelling the named selections), boundary conditions. This decides the output of the solution. One wrong step might waste the time and resources. More than a difference of 0.1m/s was there with coarse and refined mesh. This shows the effect of meshing conditions on the solution.

- Age of air depends on the efficiency of the ventilation system. If efficiency is less, age of air increases. Which results in accumulation of pollutants deteriorating the IAQ. It is observed that the air quality increases when all the windows, doors or other openings of the buildings are opened. Age of air is highest in the months of September to October, while lowest in the month of January.

- Air jet spreads according to the type of obstruction. If it gets obstructed by heat source plume generation effects the air distribution generating buoyancy force. If it is obstructed by non-thermal body it spreads horizontally over it. No plume generation takes place in this case. Length of air jet is affected by obstruction.

- Displacement ventilation can only employ low velocities. Higher velocities lead to further lower air temperature gradient not comfortable to the occupants. An induced velocity of 0.4 m/s caused an increase 0.78 m/s which increase noise as well as required temperature gradient in the room will not be achieved as it becomes too cool to be comfort.

References
[1] Sandberg H and Lindstrom S 1987 Stratified flowing ventilated rooms—a model study, Proc. of ROOMVENT’90, Engineering Aero – and Thermodynamics of Ventilated Room, Second
International Conference, Oslo, Norway, June 13 – 15, 1987.

[2] Awbi H 1991 Ventilation of Buildings. London: Routledge, https://doi.org/10.4324/9780203476307.

[3] Hall D J, Spanton A M, Kukadia V and Walker S 2003 Exposure of buildings to pollutants in urban areas – a review of the contributions from different sources, The Effects of Air Pollution on the Built Environment, chapter 12 pp 351 – 391, doi: 10.1142/9781848161283_0012.

[4] Goodfellow H and Tähti E 2001 Industrial Ventilation Design Guidebook, Academic Press, ISBN No. 978-0-12-289676-7, doi: 10.1016/B978-0-12-289676-7.X5000-0.

[5] ASHRAE Standard 55 2004 Thermal environmental conditions for human occupancy. Atlanta, GA: American Society for Heating, Refrigerating and Air Conditioning Engineers.

[6] Porras-Amores C, Mazarrón FR and Cañas I 2014 Study of the vertical distribution of air temperature in warehouses, Energies, vol 7(3), pp 1193–1206. doi: 10.3390/en7031193.

[7] Wang X, Huang C, Cao W, Gao X and Liu W 2011 Experimental Study on Indoor Thermal Stratification in Large Space by under Floor Air Distribution System (UFAD) in Summer, Engineering (Scientific Research), vol 03(04), pp 384–388. doi: 10.4236/eng.2011.34044.

[8] Kreith F, Manglik R M and Bohn M S 2012 Principles of Heat Transfer, Cengage learning: Boston, MA, USA, 2012; ISBN 1133714854.

[9] Brandan M and Alejandra M 2012 Study of Airflow and Thermal Stratification in Naturally Ventilated Rooms. PhD thesis, Massachusetts Institute of Technology, http://hdl.handle.net/1721.1/74907.